Cincom

With Guide Bushing Equipped

Introduction Manual

CITIZEN MACHINERY MIYANO CO., LTD.

(Blank page)

With Guide Bushing Equipped Introduction Manual

0. Table of Contents

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

Code No.	C-CINCOM_IGB 4E1-0000	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

Product Code	C	–	C	I	N	C	O	M	_	I	G	B			
Document Code	4	E	1	–	0	0	0	0							

1. Outline

Code No.	C-CINCOM_IGB 4E1-0100	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

1.1 Preface

This manual describes prerequisites for operating all Cincom products. The manual helps first-time users of Cincom to understand other instruction manuals provided with their Cincom products.

To operate your Cincom product safely, thoroughly read <Chapter 2 Safety Precautions> of this manual and the corresponding sections of all the instruction manuals provided with the Cincom product until fully understanding the contents.

This manual does not describe detailed information for actually operating the machine.

The following are related manuals:

- Basic Manual and Application Manual
- (Operator's Manual, Programmer's Manual, and Maintenance Manual)
- (Instruction Manual, and Maintenance Manual)

The following are the precautions related to this manual:

1. Every effort has been made to ensure the accuracy of all information in this manual. However, the manual may contain incorrect explanation or typographical errors. If you notice any part unclear, incorrect, or omitted in the manual, please contact our company.
2. The contents of this manual may be revised without prior notice.
3. The characteristics, functions, and operations of the machine explained in this manual do not apply to worldwide use. Some illustrations in the manual may not be identical to the actual machine.
4. Citizen Machinery Miyano Co., Ltd. has all copyrights regarding this manual. No part of this document may be reproduced in any form or by any means, electronic, mechanical, or photocopying, without the prior written permission of Citizen Machinery Miyano Co., Ltd.
5. Citizen Machinery Miyano Co., Ltd. owns the copyright for all the software and programs explained in this manual.
6. The company names and product names shown in this manual are trademarks or registered trademarks of the companies.

1.2 Who Should Read This Manual

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

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

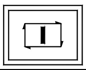
1.3 About This Manual

This manual has the following chapters:

Chapter	Title	Explanation
1.	Outline	This chapter gives guidelines for using the instruction manual.
2.	Safety Precautions	This chapter explains general notes on safety.
3.	Overview of NC Lathe	This chapter explains the overview of the NC lathe. It describes items as prerequisites for operation.
4.	Overview of Operation	This chapter explains the overview of Cincom product operation.
5.	Introduction to Programming	This chapter explains fundamentals of programming for Cincom products that are equipped with a guide bushing.
6.	Program Sample 1	This chapter explains the programming steps using a simple program sample.
7.	Program Sample 2	This chapter explains programming steps for more complex machining processes than processes explained in program sample 1.
8.	Actual Programming	This chapter explains programming for machining processes not explained in program samples 1 and 2, and also the information on advanced programming.
9.	Cutting Conditions	This chapter explains standard cutting conditions.
10.	Terminology	This chapter explains terms for helping first-time users of Cincom products to understand the contents of the manual.

1.4 Notation

The following table explains the symbols used in this manual:

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

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	1	0	0						

2. Safety Precautions

Code No.	C-CINCOM_IGB 4E1-0200	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

2.1 Safety Precautions of Automatic Operation

- To prevent any secondary accident from being triggered by an unexpected incident, such as jammed chips, be sure to monitor the operation status at appropriate intervals during the automatic operation and troubleshoot a failure, if any.
- Accumulation of chips considerably lowers the cooling effect of coolant and can cause fire. To ensure safe operation, it is necessary to select tooling that allows smooth chip flow and clear chips from the machine during operation.
- Confirm that the coolant discharge is enough and that the coolant flow direction is correct.
- Manage tools correctly. Generation of overload due to chipping or break of a tool can cause an unforeseeable accident.

2.2 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.

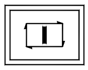
CAUTION; indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate personal injury and/or possible damage to the machine and its components.

2.3 General Precautions During Operation

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

DANGER

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

- **This Cincom is a machine that aimed to cut the workpiece (e.g., metal or resin) with the cutting tool in automatic operation mode. Do not use the machine in any other purpose than cutting.**
- **Do not operate the machine with any cover open.**
- **Do not put your fingers or hands into any moving part of the machine during machine operation. When the** START  **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 any live electrical component of the electrical system.**
  You could be electrocuted if you touch live electrical components.
  All the control unit covers and doors must be closed during machine operation.
  Shutdown the main breaker of the machine before removing the control unit covers and doors.
- **Never change the machine and control circuit.**
- **Press the emergency stop button to 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.

WARNING

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

- **Never disable any safety devices while operating the machine during automatic operation.**
- **Do not open all front left and right doors unless the machine is completely stopped.**
- **Make sure that all front left and right doors are closed and locked (if equipped with locks) and all safety devices are activated before operating the machine.**
- **When operating the machine, sufficiently understand the operation and visually confirm the operation switch to be used before actually pressing that switch.**
- **When machining a material combustible (flammable) during machining by cutting, operate the machine in a state in which the operator can always monitor the machining process.**

CAUTION

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

- **The machine must be properly grounded. The ground must be electrically separated from power lines or the grounding wires of another machine that could be a source of massive electrical noise. See <Relocating the NC Machine> in Maintenance Manual.**
- **Make sure that there is enough coolant in the machine and it is being supplied smoothly to all necessary parts.**
- **Check the tooling to see that it is securely clamped in place before starting the machine.**
- **Be sure to do the periodical checking described in the manual.**
- **Clean the machine regularly to remove any chips and debris from the cutting area and the chip receiving area.**
- **Remove stray chips from the coolant tank as required, depending on cutting condition and type of material being machined.**

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	2	0	0						

3. Overview of NC Lathe

Code No.	C-CINCOM_IGB 4E1-0300	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

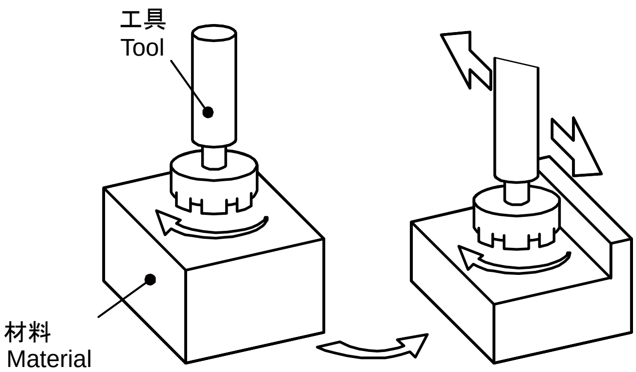
3.1 Machine Tools

A machine tool refers to equipment that produces parts by removing metal from metallic material through cutting. An NC lathe is one of various types machine tools. A machine tool is sometimes called “mother machine” since it is used to manufacture machine tools. Many of industrial products used in daily life incorporate the parts produced by machine tools.

Machine tools are classified into various types according to design, functions, and how a workpiece is machined. From the standpoint how a material is cut with a tool, they are largely classified into two types as explained below.

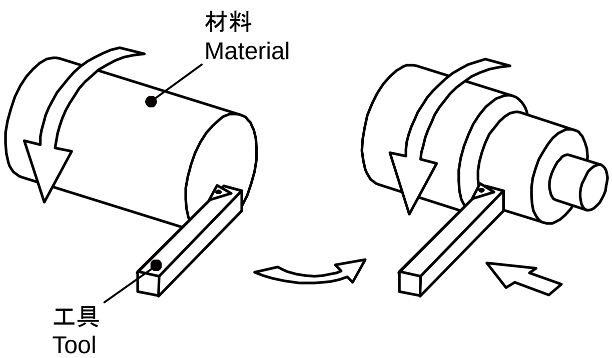
1. A material is fixed and machined with a rotating tool. This type of machine tool is called “milling machine” and it is used to machine rectangular materials. “Machining centers” are developed on the basis of milling machines.

   How a material is machined on a machining center: A fixed material is machined by a rotating tool.



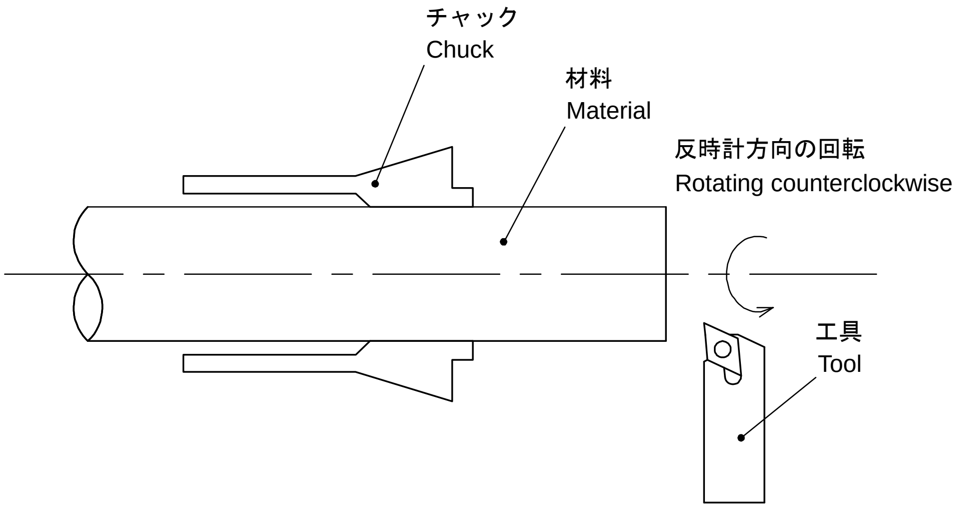
2. A rotating workpiece is machined with a fixed tool. This type of machine tool is called “engine lathe” or simply “lathe” and it is used to machine bar materials. “Turning centers” are developed on the basis of lathes.

   How a material is machined on a lathe: A material is rotated to be machined with a fixed tool.



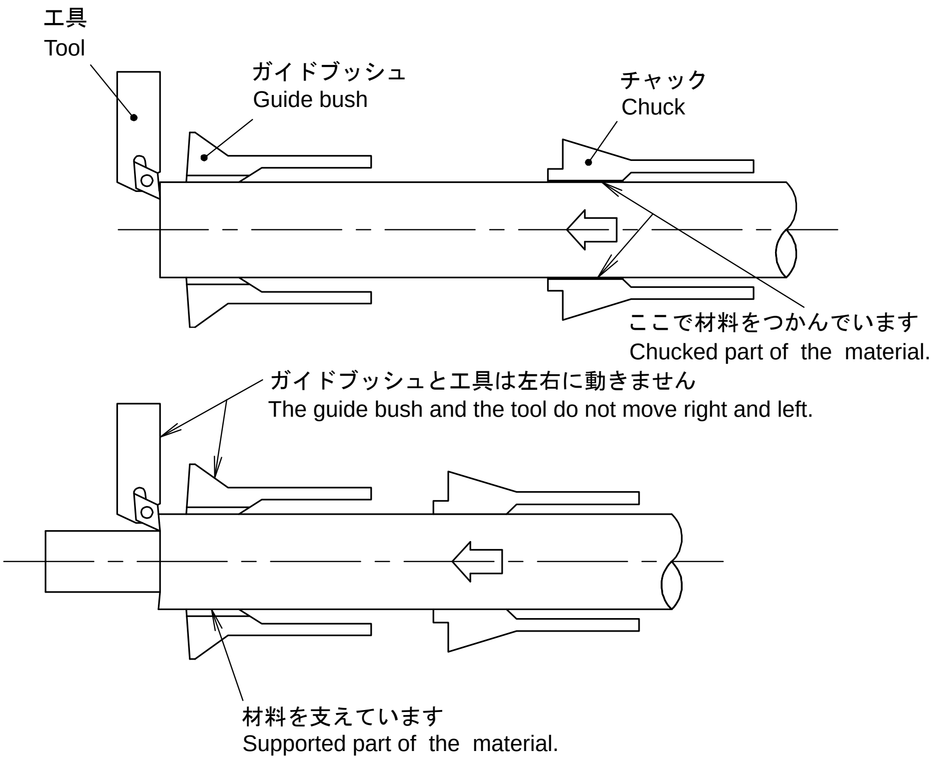
3.2 Lathe

On a lathe, a tool is brought into contact with a rotating material. Basic elements relating to machining are a chuck, a material and a tool. Here, “chuck” is a device that clamps a material to rotate it.



If a point where a tool is brought into contact with a material is apart from the point where the material is chucked, accurate machining is made difficult due to deflection of the material and the force generated during machining. Generally, the material length (L) extending from the chuck is limited to 3 to 3.5 times the material diameter (D); L / D < 3 to 3.5.

To achieve high accuracy when producing a slender part, a device called “guide bushing” that supports the material is installed between a chuck and a tool to allow the chuck to move in the longitudinal direction of the material. This type of machine is called “Swiss-type automatic lathe”. On this type of machine, deflection of the material caused by the force generated during machining is reduced since the guide bushing supports the material near the tool so that high machining accuracy can be achieved. Recently, this Swiss-type automatic lathe is the main stream of compact NC lathes.



3.3 Guide Bushing

Guide bushings are classified into two types by their structures: a guide bushing which does not rotate with the spindle (material) and a guide bushing which rotates with the spindle (material). The former is called a fixed guide bushing; the latter is called a rotary guide bushing.

3.3.1 Fixed guide bushing

A fixed guide bushing is used with thin materials like watch parts in many cases.

A material rotates while it is supported by the inner circumference of the guide busing. Accordingly, the material is constantly in contact with the inner circumference of the guide bushing in both the rotating and longitudinal directions. However, this contact does not usually cause “galling” since oil film exists between the inner circumference of the guide bushing and the material.

However, if surface speed is increased there are cases the oil film loses lubricity to cause “galling”. In comparison with the rotary guide bushing, the fixed guide bushing provides better roundness of produced parts.

3.3.2 Rotary guide bushing

A rotary guide bushing rotates with the material. Thanks to this structure, fractioning in the rotating direction hardly occurs thus galling does not occur even in machining with high spindle speed. In the longitudinal direction, however, galling can occur since longitudinal sliding takes place as with the fixed guide bushing. Rotary guide bushings are classified into two types such as synchronous type and asynchronous type.

- Synchronous type

  A synchronous rotary guide bushing device is linked mechanically or electrically with the spindle. Accordingly, the guide bushing device rotates at the same speed as the spindle.

- Asynchronous type

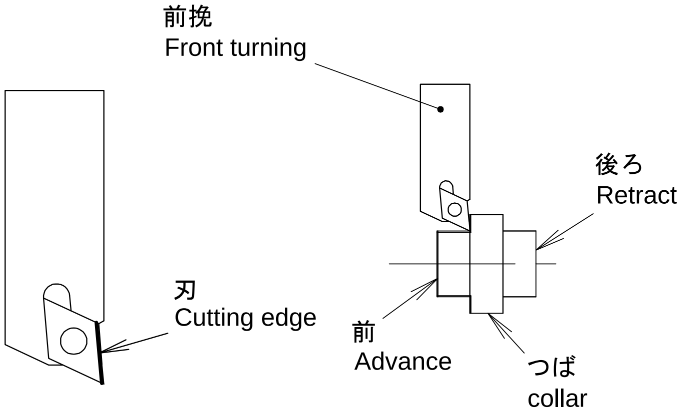
  An asynchronous rotary guide bushing device is not linked with the spindle. Accordingly, the guide bushing device does not always rotate at the same speed as the spindle.

3.4 Tools

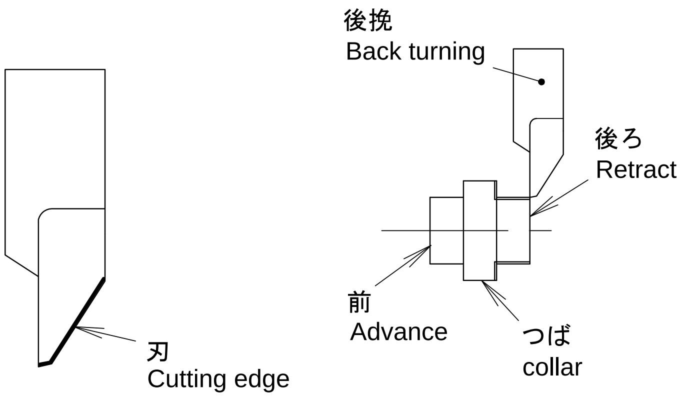
3.4.1 Tool types

There are various types of tools and it is necessary to select the tool meeting the type of machining. The following shows typical types of tools.

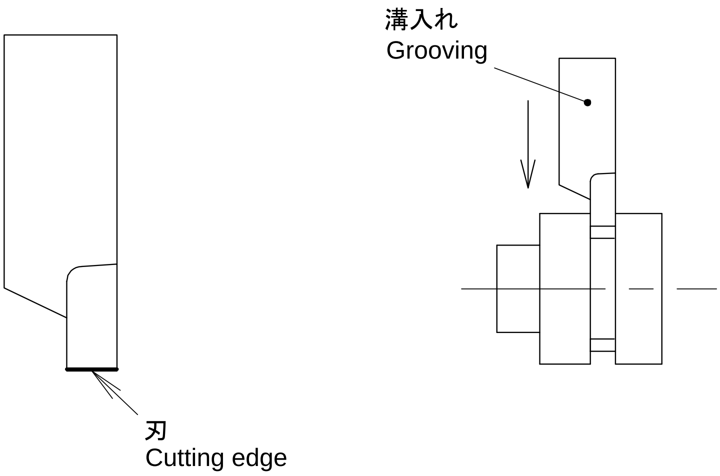
- Front turning tool: Used to machine the outer diameter of a material at the front side part from the collar.



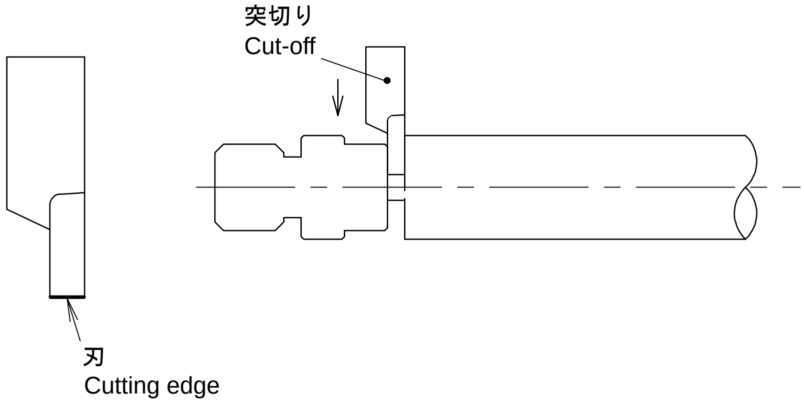
- Back turning tool: Used to machine the outer diameter of a material at the back side part from the collar.



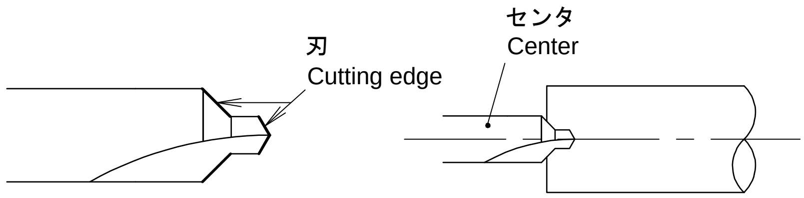
- Grooving tool: Used to machine grooves that cannot be machined with a front or back turning tool.



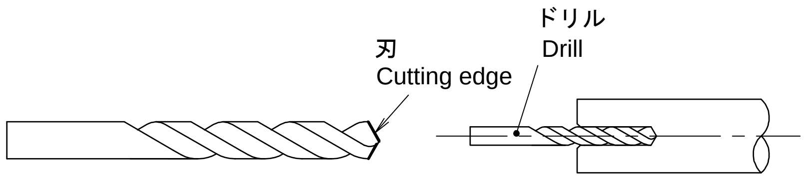
- Cut-off tool: Used to separate a product from the material. A cut-off tool is similar to a grooving tool but has a longer cutting edge so that it can cut the material deeper.



- Centering tool: Used to perform centering operation. Centering is necessary to ensure stable drilling.



- Drill: Used to machine a hole.



3.4.2 Tool materials

There are various types of material for the tools used to machine a workpiece. To produce good products, it is important to select tool materials meeting the workpiece material to be machined.

- Diamond

  A tool formed from single crystal of diamond

- CBN sintered compact

  Poly-crystal sintered compact produced by sintering CBN (cubic boron nitride) by adding cobalt, titanium carbide, etc.

- Ceramics

  Ceramics used for tools is sintered compact of aluminum oxide, titanium carbide, silicon nitride, etc. It has excellent hardness at high temperature and good wear resistant characteristics since no bonding additive is included.

- Cernet

  Sintered alloy manufactured from titanium carbide and titanium nitride in powder metallurgy process by adding nickel, etc.

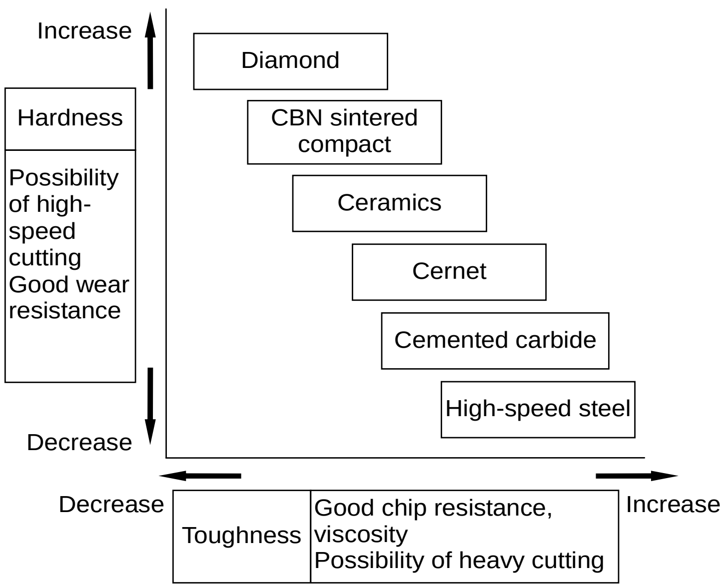
- Cemented carbide

  Sintered alloy manufactured from tungsten carbide powder in powder metallurgy process by adding titanium carbide, tantalum carbide, cobalt, etc. JIS classifies cemented carbide into P, M and K types according to the use. The CIS (Japan Cemented Carbide Tool Manufacturers' Association) standards stipulate additional classifications V and Z.

- High-speed steel

  Iron base alloy tool steel consisting of tungsten, chromium, vanadium, molybdenum and other metallic constituents

Characteristics of typical tool materials

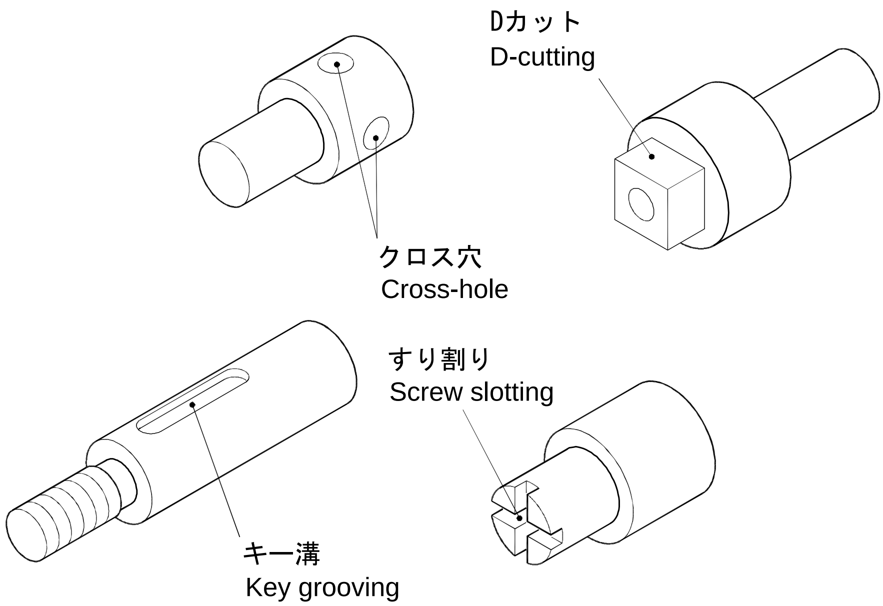


3.5 Turning and Secondary Machining

On a lathe, a tool is brought into contact with a rotating material to produce products by removing metal. This type of machining is generally called “turning”. There is another machining method, in which a material is held and machined with a rotating tool. This type of machining is called “secondary machining”.

Originally, the term “secondary machining” meant machining to be performed in the second process. In the past, machining that uses rotary tools as shown below could not be performed on a lathe and a dedicated machine had to be used to perform secondary machining. Inheriting this fact, the processes to be performed using rotary tools are called “secondary machining” although such processes can now be performed on lathes.

Machining samples



3.6 What is NC?

The term NC stands for Numerical Control. An NC unit controls machine tools by programs (instructions describing operations).

3.7 Program

To operate the NC unit, each operation must be translated into a machine language (instructions).
The machine language forms a program and the translating creates a program (programming).

3.8 What is an Axis Control Group?

The Cincom provides machines of multi-axis (2-axis or 3-axis) control groups. A program for operating a machine, having many axes, in a flow (single axis control group) becomes complicated because many axis names (e.g., X and Z) are necessary. Also, many axes cannot be moved efficiently at a time.

To solve the above problems, the multi-axis control group program, which consists of a single program, divides axes into groups and executes the relevant program for each group at the same time. The group is called an axis control group.

For the two axis control groups, axis control group 1 is expressed as $1 and axis control group 2 is expressed as $2. Similarly, for the three axis control groups expression of the individual axis control groups is: axis control group 1 is expressed as $1, axis control group 2 is expressed as $2, and axis control group 3 is expressed as $3.

$1	$2	$3
G50 Z0 ;	G610 ;	G610 ;
G610 ;	G640 ;	G640 ;
G99 M6 ;	G0 Z–1.0 ;	G0 Z–1.0 ;
G0 X13.0 Z–1.0 ;	X–5.0 ;	!1!2 L1 ;
M3 S1=3000 M23 S2=3000 ;	!1!3 L1 ;	G99 G1 Z4.0 F0.05 T37 ;
T0200 ;	G1 Z4.0 F0.05 T27 ;	G0 Z–1.0 T00 ;
T2700 ;	G0 Z–1.0 T00 ;	!2 L2 ;
G0 X13.0 Z–0.5 T02 ;	!3 L2 ;	G630 ;
G640 ;	T2800 ;	G632 ;
G1 X3.0 F0.2 ;	M23 S2=3200 ;	G650 ;
X5.0 Z0.5 F0.05 ;	G630 ;	G0 Z–10.0 M24 S2=3500 ;
!2!3 L1 ;	G632 ;	G98 G1 Z26.0 F1000 ;
Z10.0 F0.05 ;	G0 X13.0 Z–1.0 T28 ;	G4 U1.0 ;
X6.0 ;	G1 X4.0 F0.2 ;	M15 ;
X7.0 Z10.5 ;	X7.0 Z0.5 F0.05 ;	!1 L3 ;
Z20.0 ;	Z10.0 ;	G610 ;
X11.0 ;	X11.0 ;	M25 ;
X13.0 Z21.0 ;	X13.0 Z11.0 ;	G999 ;
G630 ;	T2000 ;	N999 ;
M5 G98 M80 S3=2500 ;	G0 X20.0 Z–1.0 M25 ;	M2 ;
M28 S0 ;	M16 ;	M99 ;

A program is executed from the top line toward the bottom line in order. Execution of the program advances line by line, in other words, execution of the program shifts to the next line only after the commands in the current line have been completed.

Programs in different axis control groups are usually executed independently. There can be a case that the tenth line in axis control group 2 program is executed although the first line in axis control group 1 program is still executed.

(Blank page)

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	3	0	0						

4. Overview of Operation

(Blank page)

4.1 Operation Flow

4.1.1 Procedure from power-on to continuous operation

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

1. Empty the chip tank if it contains chips.
2. Set the main breaker to ON.
3. Press the Power ON switch on the operation panel.

   The screen appears after a short time.
4. Select the program for the workpiece to be machined.
5. To prevent interference with the tool, move the material back to the end face of the spindle.
6. Open the spindle chuck.
7. Close the safety cover.
8. Execute Start Position in the Preparation mode.

   Each axis moves to the start position for the program.
9. Slowly push the material against the cut-off tool by hand.
10. Operate the bar loader to apply torque to the push rod.
11. Select the Automatic Operation mode.
12. Select 1 Cycle.
13. Supply coolant.

    Confirm that the coolant discharge is enough and that the discharge position is correct.
14. Press the Start key.

    The machine performs 1-cycle machining in the Automatic Operation mode.
15. Measure the workpiece diameter and the longitudinal dimension, and set offset values and change the program to correct the product dimensions.
16. Select Continuous Cycle in the Automatic Operation mode.
17. Press the Start key.

    The machine performs continuous machining in the Automatic Operation mode.

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

4.1.2 Procedure from continuous operation to power-off.

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

1. Select 1 Cycle in the Automatic Operation mode.
2. Wait until the Start key lamp goes off.
3. Stop coolant.

   (4. Stop the spindle.)

5. Press the Power OFF switch on the operation panel.

   The screen display disappears after a short time.

6. Set the main breaker to OFF.

4.1.3 Re-arrangement procedure

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

1. On the Edit screen, enter the program and machining data for the workpieces to be machined next.

   If the program to be used next is already prepared, load the program in the machine.

2. Dismount the tools used to machine the previous workpieces. If you use the same tools, you do not have to dismount them.

   When you set a tool in a machine equipped with the back machining function, be sure to dismount the back machining tool to prevent interference.

3. Replace/adjust the chuck and the guide bushing.

   If the machine is equipped with the back machining function, replace the knock-out jig, and replace/adjust the back spindle chuck.

4. Select the program for the workpieces to be machined in the Preparation mode (the Program Selection mode depending on the machine). Confirm that the selected machining data is correct.

5. Execute Start Position in the Preparation mode.

   Each axis moves to the start position specified in the program.

6. Set offset values on the Offset screen.

   Generally, clear all the offset setting values of the dismounted tool to 0.000.

   If the tool nose radius compensation function is used, set the tool nose data.

7. In the Program Check mode or Automatic Operation mode, run the machine with no material inserted (dry run) to check if the machine operates normally.

   If an alarm is issued or a program error is detected, correct the program.

8. Set the turret, gang, and front 3-spindle tools.

   Use the Preparation mode to set the tools.

   Set the back 3-spindle tools in step 14.

9. In the Program Check mode or Automatic Operation mode, run the machine with no load to check if the tools move as expected.

   Correct the program if necessary.

10. Select 1 Cycle in the Program Check mode or Automatic Operation mode to machine a workpiece actually.

    Since no offset values have been set at this stage, the back spindle and the product may interfere with each other. To prevent the interference, perform cut-off machining without the product being chucked by the back spindle.

11. Measure the workpiece diameter and the longitudinal dimension, and set offset values or change the program to correct the product dimensions.

12. When the back spindle did not chuck the product in Step 10, get the back spindle to chuck it after the front machining dimension is corrected in step 11, and change the program to perform cut-off machining.

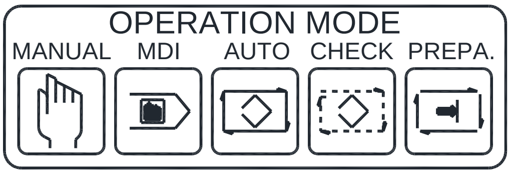
13. Select 1 Cycle in the Automatic Operation mode to machine the workpiece.

14. Set the back machining tools.
15. Select 1 Cycle in the Automatic Operation mode to machine the workpiece.
16. Measure and correct the dimensions (including the back machining dimension) of all the workpieces.
17. Check the product count set in the product counter and set a new value.
18. Select Continuous Cycle in the Automatic Operation mode to continuously machine workpieces.

4.2 Operation Modes

Functions to operate the machine are classified by purpose. They are defined as operation modes.

To select an operation mode, press the corresponding button in the Operation mode field on the operation panel. The following shows the operation modes:



PREPARATION
The preparation mode prepares to execute program operation (e.g., automatic operation or program check). This mode performs tool setting and the start position operation (program start position) that automatically moves each axis to the specified position.

PROGRAM CHECK
The program check mode runs the machine actually to check the program. Like the automatic operation mode, this mode permits you to correct the program during operation and to turn back program operation.

AUTO
The automatic operation mode runs the machine according to the program stored in memory. You can select 1 Block operation, 1 Cycle operation, or Continuous Cycle operation on the mode.

MDI
The MDI mode permits you to execute the program with commands issued by pressing keys on the operation panel. This mode operates the machine according to the commands.

MANUAL
The manual mode permits you to move each axis with the handle or switches on the operation panel.

Product Code	C	–	C	I	N	C	O	M	_	I	G	B	
Document Code	4	E	1	–	0	4	0	0					

5. Introduction to Programming

Code No.	C-CINCOM_IGB 4E1-0500	MFG No.	CINCOM	Issue Date	2012.6

5.1 What is an NC Program?

An NC program consists of a series of commands required for operating the machine under numeric control (NC).

Each command is expressed in a simple combination of alphanumeric characters.

This section explains the concept of NC programs, showing an example of creating a program.

This example uses the following four commands to create command statements for moving from A to B in the figure.

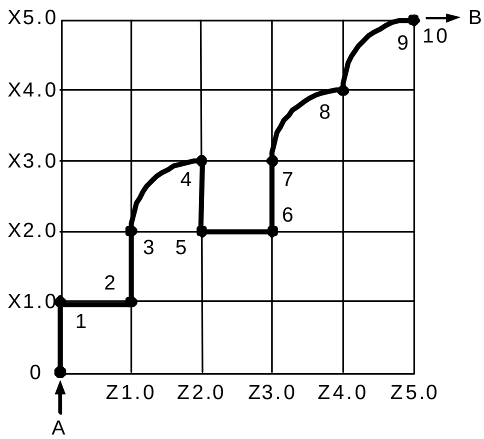
G01..Move straight to ○○.

G02..Move circularly to ○○.

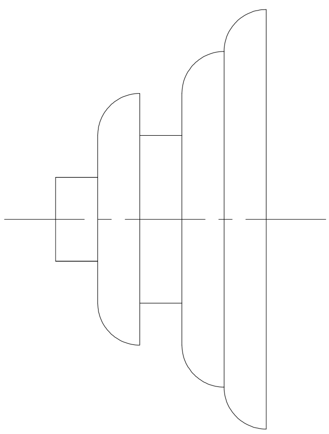
M03..Turn the Start switch ON.

M02..Turn the Goal switch OFF.

0	M03			(Start switch ON)
1	G01	X1.0	Z0	(Straight move to the position where X = 1.0 and Z = 0)
2	G01	X1.0	Z1.0	(Straight move to the position where X = 1.0 and Z = 1.0)
3	G01	X2.0	Z1.0	(Straight move to the position where X = 2.0 and Z = 1.0)
4	G02	X3.0	Z2.0	(Straight move to the position where X = 3.0 and Z = 2.0)
5	G01	X2.0	Z2.0	(Straight move to the position where X = 2.0 and Z = 2.0)
6	G01	X2.0	Z3.0	(Straight move to the position where X = 2.0 and Z = 3.0)
7	G01	X3.0	Z3.0	(Straight move to the position where X = 3.0 and Z = 3.0)
8	G02	X4.0	Z4.0	(Circular move to the position where X = 4.0 and Z = 4.0)
9	G02	X5.0	Z5.0	(Circular move to the position where X = 5.0 and Z = 5.0)
10	M02			



The above command statements are equivalent to NC programs. When they are executed on a lathe, the product shown below is produced.



5.2 Procedure for Creating an NC Program

The following is the flowchart of programming.

1. Drawing the workpiece

   The NC program is created based on the drawing.
   Confirm that all the dimensions are written in the drawing.

2. Creating the tool layout

   Select the tools to be used.
   Determine the sequence of machining.
   (Selection of the machine to be used)
   Specify tool shapes roughly, groove widths and tool numbers.

3. Creating the machining layout

   Describe the cutting path of each tool and the sequence of machining.

4. Setting the cutting conditions

   Select suitable cutting conditions.

5. Reading the coordinate values

   Read the coordinate values from the cutting path of each tool.
   Infeed amount (X) and longitudinal feed amount (Z)

6. Creating the program

   Describe the tool to be used, the cutting distance, and the machining conditions on a sheet of paper according to the programming rules.

7. Input the program

   Input the created program into the NC unit.

8. Input the machining data

   Input the machining data such as the names of the holders to be used, the material diameter, and the machining length in advance.

9. Input the tool set data

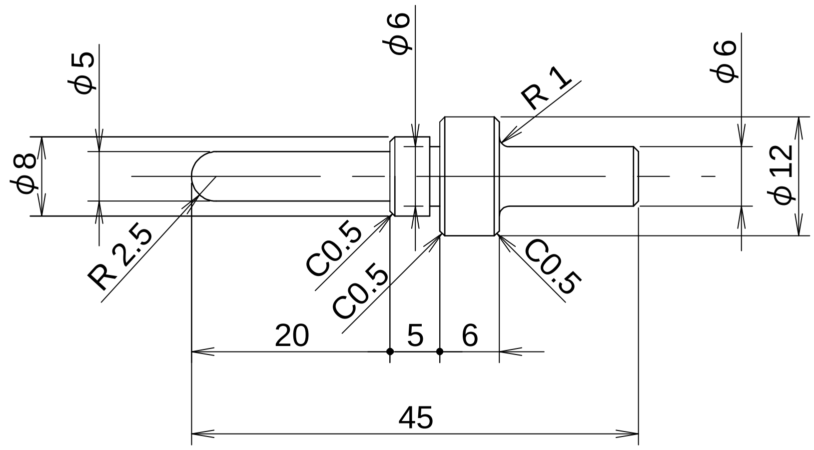
   Input data such as the core position, the diametrical direction, and the longitudinal direction in advance.

10. End

5.2.1 Drawing

To create an NC program, prepare the drawing in which all the dimensions are written.

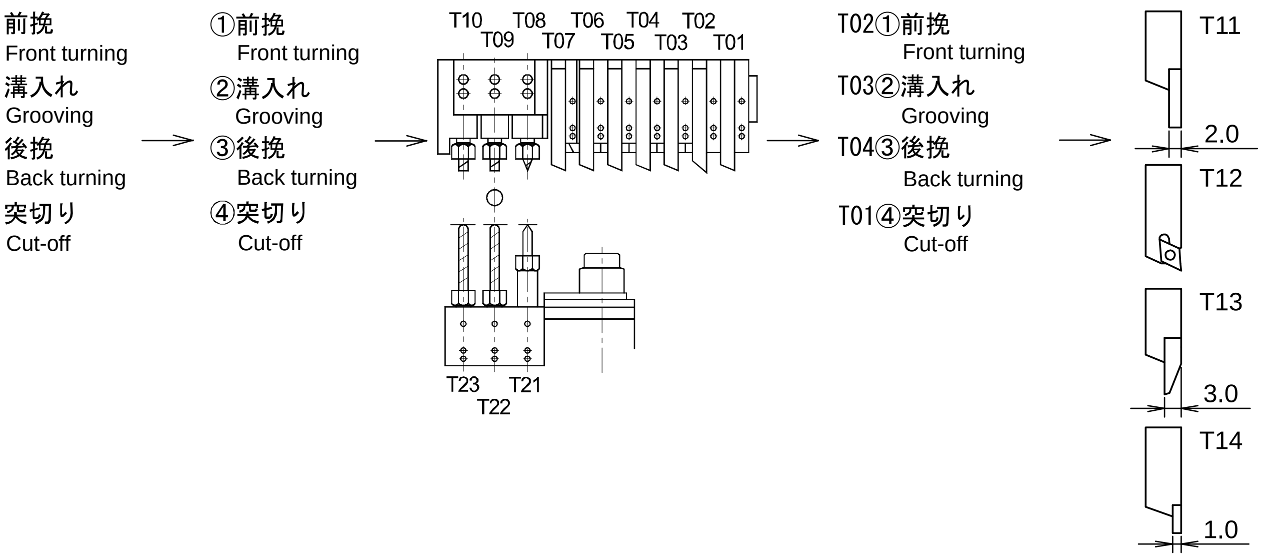
Material: Brass



5.2.2 Creating the tool layout

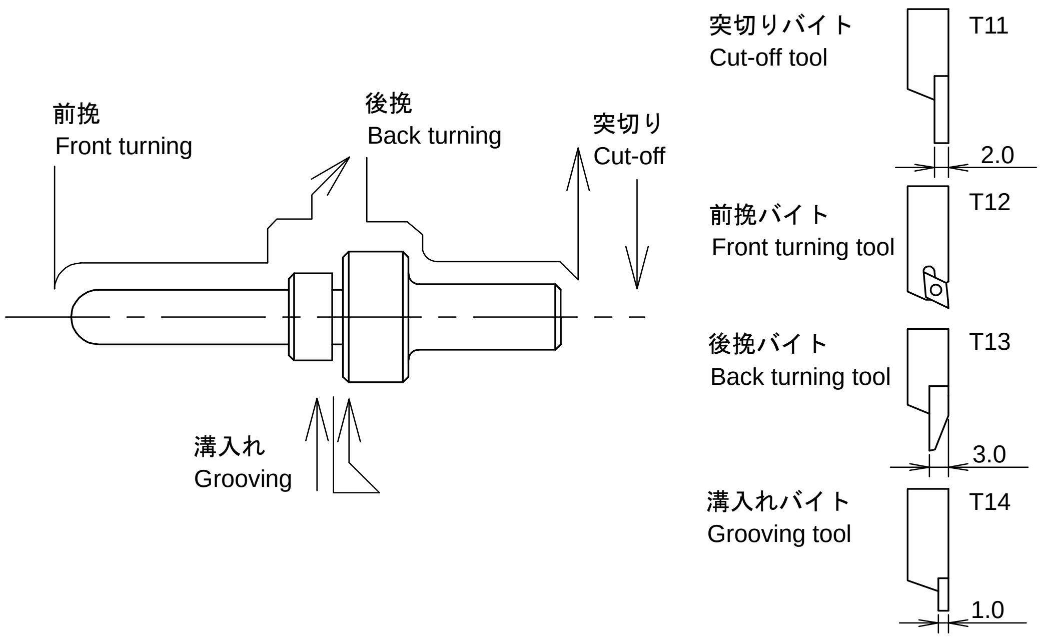
First, select the tools to be used for machining the workpiece and determine the machining sequence. Next, select the machine to be used and then determine the tool mounting positions meeting the selected machine. Finally, determine the tool shapes. Keep in mind that these items to be determined and the steps may not be always followed as explained above when you obtain knowledge on programming.

Tool selection → Machining sequence → Machine selection → Tool mounting position → Tool shape



5.2.3 Creating the machining layout

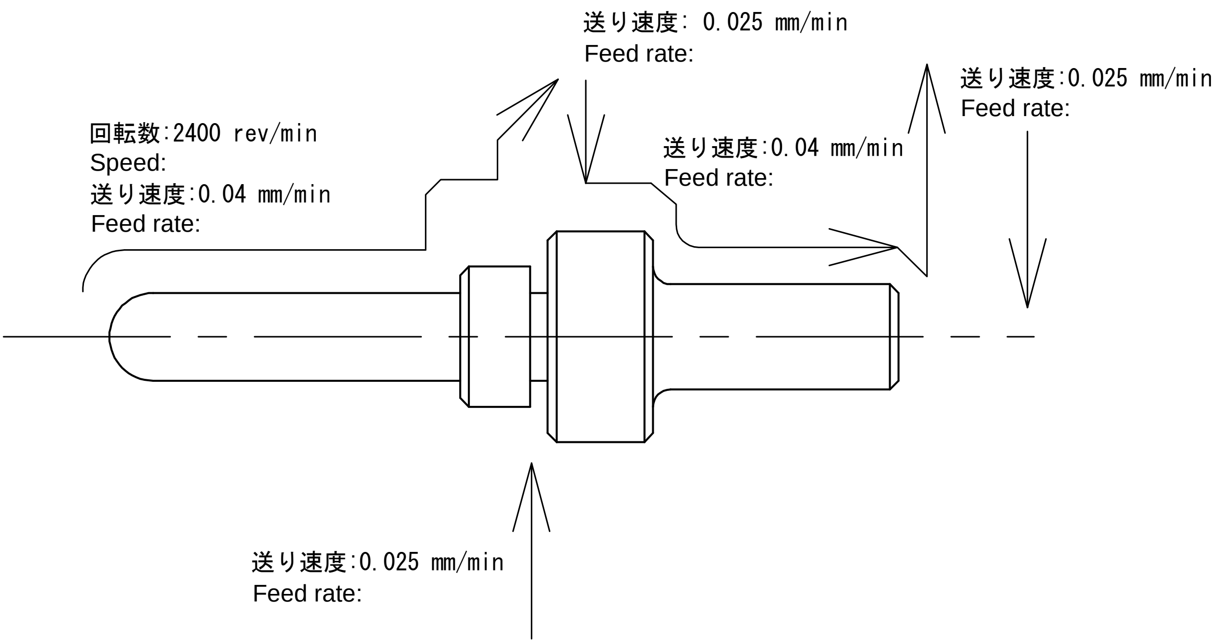
On the machining layout drawing, describe the machining sequence and the cutting path to be generated by the tool nose, and enter the tool number of the tools to be used.



5.2.4 Setting the cutting conditions

Cutting conditions mean the spindle speed and the feed rate.

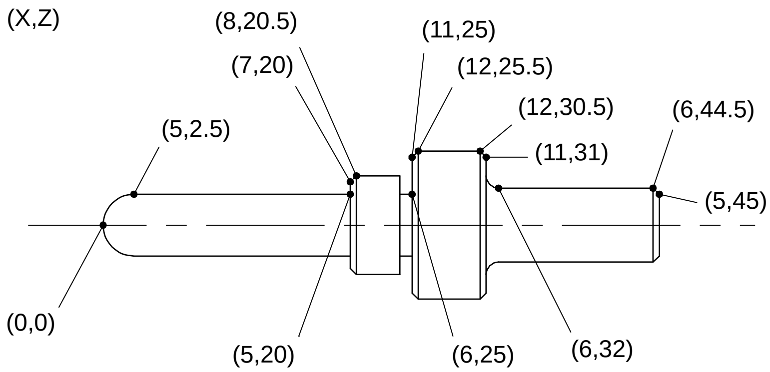
Set the cutting conditions in consideration of the element of the material, the outer diameter of the material, the types of the tools to be used, and the infeed amounts.



5.2.5 Reading the coordinate values

Read inflection points (the points at which the tool move direction changes) of the tool nose path correctly on the basis of the part drawing and the machining layout.

X coordinates specify diameters. Therefore, an X coordinate value must be doubled when it is read.



5.2.6 Creating the program

Create the program with the commands for machine operation (e.g., tool selection and the spindle speed) and the read coordinate values.

5.2.7 Inputting the machining data

The machining data is necessary for automatic operation and preparation for operation. Be sure to set the machining data (e.g., the material diameter and machining length) for each program.

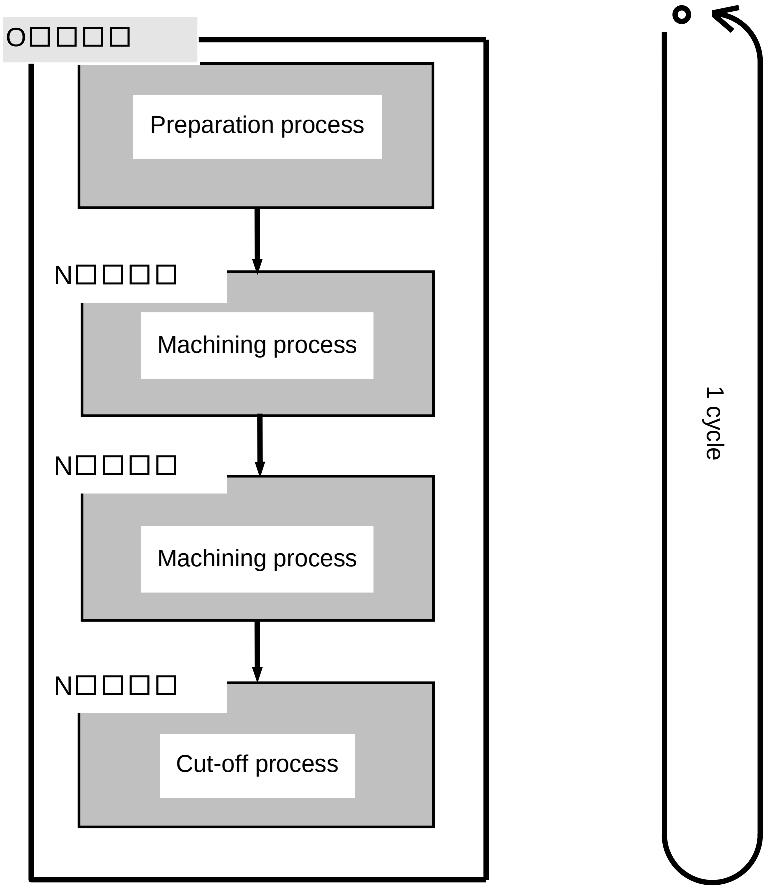
Be careful to set the machining data correctly before running the machine. The machine fails to operate correctly if wrong machining data is set.

5.2.8 Input the tool set data

When tools are to be set in the machine (setting inside the machine), the adjusted tool positions are set as tool set data.

When tools are to be set with the tool presetter (setting outside the machine), there is no need to set tool set data.

5.3 Program Structure



A program begins with the preparation process and ends with the cut-off process. Machining processes such as front turning and grooving are described between the preparation process and the cut-off process. A program must be created so that it ends at the same point as the program start point. A series of program, from the start to the end, is called one cycle.

Address O specifies a program number.

Specify address O in the format of O□□□□.

Usable program numbers vary depending on the machine type. See the instruction manual for the machine to be used to confirm them. A drawing number is often used as a program number. Since a new program is created after specifying a program number, it is not necessary to input one on the Edit screen.

Address N specifies a sequence number.

Specify address N in the format of N□□□□□. A numeric value of up to five digits can be specified.

The NC unit does not perform any operation with a sequence number. Sequence numbers are used simply as programmers' memorandums.

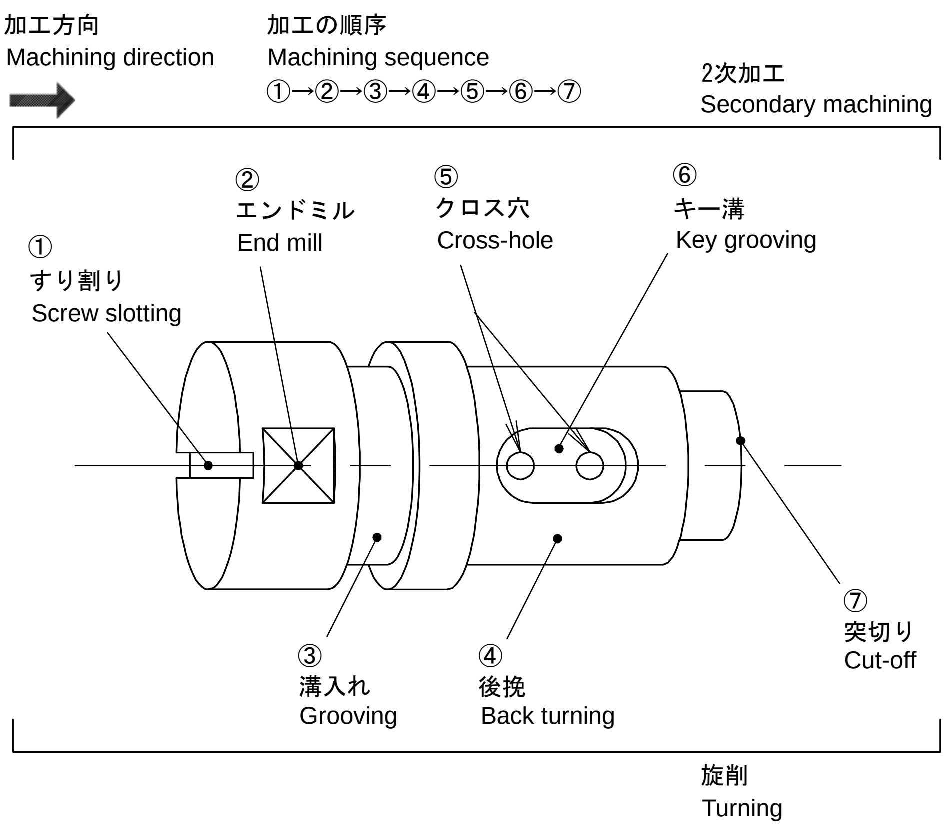
For example, when address N is specified in the format of N + a numeric value of four digits, define the first two digits as a process sequence and the last two digits as a tool number.

5.4 Machining Sequence

Workpieces that require secondary machining are finished in a single program like workpieces that can be finished in turning operations only. Generally, the machining sequence is determined according to the rules explained below.

Machining sequence rules

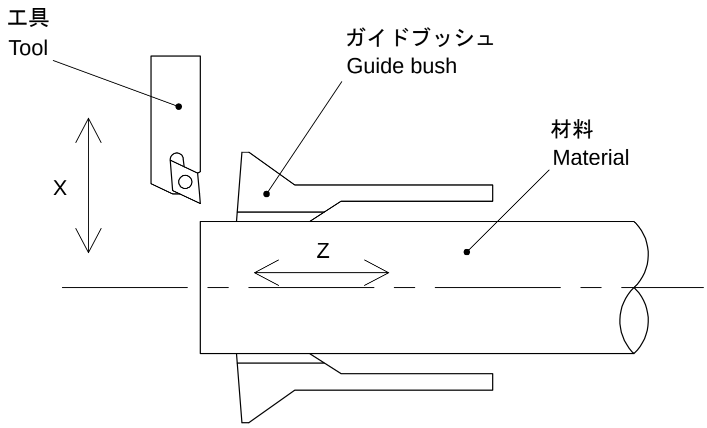
- Set a workpiece in the manner the centerline of rotation is set horizontally and, in this state, proceed machining from the left to the right. Machining in the reversed movement (e.g. from the right to the left) should be avoided as long as practical.
- First carry out the hole machining process at the center of rotation and the secondary machining process on the end face of a workpiece.
- If the turning process and the secondary machining process are required for the same place, carry out the secondary machining process after the turning process.



5.5 Coordinate System

5.5.1 Coordinate axes

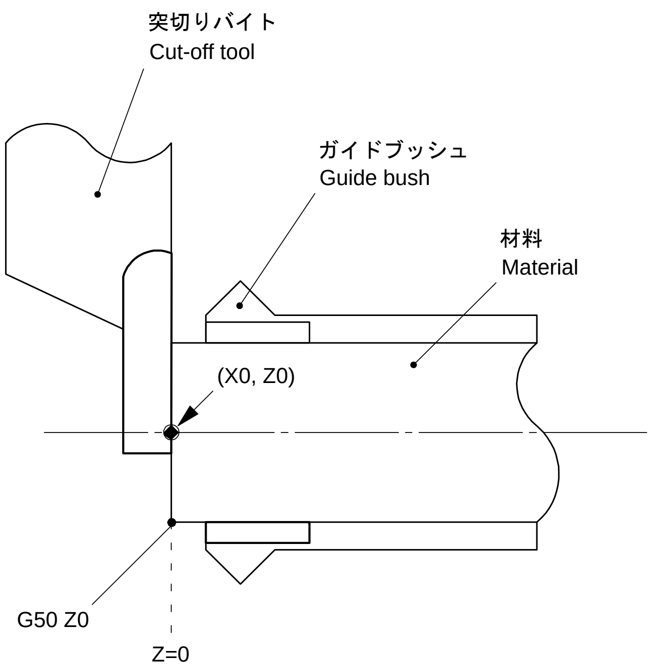
The axis along the center axis of material rotation is determined as the Z axis and the axis perpendicular to the Z axis as the X axis. In other words, the Z axis refers to the direction the material moves longitudinally and the X axis the direction in which an outer diameter turning tool moves into the material. The Z and X axes are sometimes called the longitudinal axis and diametric axis, respectively.



5.5.2 Setting the coordinate system (program zero point)

For the X and Z axes defined as explained above, set the zero point of them. For the Z axis (longitudinal direction), set the position of the material end face that is in contact with the cut-off tool as the zero point (Z = 0). This setting is possible by specifying “G50 Z0” in a program. This command means “Define the current Z axis position as Z = 0.”

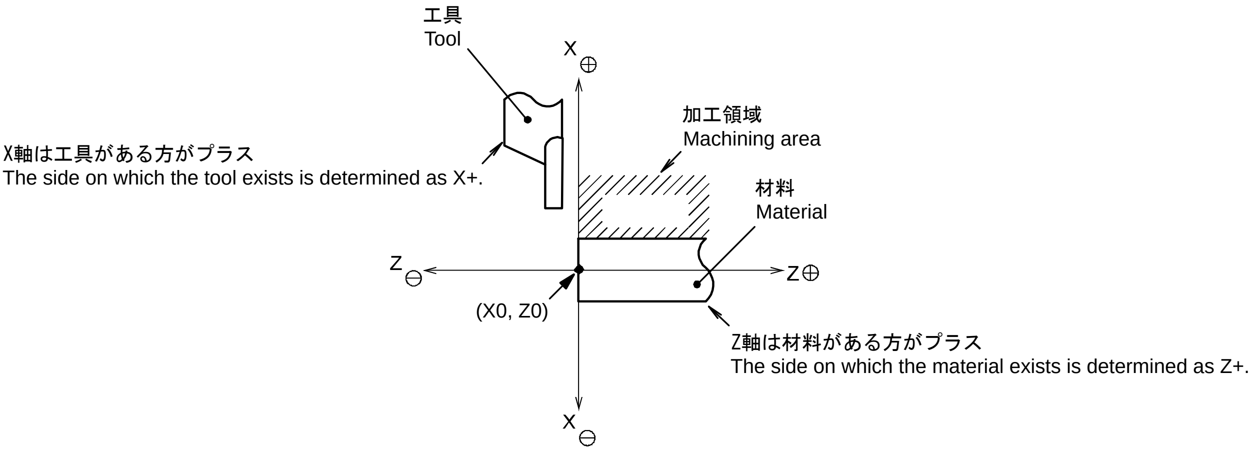
For the X axis (diametric direction), set the center of material rotation as the zero point (X = 0). For this setting, it is not necessary to specify the corresponding commands in a program because the tool setting is made so that tool nose comes to the zero point (X = 0) when a tool is mounted. Now, the center of rotation at the material end face takes the coordinate values (X0, Z0) and this point is called the program zero point.



5.5.3 Signs

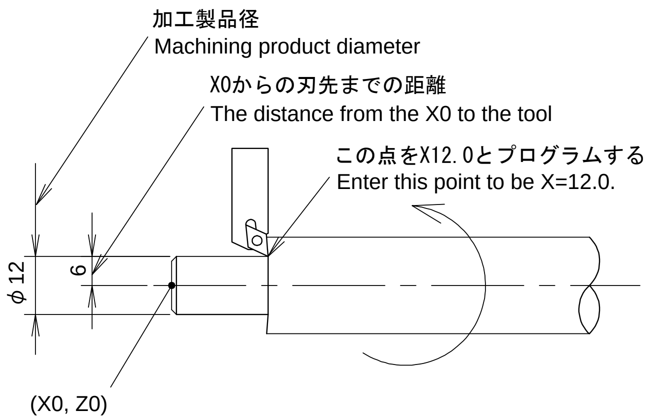
In the coordinate system, location of a point is defined by a coordinate value and a plus/minus sign. In other words, a sign indicates at which side a point in question lies in reference to the zero point; for the X axis, the side on which a tool exists is determined as X+, and for the Z axis, the side at which the material exists is determined as Z+.

Usually, machining is carried in the X+ and Z+ area.



5.5.4 Coordinate values

Along the Z axis, a coordinate value indicates the distance between a point in question and the end face of a material (Z = 0). Along the X axis, it indicates a value two times the distance between a point in question and the center of rotation of a material (X = 0). The reason for this is a material is rotated around the centerline and thus the diameter of a product to be machined is two times the distance a point lies from the center of rotation. Since a coordinate value to be specified for the X axis is two times the distance, an X axis command is called a diametric command. For a numeric value, a value in “mm” unit with a decimal point is used. Note that a decimal point must always be specified in a coordinate value.



5.5.5 Absolute and incremental commands

The move commands are classified into absolute commands (for absolute positions) and incremental commands (for relative positions).

Absolute commands

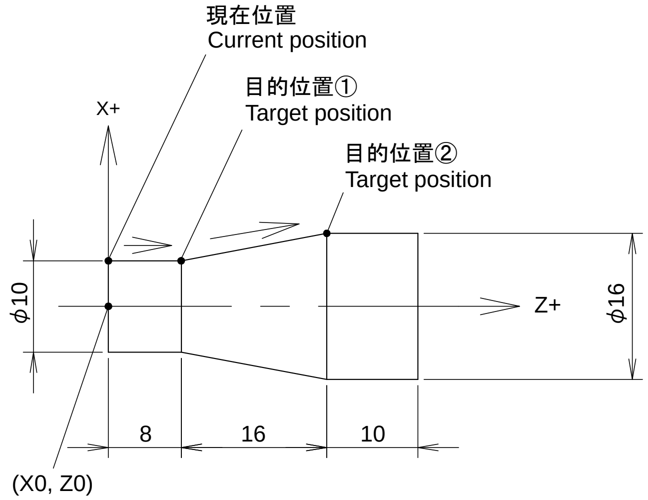
An absolute command specifies the distance from the program zero point.

Specify an absolute command in the format of X□, Z□.

Incremental commands

An incremental command is called a relative position command, which specifies the distance from the current position to the target position.

Specify an incremental command in the format of U□, W□.



Take the center of rotation on the material end face as the program zero point (X0, Z0).

Absolute		Incremental	
Current position	X10.0 Z0	**Current position**	U0 W0
	↓		↓
Target position (1)	X10.0 Z8.0	Target position (1)	U0 W8.0
	↓		↓
Target position (2)	X16.0 Z24.0	Target position (2)	U6.0 W16.0

In actual programming, specification of an axis that does move is omitted.

Absolute		Incremental	
Current position	X10.0 Z0	**Current position**	U0 W0
↓		↓	
Target position (1)	Z8.0 **(X10.0 is omitted)**	Target position (1)	W8.0 **(U0 is omitted)**
↓		↓	
Target position (2)	X16.0 Z24.0	Target position (2)	U6.0 W16.0

5.6 Codes

An expression consisting of address G, M or T and numeric characters (1 to 4 digits) is called a code. Many of codes have an argument, which is also expressed by the combination of an alphabetic character and numeric characters. Usable arguments vary depending on codes.

G code: Specifies the axis control mode for the X, Z, and other axes.

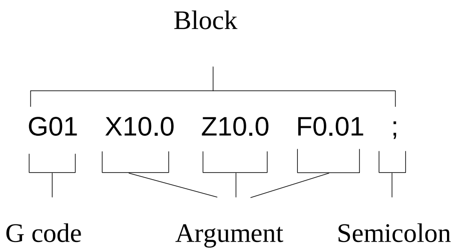
M code: Specifies an auxiliary command other than an axis control command.

T code: Specifies a tool selection command and reading of offset values, etc.

5.6.1 Block

A group of commands delimited by a semicolon is taken as one block, which consists of codes and the relating arguments.

It is possible to specify more than one code in a block. However, since there are combinations of codes that must not be specified simultaneously in the same block, it is recommended to specify only one code in a block until you get fully acquainted with programming.



A semicolon is also called EOB (end of block).

Semicolons (;) are omitted from the ends of lines in this manual.

5.7 G Codes

A G code specifies the axis control mode.

Specify address G in the format of G☐☐. The following table lists frequently used G codes:

Code	Function	Group	Remarks	Refer to
G00	Rapid feed positioning	A	○	5.7.1
G01	Linear interpolation	A		5.7.2
G02	Circular interpolation (clockwise)	A		5.7.3
G03	Circular interpolation (counterclockwise)	A		5.7.4
G04	Dwell		*	5.7.5
G25	Spindle speed change detection OFF (FANUC only)			8.8
G26	Spindle speed change detection ON (FANUC only)			8.8
G32	Thread cutting (tapping)	A		8.1
G50	Setting coordinate systems		*	5.5.2
G92	Thread cutting canned cycle	A		8.4
G96	Constant surface speed control ON	B		8.9
G97	Constant surface speed control OFF	B	○	8.9
G98	Specification of feed per minute (mm/min)	C		5.7.6
G99	Specification of feed per rotation (mm/rev)	C	○	5.7.6

- The G codes for which ○ symbol is entered in the Remarks column are initially set when the power is turned on.
- The G codes for which * symbol is not entered in the Remembers column are called modal G codes. Once a modal G code is specified, the called function remains valid until another G code in the same group is specified.
- More than one G code can be specified in a block as long as they belong to different groups.

5.7.1 G00 Rapid feed positioning

In the G00 mode, positioning is executed from the current position to the specified position at the maximum feedrate of the machine (rapid feed).

This mode is mainly used for moving a tool to the approach point (a point just ahead the point where machining starts).

Command format

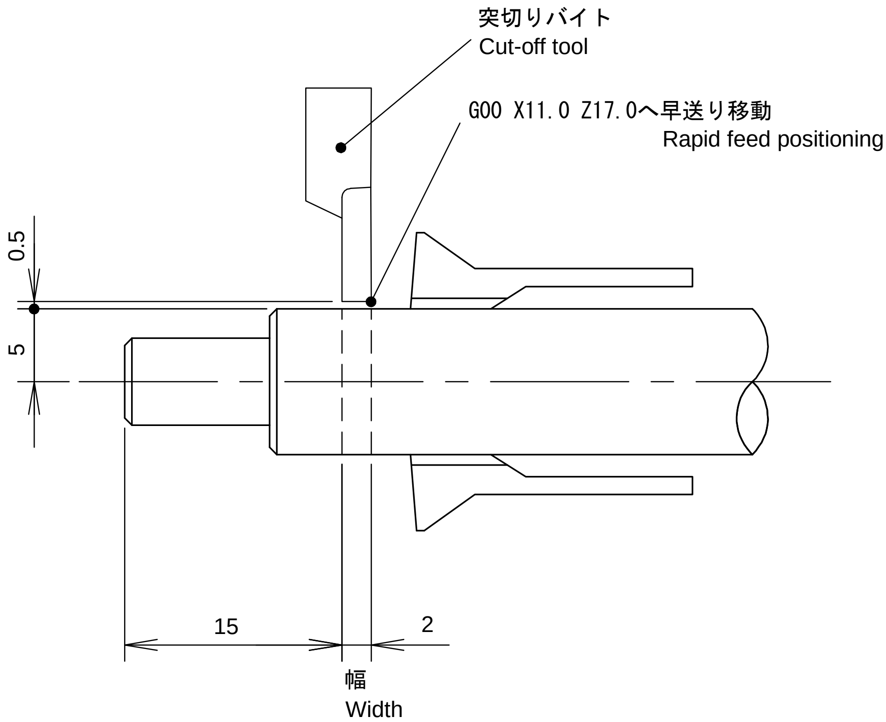
G00 X☐ .... Rapid feed in the diameter (X axis) direction
The Z axis does not move.

G00 Z☐ .... Rapid feed in the longitudinal (Z axis) direction
The X axis does not move.

G00 X☐ Z☐ .... Simultaneous rapid feed in the diameter (X axis) direction and the longitudinal (Z axis) direction
Both X and Z axes move.

Program

G00 X11.0 Z17.0.......................... Rapid feed positioning



In the cut-off machining, the width equivalent to the cut-off tool width is removed from the material. Accordingly, to machine a product to the desired length the cut-off tool width must be taken into consideration when specifying a Z coordinate value for cut-off machining. To machine a 15 mm long product, it is necessary to add cut-off tool width 2 mm to the desired product length; 15 mm + 2 mm = 17 mm. Specify the calculated value as the Z coordinate value for executing cut-off machining, e.g. specify Z17.0.

5.7.2 G01 Cutting feed (linear interpolation)

This mode is used to carry out cutting from the current position to the specified position along a straight line at the specified feed rate. This mode of control is called linear interpolation. An F$\boxed{}$ command is used to specify the feed rate. An F$\boxed{}$ command is a modal command and the specified value remains valid until another F$\boxed{}$ command is specified.

$\boxed{\text{Command format}}$

G01 X$\boxed{}$ F$\boxed{}$.....................Feed in the diameter (X axis) direction
The Z axis does not move.

G01 Z$\boxed{}$ F$\boxed{}$.....................Feed in the longitudinal (Z axis) direction
The X axis does not move.

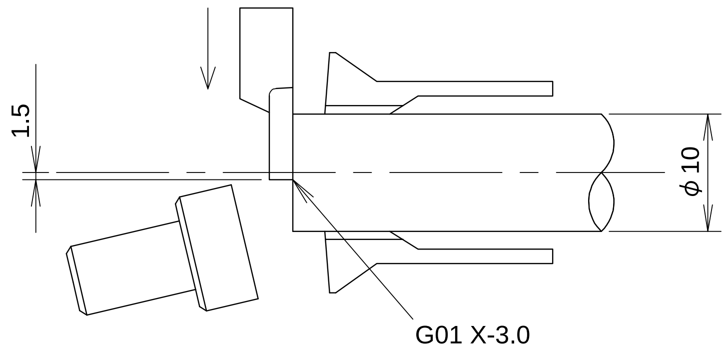
G01 X$\boxed{}$ Z$\boxed{}$ F$\boxed{}$.......Simultaneous feed in the diameter (X axis) direction and the longitudinal (Z axis) direction
Both X and Z axes move.

$\boxed{\text{Program 1}}$

The following program is an example for carrying out cut-off machining with a cut-off tool. In the cut-off machining, the tool moves toward the center of a workpiece.

Some cut-off tools have slanted tool nose to minimize a dowel left at the center of the cut-off face of a workpiece.

G00 X11.0 Z17.0..........................Rapid feed positioning



G01 X–3.0 F0.03..........................Cutting up to X–3.0 in the linear interpolation mode at the feed rate F0.03.

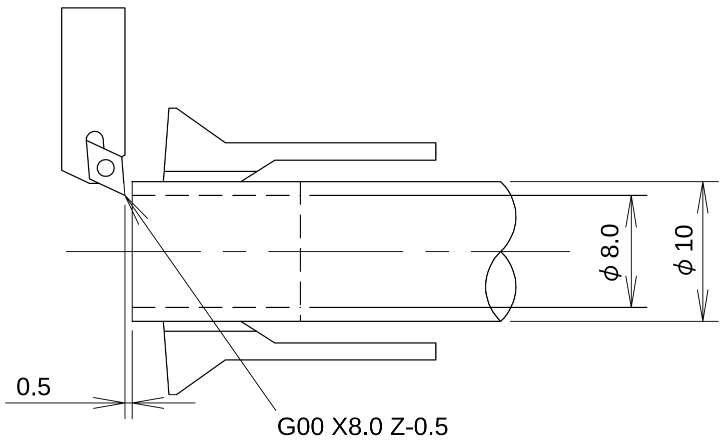
A pyramid-shaped dowel will be left on the workpiece end face if the tool having the slanted tool nose is fed only to X0. The tool is fed up to X–3.0 to so that a dowel will not be left.

Program 2

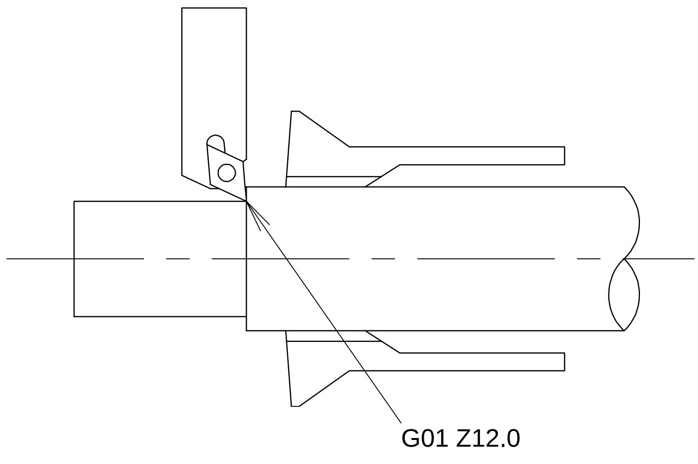
This sample shows cylindrical machining by moving the tool in only the longitudinal direction.

In this sample, machining starts at Z–0.5 along the Z axis. Generally, a clearance of **about 0.5 mm** should be secured when positioning a tool to the start position of machining at a rapid feed rate. The position where a tool is positioned before starting machining is called **approach point**.

G00 X8.0 Z–0.5 Rapid feed positioning



G01 Z12.0 F0.03..........................Cutting up to Z12.0 in the linear interpolation mode at the feed rate F0.03.

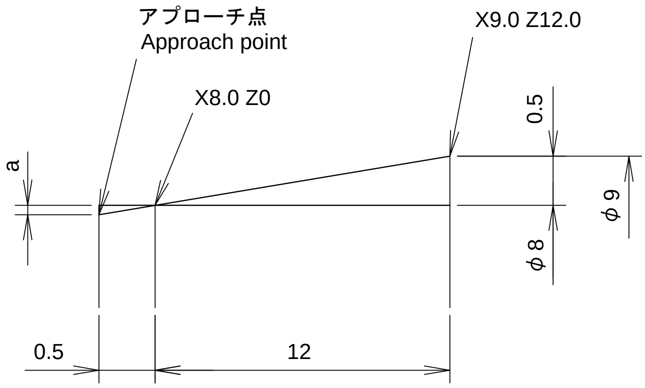


G01 X11.0 F0.1............................Moving the tool back to the **positioning point**, the position ø1.0 mm + material outer diameter.

Program 3

This sample shows a machining that requires simultaneous move in the diameter direction and the longitudinal direction.

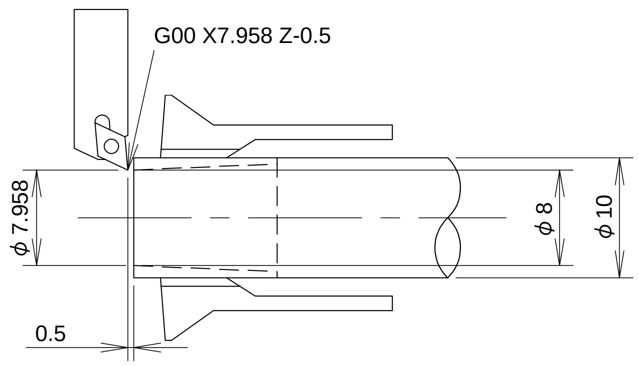
In this sample program, a taper expanding from ø8 mm to ø9 mm is machined. To move a tool from the approach point straight to the end point of the taper, specify the approach point as X7.958 Z–0.5. The X coordinate value of the approach point is calculated as explained below so that the X coordinate value will be X8.0 when the tool moves along the tapered path to Z0.



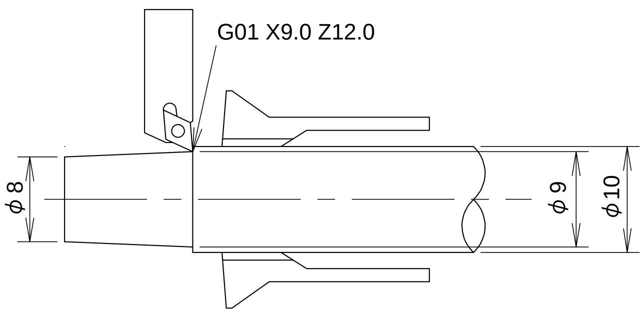
From the figure above, it is found that 12 : 0.5 = 0.5 : a. Since the product of inner terms equals the product of outer terms, the expression 12 × a = 0.5 × 0.5 is derived. From this, value a is calculated as a = 0.25/12 = 0.020833.. Since value a is a radial value, it must be converted to a diametric value and then subtracted from X8. 0. 8 – 0.02083 × 2 ≈ 7.958.

Therefore, the X coordinate of the approach point is found as X7.958.

`G00 X7.958 Z–0.5`........................Rapid feed positioning



`G01 X9.0 Z12.0 F0.03`................Cutting up to X9.0, Z12.0 in the linear interpolation mode at the feed rate F0.03.



`G01 X11.0 F0.03`..........................Moving the tool back to the **positioning point**, the position ø1.0 mm + material outer diameter.

5.7.3 G02 Circular interpolation (clockwise)

This mode is used to carry out machining from the current position to the specified position along an arc of radius R at the specified feed rate. This mode of control is called circular interpolation. Circular interpolation **G02** produces an arc in convex form.

It is not necessary to specify the center of the circle.

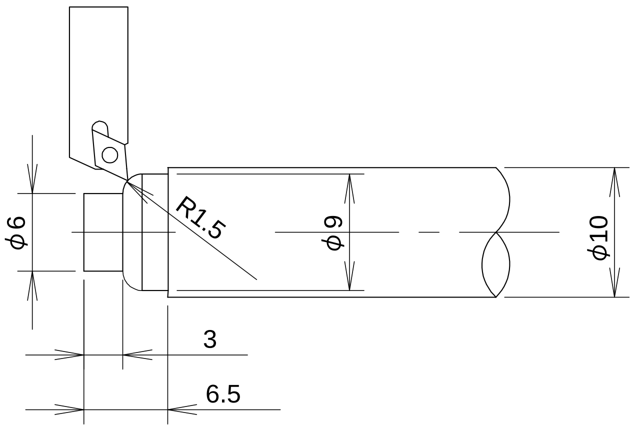
An F☐ command is used to specify the feed rate. An F☐ command is a modal command and the specified value remains valid until another F☐ command is specified.

Command format

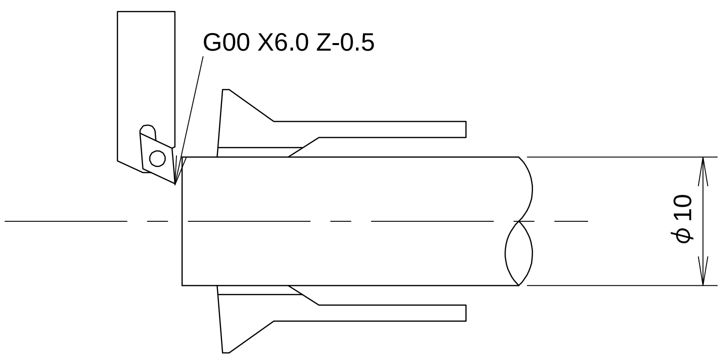
G02 X ☐ Z ☐ R ☐ F ☐…….Circular interpolation (clockwise) at a specified feed rate

Program 1

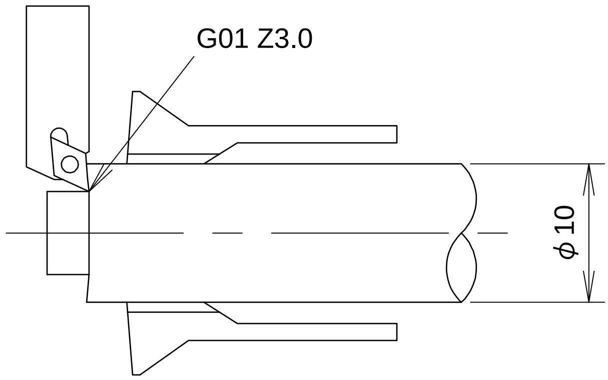
This sample shows G02 arc machining with a front turning tool.



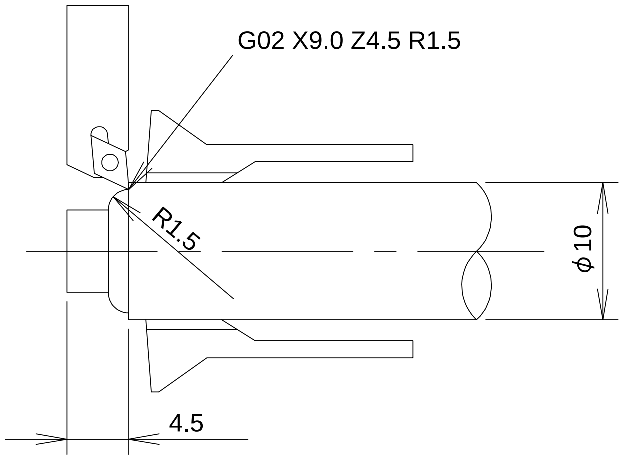
G00 X6.0 Z–0.5 ....Rapid feed positioning



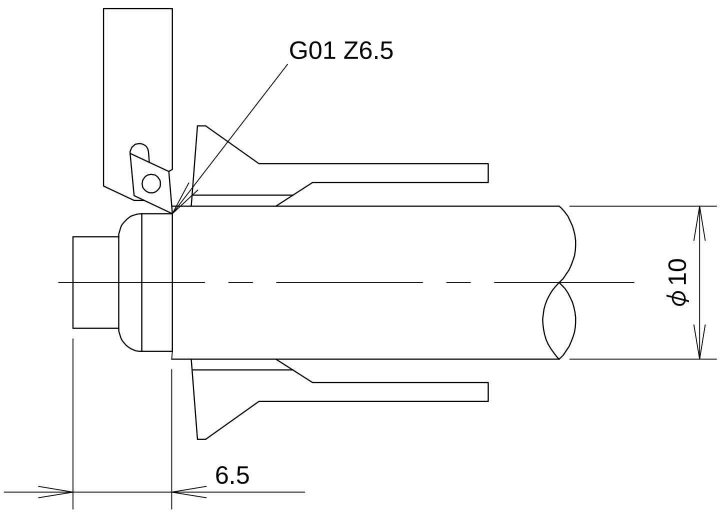
G01 Z3.0 F0.03............................Cutting up to Z3.0 in the linear interpolation mode at the feed rate F0.03.



G02 X9.0 Z4.5 R1.5 F0.03........Cutting up to X9.0, Z4.5 in the circular interpolation mode along an arc of radius R1.5 at the feed rate F0.03.



G01 Z6.5 F0.03............................Cutting up to Z6.5 in the linear interpolation mode at the feed rate F0.03.



G01 X11.0 F0.03..........................Moving the tool back to the **positioning point**, the position ø1.0 mm + material outer diameter.

5.7.4 G03 Circular interpolation (counterclockwise)

This mode is used to carry out machining from the current position to the specified position along an arc of radius R at the specified feed rate. This mode of control is called circular interpolation. Circular interpolation **G03** produces an arc in concave form.

It is not necessary to specify the center of the circle.

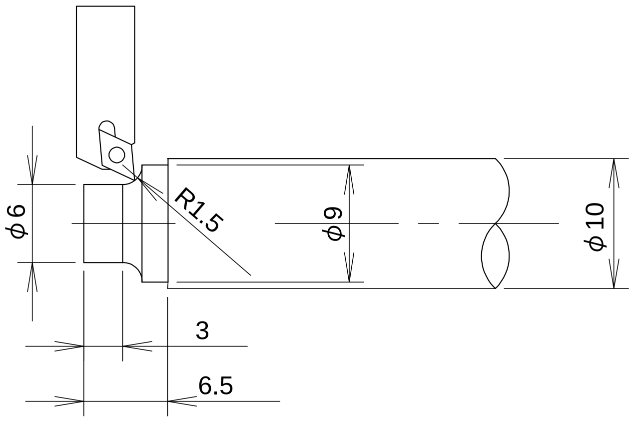
An F☐ command is used to specify the feed rate. An F☐ command is a modal command and the specified value remains valid until another F☐ command is specified.

Command format

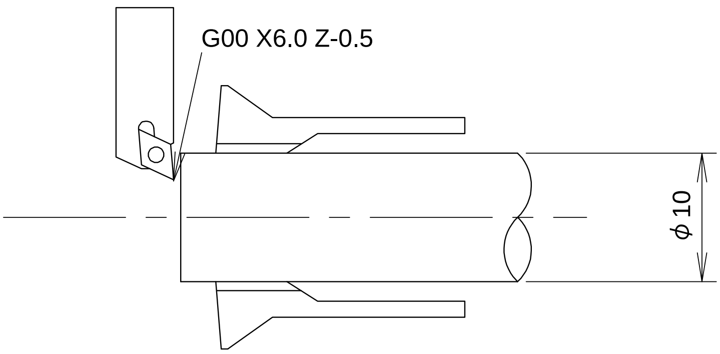
G03 X☐ Z☐ R☐ F☐……Circular interpolation (counterclockwise) at a specified feed rate

Program 1

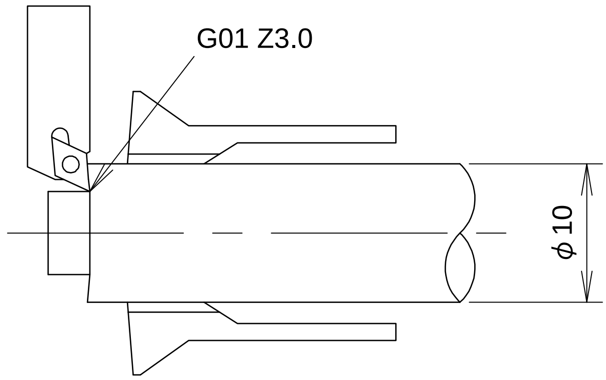
This sample shows G03 arc machining with a front turning tool.



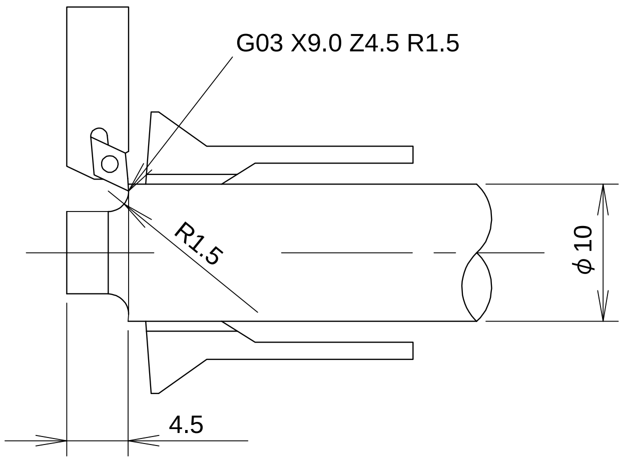
G00 X6.0 Z–0.5 ....Rapid feed positioning



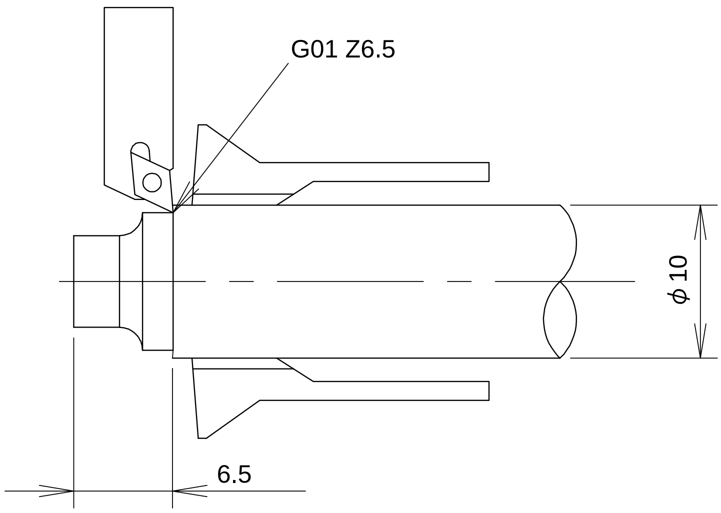
G01 Z3.0 F0.03............................Cutting up to Z3.0 in the linear interpolation mode at the feed rate F0.03.



G03 X9.0 Z4.5 R1.5 F0.03........Cutting up to X9.0, Z4.5 in the circular interpolation mode along an arc of radius R1.5 at the feed rate F0.03.



G01 Z6.5 F0.03............................Cutting up to Z6.5 in the linear interpolation mode at the feed rate F0.03.



G01 X11.0 F0.03..........................Moving the tool back to the **positioning point**, the position ø1.0 mm + material outer diameter.

5.7.5 ,G04

G04 suspends program execution for the specified length of time while retaining the current status.

For G04, although U and P arguments can be used, U argument is generally used.

The function to suspend program execution is called dwell.

Purpose of G04

- To stabilize a groove diameter
- To stabilize the spindle speed when the speed is changed largely
- To provide time for oil supply to a tool during the drilling process
- To stabilize machining when outer diameter machining is switched to end-face turning
- To break chips

The allowable maximum value that can be specified for G04 is 9999.999 seconds.

Command format

G04 U$\boxed{.\square}$.................................. For U argument, specify the length of dwell time in units of seconds. The numeric value must always contain a decimal point.

G04 P$\boxed{}$.................................. For P argument, specify the length of dwell time in units of 1/1000 seconds. No decimal point is used.

Program

G04 U2.5.. 2.5 seconds as dwell time. The machine holds the status for 2.5 seconds.

G04 P2500...................................... 2.5 seconds as dwell time. Specifying P2500 has the same effect as specifying U2.5.

5.7.6 G98 and G99

G98 and G99 specify how an F argument that determines feed rate in such as the G01 or G02 mode axis feed operation is interpreted. G98 calls the feed per minute mode and a numeric value specified by an F argument in this mode is interpreted as the distance a tool moves per minute. G99 calls the feed per rotation mode and a numeric value specified by an F argument in this mode is interpreted as the distance a tool moves per spindle rotation. Once specified, G98 and G99 retain the mode.

G98: Used for the machining to be carried out while stopping the spindle. In other words, this mode is used for the machining that uses a rotary tool such as an end mill.

G99: Used for the turning operation, such as front turning, to be carried out while rotating the spindle. Axis feed is not possible unless the front spindle is rotating. G99 mode is selected when the power is turned on.

Command format

G98..Feed per minute: Feed distance (mm) of a tool per minute (mm/min)

G99..Feed per rotation: Feed distance (mm) of a tool per one turn of the front spindle (mm/rev)

Program (G98)

G98..Feed per minute mode

G01 X5.0 F110..............................Cutting up to X5.0 at the feed rate of 110 mm/min.

G01 Z13.0.....................................Cutting up to Z13.0 at the feed rate of 110 mm/min.

……..

……..

Program (G99)

G99..Feed per rotation mode

G01 X5.0 F0.05............................Cutting up to X5.0 at the feed rate of 0.05 mm per front spindle rotation.

G01 Z13.0.....................................Cutting up to Z13.0 at the feed rate of 0.05 mm per front spindle rotation.

……..

……..

5.8 M Codes

The function called by M codes is called the auxiliary function which specifies auxiliary machine control functions such as front spindle forward rotation / stop / reverse rotation excluding axis controls. Some auxiliary functions expressed by the same M code provide different functions depending on the machine type. An auxiliary function is specified in the format of M☐☐.

The following table lists frequently used M codes:

Code	Description	Refer to
M00	Program stop	
M01	Optional stop	
M02	Cycle stop	5.8.5
M03	Front spindle forward rotation	5.8.1
M04	Front spindle reverse rotation	5.8.2
M05	Front spindle stop	5.8.3
M06	Chuck close	
M07	Chuck open	
M08	Material replacement program ON	8.7.2
M09	Material replacement program OFF	8.7.2
M32	Workpiece separator advance	
M33	Workpiece separator retract	
M52	Coolant ON	
M53	Coolant OFF	
M54	Bar loader torque OFF	8.7.1
M55	Material replacement start	8.7.1
M56	Product count	5.8.4
M98	Subprogram call	8.6
M99	Program return	5.8.6

5.8.1 M03 Front spindle forward rotation

M03 rotates the front spindle forward. The front spindle rotates in the counterclockwise direction when viewed from the end face of the material. Once the spindle rotation is specified, the spindle speed can be changed by only specifying an S argument command. Spindle speed command format differs depending on the NC unit.

NC unit produced by FANUC

Command format

M03 S☐ .... A numeric value specified in ☐ following address S indicates a spindle speed or the number of spindle rotations per minute (min^{-1}). A spindle speed value must be specified without a decimal point.

Program

M03 S2000 .... Indicates the front spindle to rotate forward at a speed of 2000 min^{-1}.

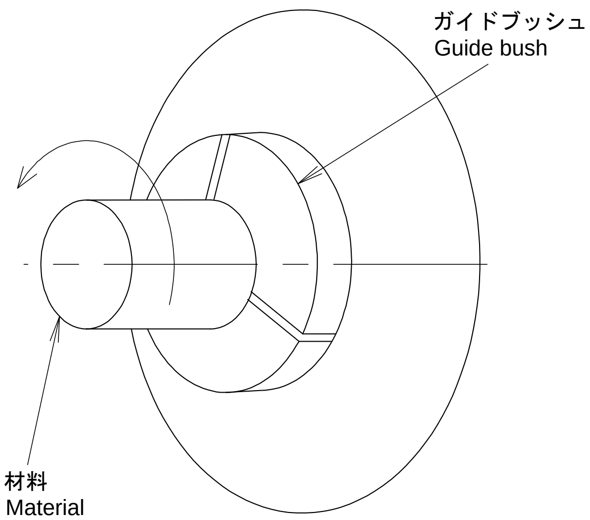
NC units produced by Mitsubishi Electric and some NC units (A20 and K series) produced by FANUC

Command format

M03 S1=☐ .... A numeric value specified in ☐ following S☐ = indicates a spindle speed or the number of spindle rotations per minute (min^{-1}) and a numeric value specified following address S indicates the spindle number. S1 indicates the front spindle. A spindle speed value and a spindle number must be specified without a decimal point.

Program

M03 S1=2000 .... Indicates the front spindle to rotate forward at a speed of 2000 min^{-1}.



5.8.2 M04 Front spindle reverse rotation

M04 rotates the front spindle reverse. The spindle rotates in the clockwise direction when viewed from the end face of the material. Once the spindle rotation is specified, the spindle speed can be changed by only specifying an S argument command. Spindle speed command format differs depending on the NC unit.

NC unit produced by FANUC

Command format

M04 S☐ .... A numeric value specified in ☐ following address S indicates a spindle speed or the number of spindle rotations per minute (min^{-1}). A spindle speed value must be specified without a decimal point.

Program

M04 S2000..................................... Indicates the front spindle to rotate reverse at a speed of 2000 min^{-1}.

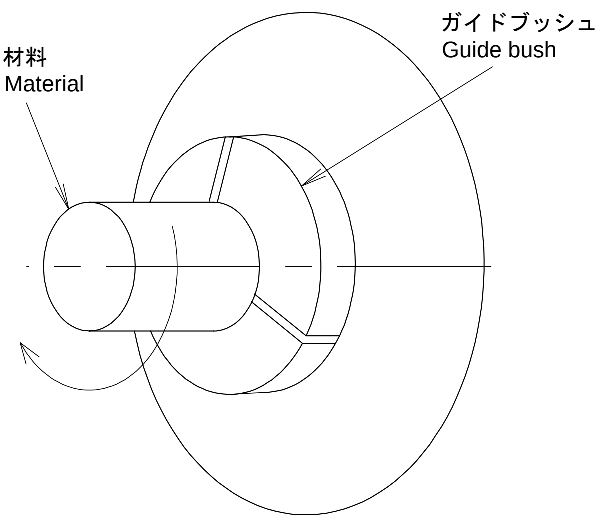
NC units produced by Mitsubishi Electric and some NC units (A20 and K series) produced by FANUC

Command format

M04 S1=☐ .... A numeric value specified in ☐ following S☐ = indicates a spindle speed or the number of spindle rotations per minute (min^{-1}) and a numeric value specified following address S indicates the spindle number. S1 indicates the front spindle. A spindle speed value and a spindle number must be specified without a decimal point.

Program

M04 S1=2000................................. Indicates the front spindle to rotate reverse at a speed of 2000 min^{-1}.



5.8.3 M05 Front spindle stop

Command format

M05...Stops the front spindle.

5.8.4 M56 Product count

M56 counts the number of products only when automatic continuous operation is performed.

Command format

M56...Product count

5.8.5 M02 Cycle stop

Be sure to specify cycle stop at the end of a program.

Command format

M02...Cycle stop

5.8.6 M99 Program return

Command format

M99...Return to main program

The program return function of M99 is included in M02 depending on the machine type (B12 or R04). In this case, you can omit M99.

5.9 T Codes

A T code has different meaning according to the number of digits of numeric characters specified following address T.

A 4-digit number specifies a tool selection command and reading of a tool position compensation data, and a 2-digit number specifies reading of tool position compensation data.

Command format

4-digit T code

T□□△△ — This command selects a tool, and reads the compensation data at the same time. The first two digits indicate the tool number, and the last two digits the tool position compensation number.

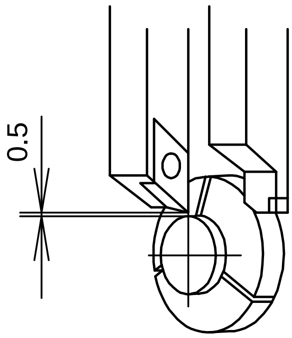
For the last two digits, "00" indicating compensation cancel is usually specified so that the tool position compensation data is not read simultaneously with the tool selection.

2-digit T code

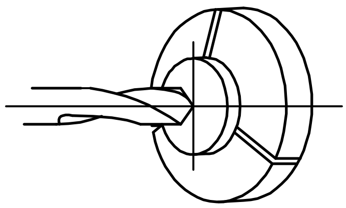
T△△ — Use this command to enter or change the tool position compensation data when using the currently selected tool for the subsequent operation. The numeric characters indicate the tool position compensation number.

T00 — **Cancels tool position compensation function.**

On Cincom, tool mounting positions are fixed and a tool number is allocated to these positions. When a T code is specified, the tool mounted at the position specified by the tool number moves to the specified position. When T1100 is specified, for example, the tool mounted at the T11 position moves to the position specified for carrying out machining.



When an outer diameter turning tool number is specified, the corresponding tool is positioned to the position of "material diameter + 0.5 mm" as shown in the figure above (0.5 mm is the value set at the tool position point (DIA) to the machining data and it can be changed according to the setting). In this positioning movement, there are no movements along the Z axis. This point is called the positioning point.



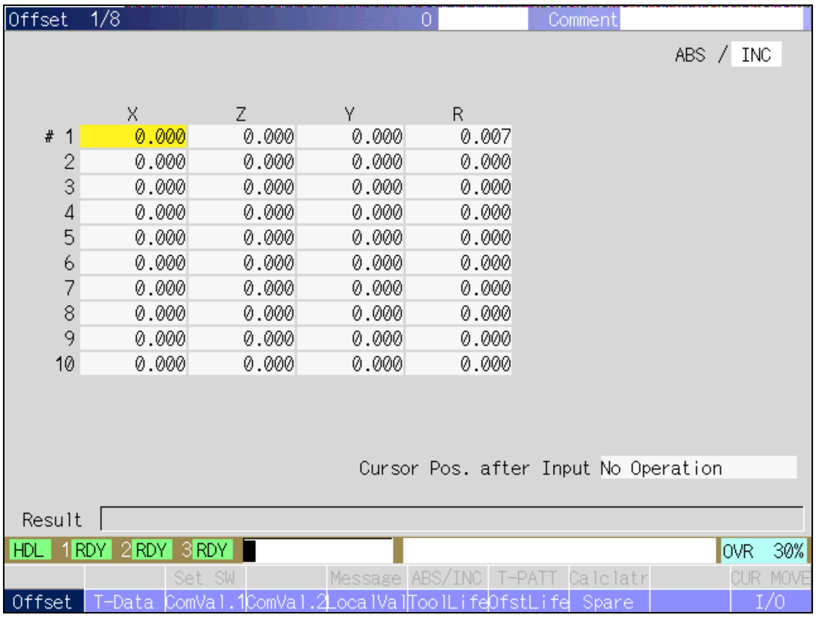
When an inner diameter turning tool number is specified, the corresponding tool is positioned at the center of the material (X0) shown in the figure above. In this positioning movement, there are no movements along the Z axis. When carrying out inner diameter turning, always specify the command such as "G0 Z–1.0" before specifying a tool selection command. If a tool selection command is specified without such commands that moves a tool apart from the end face of the material, the tool can interfere with the material.

5.10 Tool Position Compensation (Offset)

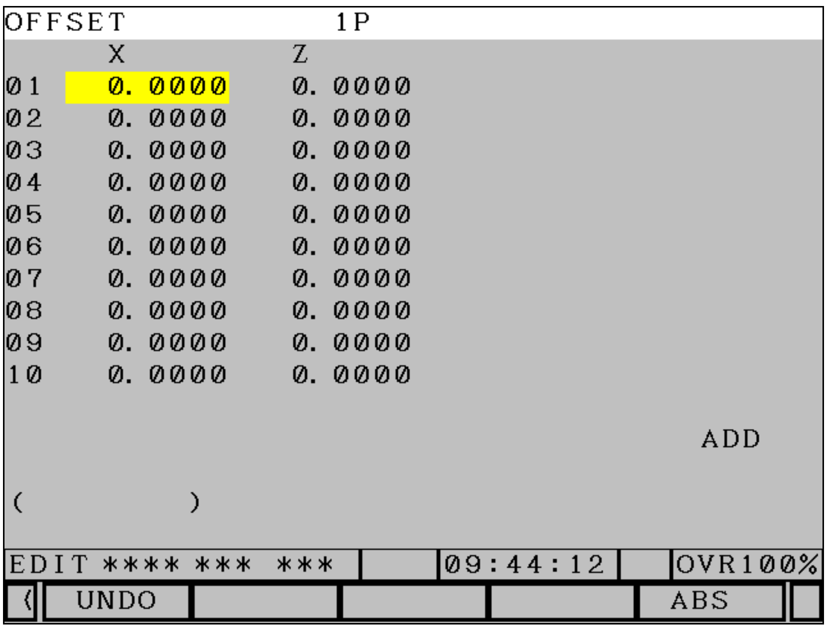
Tool position compensation means the function that corrects the difference between the dimensions in which a workpiece was actually machined and the dimensions specified in the drawing. Using this function, you do not have to change any coordinate values in the program in order to correct workpiece dimensions. All you have to do is only to input the numeric value as the difference between the measured dimensions and the dimensions specified in the drawing. To use the compensation function, however, you have to write the corresponding commands in the program. The tool position compensation function is sometimes called the offset function.

The offset data inputting screens differ depending on the machine type. The following shows the representative screens.

NC unit produced by Mitsubishi Electric



NC unit produced by FANUC



- The compensation value is added when an axis movement command is executed.
- The current compensation values are effective until the next tool selection command or other compensation command is executed.
- Generally, use values 01 to 16 as compensation numbers. Note that the numeric characters that can be used as compensation numbers vary depending on the machine type.
- The compensation function must be canceled when a tool returns to the start position after completing cut-off machining.
- When the next process is not tool selection (changing the machining pattern or the like), specify the tool position compensation cancel command after the completion of the current process.

Program

Assume that compensation values 0.1 and –0.05 are set for X and Z of tool compensation number T01.

T0300... T03 tool moves to the positioning point while canceling the tool position compensation function.

G00 Z–0.5 .... The tool moves to Z–0.5 but the tool position compensation value is not added.

G00 X8.0 T01............................... Compensation value X set at compensation number T01 is added to tool movement along the X axis so that the tool moves to X8.1. In this block, the tool does not move along the Z axis.

G00 Z5.0....................................... Compensation value Z set at compensation number T01 is added to tool movement along the Z axis so that the tool moves to Z4.95.

5.10.1 Using the tool position compensation function for outer diameter turning tools

For front turning tools, secondary machining tools and boring tools, call the compensation function when executing positioning at a rapid feed rate.

Program

T0300..Selects T03 tool.

G00 X8.0 Z–0.5 T01...................Compensation values are added in rapid feed positioning to X8.0, Z–0.5.

…….

…….

G00 X13.0 T00.............................Moving the tool to the positioning point at a rapid feed rate. The tool position compensation function is canceled during this positioning movement. If cutting is carried out while a tool moves to the positioning point, do not cancel the compensation function.

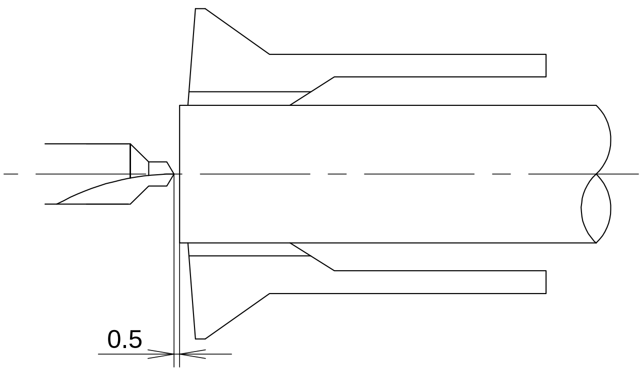
5.10.2 Using the tool position compensation function for center hole machining tools

For center hole machining tools, such as drills and taps, that do not require feed along the X axis, call the tool position compensation function when carrying out machining along the Z axis. Cancel the compensation function when the tool returns after the completion of machining. By canceling the compensation function in this manner, the tool can always be positioned at the machining start position disregarding the compensation function.

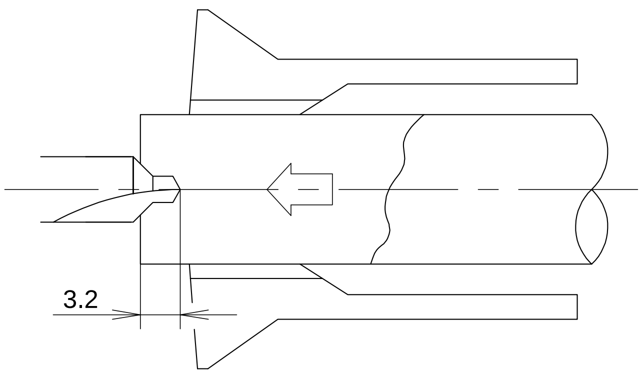
Program

`T2100`..Selects T21 tool.

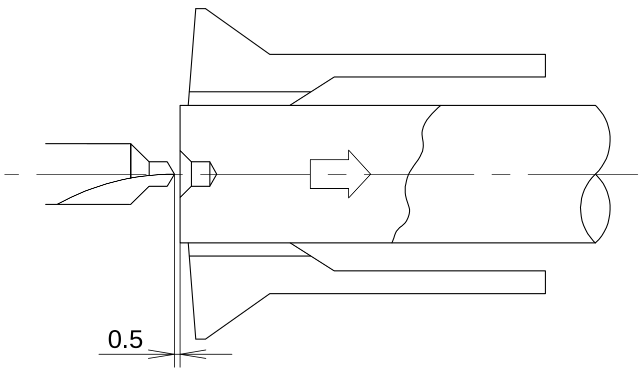
`G00 Z–0.5` ....Positioning to the machining start position at a rapid feed.



`G01 Z3.2 F0.1 T05`......................Drilling while adding the compensation value Z set at compensation number T05.



`G00 Z–0.5 T00` ....Returning to the machining start position Z–0.5 while canceling the tool position compensation function.



5.11 Machining Processes

Machining processes include various processes such as front turning, grooving and back turning. When programming such processes, however, the basic structure of programming explained below can be used.

(1) Sequence number

Specify any number that can identify the respective processes. As an example, specify a process sequence with the first two digits and a tool number with the last two digits.

(2) Spindle speed of each spindle

Specify the main spindle rotation or orientation command. Rotary tool rotation command is also specified here.

(3) Tool selection

Specify the tool number where the tool to be used is mounted. The selected tool moves to the positioning point.

(4) Coordinate system shift

Shift the coordinate system meeting the shaped of the tool to be used. It is not necessary to specify the coordinate system shift command for tools such as a front running tool.

(5) Positioning to the approach point

Move the tool to the position where machining starts in the G00 (rapid feed) mode. Usually, 0.5 mm clearance is taken between the material and the tool nose in this positioning. This point is called the approach point. Call the tool position compensation function when specifying the positioning to the approach point for machining processes other than center hole machining processes such as centering and drilling.

(6) Machining program

Create the program for carrying out machining with the selected tool.

(7) Moving to the positioning point (safe position)

Move the tool nose away from the material to the positioning point (safe position) after completing machining. Cancel the tool position compensation function in this step.

(8) Canceling the coordinate system shift

Cancel the coordinate system shift.

Program

```
N0121.............................................. (1) Enter a sequence number.
M03 S2300 (M03 S1=2300).......... (2) Start the main spindle in the forward direction.
T2100.............................................. (3) Select T21 tool.
G50 W-10.0 .................................. (4) Shift the coordinate system meeting the tool shape.
G00 X13.0 Z0.0........................... (5) Move the tool to the approach point at a rapid feed rate.
…….
…….
…….                                        (6) Machining program
…….
G50 W10.0.................................... (7) Cancel the compensation function and move the tool to the positioning
                                                   point.
G00 X13.0 T00.............................. (8) Cancel coordinate system shift.
```

5.12 Start Position

A series of programmed machining processes ends with the cut-off process, and the status of the end of a program is taken as the start of the next program. That is, beginning with the status the material is in contact with the cut-off tool after the completion of cut-off of the previous product, the headstock moves to the position where the material length necessary for machining the next product is secured. The machining for the next product starts in this status. This status that allows the starting of the next product machining is called the start position.

The start position is determined by the cut-off machining end position, the machining length and the cut-off tool number. On some types of machines, a cut-off tool number is not used. For details of machining data, refer to <8.12 Machining Data>.

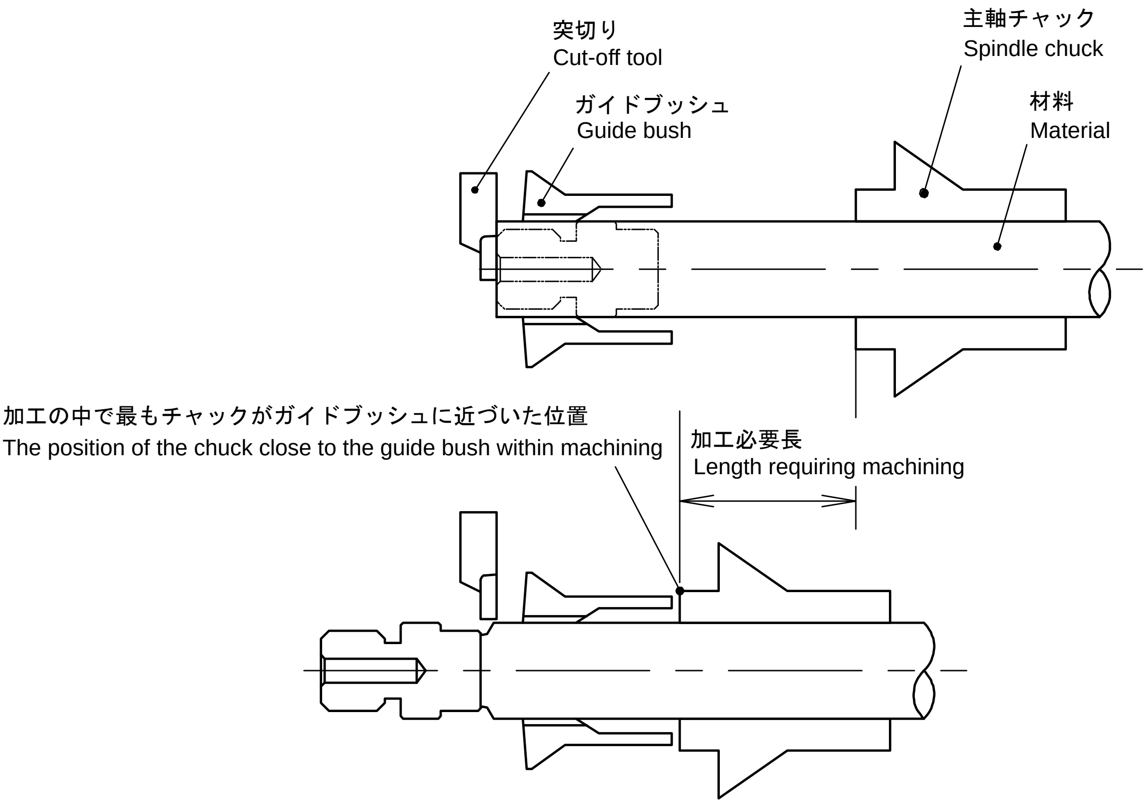


Fig. 5.12 Start position

- To execute a program, carry out the start position operation by opening the front spindle chuck in the preparation mode.

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	5	0	0						

6. Program Sample 1

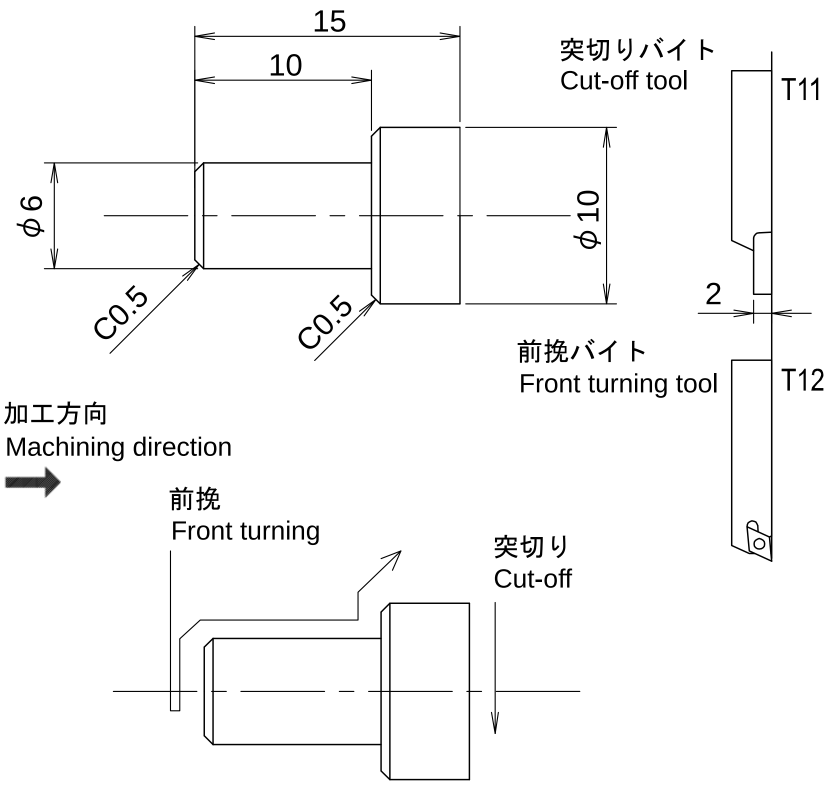
Code No.	C-CINCOM_IGB 4E1-0600	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

6.1 Machining Layout

This section explains programming steps using the machining layout shown below as an example.

Material: SUS 303 (free-cutting stainless steel)
Material diameter: ø10.0 mm



6.2 Preparation Process

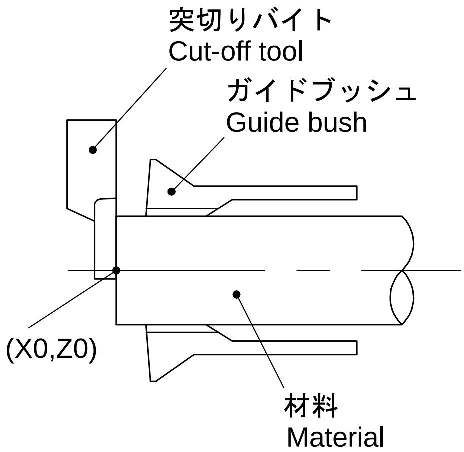
Preparation	Front Turning	Cut-off End

Key Point

In the preparation process, set the conditions required for the 1-cycle operation and start the operation. A program starts from the status the cut-off machining has been finished (start position).

Program

G50 Z0..Set the coordinate system taking the current Z axis position (start position) as Z0. In other words, the Z coordinate of the end face of the material is set as Z0.



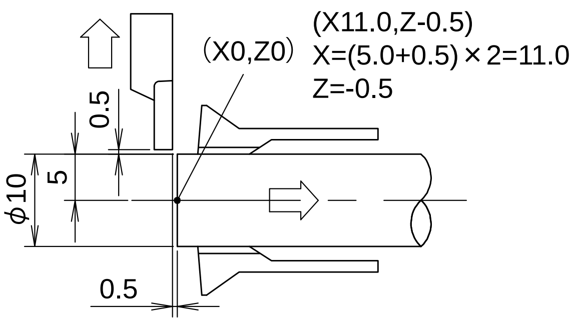
M06..Close the front spindle chuck.

G04 U0.2..Dwell time

M96(G26) ....Turn the spindle speed change detection function ON. (Refer to 8.8 for details.)

G99..Specify the feed per rotation mode.　Use this mode for normal turning operations (front turning, cut-off, etc.).

G00 X11.0 Z–0.5..........................Move the cut-off tool away from the material to start the next process. Specifying X11.0 (material diameter + 1.0 mm) moves the tool nose to the position 0.5 mm away from the material outer diameter. The Z–0.5 command also moves the tool nose to the position 0.5 mm away from the material end face in the longitudinal direction.



M03 S5300 (M03 S1=5300) ....Start the front spindle in the forward direction.

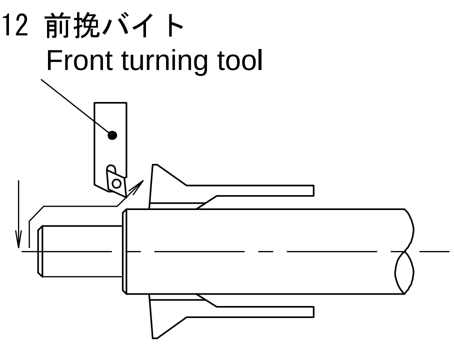
6.3 Front Turning

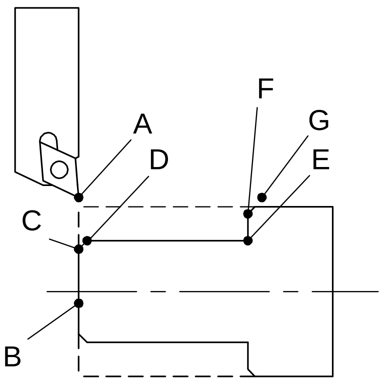
Preparation	**Front Turning**	Cut-off End

Key Point

Front turning is the most common type of turning operation performed on a lathe.

In a machining program, define the path along which the tool nose moves by specifying the points in the sequence it moves, starting from point A, point B and then to point G, in the G01 (cutting feed) mode.





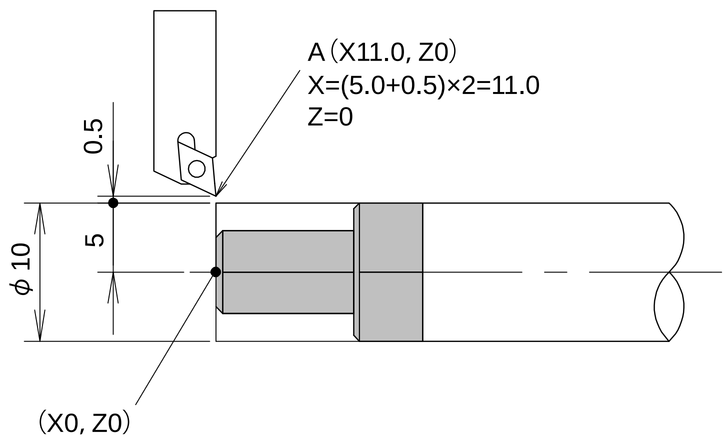
Program

N0112..Enter the sequence number.

M03 S5300 (M03 S1=5300) ....Start the front spindle in the forward direction.

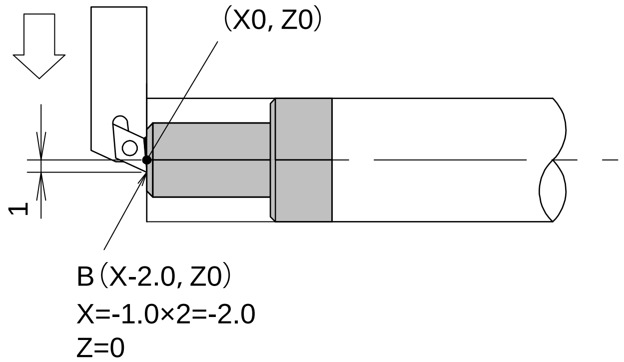
T1200..Select the front turning tool T1200.

G00 X11.0 Z0 T12........................Execute positioning to X11.0, Z0 at a rapid feed rate. The tool nose is positioned 0.5 mm away from the material. Specify the tool position compensation command T12.
The commands feed the tool to Point A where machining starts (approach point).

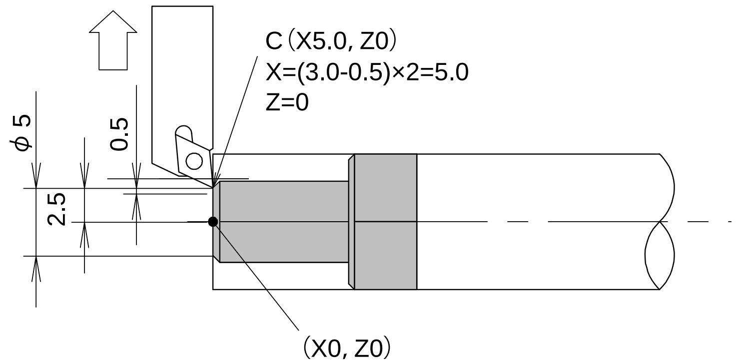


Preparation	**Front Turning**	Cut-off End

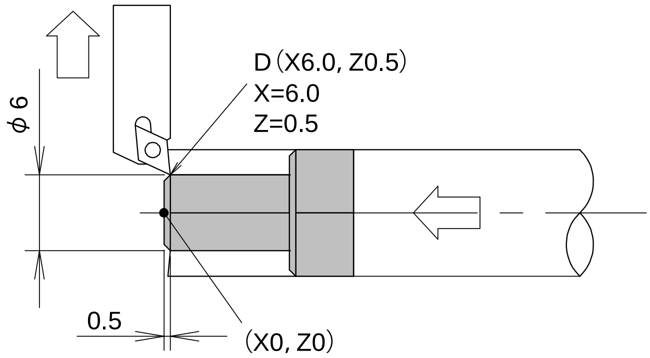
`G01 X–2.0 F0.015`........................Execute cutting up to X–2.0.
The commands move the tool to Point B to finish the end face of the material in front turning operation.



`G01 X5.0 F0.2`..............................Move the tool to X5.0 at a cutting feed rate.
The commands move the tool to Point C where chamfering start. In this operation, the tool moves at a fast feed rate F0.2 since the material is not cut.

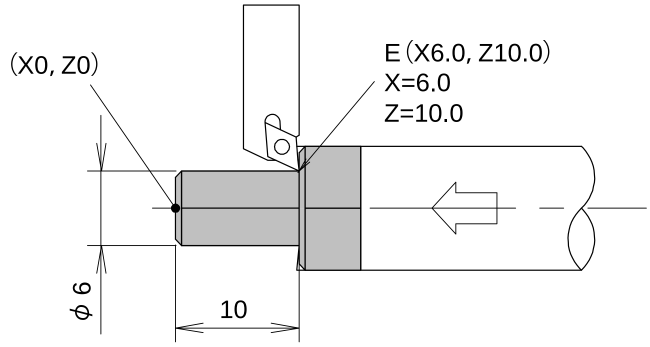


`G01 X6.0 Z0.5 F0.015`................Execute cutting up to X6.0, Z0.5.
The commands move the tool to Point D to chamfer the edge to C0.5.

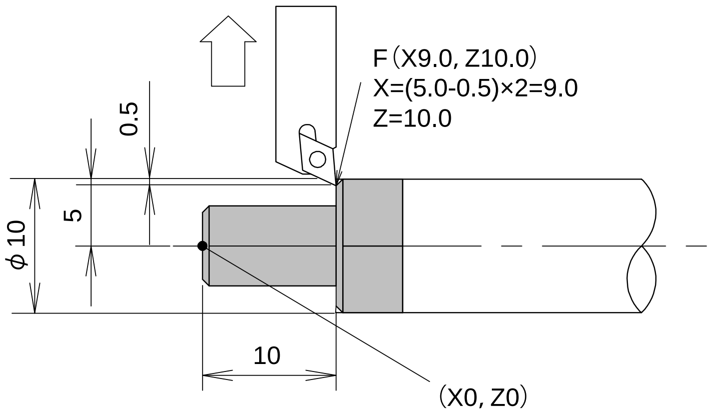


Preparation	**Front Turning**	Cut-off End

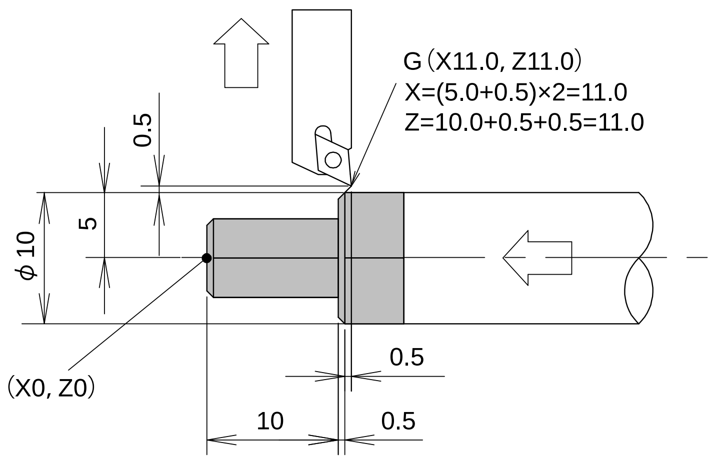
G01 Z10.0 F0.03..........................Execute cutting up to Z10.0.
The commands move the tool to Point E.



G01 X9.0 F0.05............................Execute cutting up to X9.0.
The commands move the tool to Point F where chamfering starts.



G01 X11.0 Z11.0 F0.015...........Execute cutting up to X11.0, Z11.0.
The commands move the tool to Point G to chamfer the edge to C0.5 and further to a position away from the material.



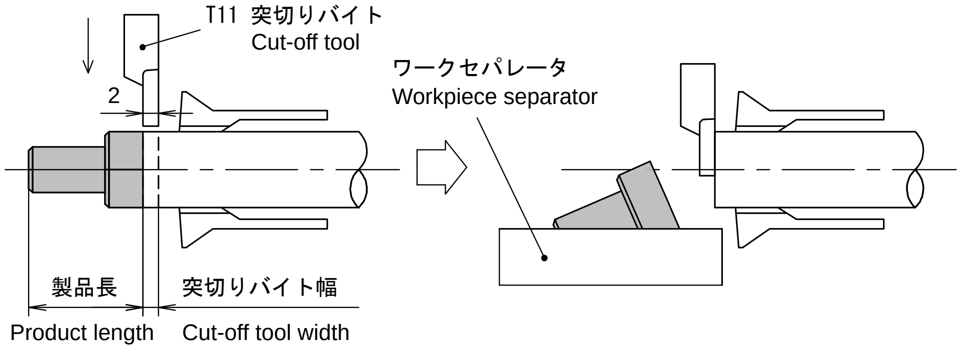
6.4 Cut-off End Process

Preparation	Front Turning	**Cut-off End**

Key Point

In the cut-off end process, set the cut-off commands and the commands required for ending the 1-cycle operation.

The cut-off machining separates the finished product from the material.



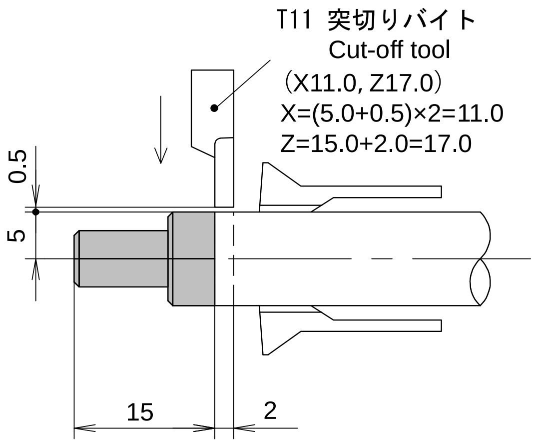
Program

N0211... Enter the sequence number.

M03 S3100 (M03 S1=3100).......... Start the front spindle in the forward direction.

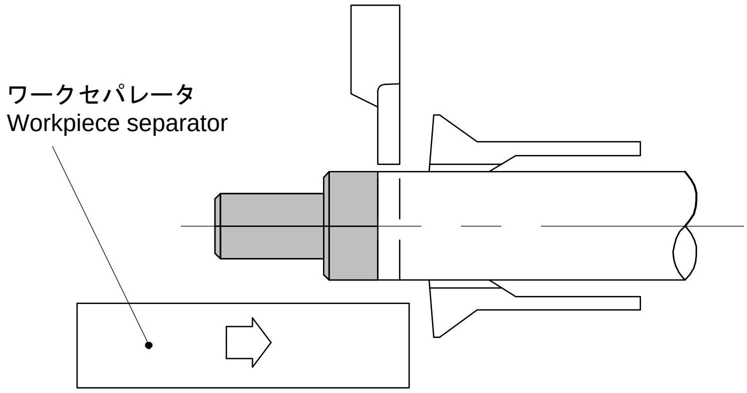
T1100... Select the cut-off tool T1100.

G00 X11.0 Z17.0 T11.................. Execute positioning of the cut-off tool to cut off the product in the correct length (Product length + Cut-off tool width; Z15.0 + 2.0 = Z17.0). Specify the tool position compensation command T11.
The commands move the tool to the position where machining starts (approach point).

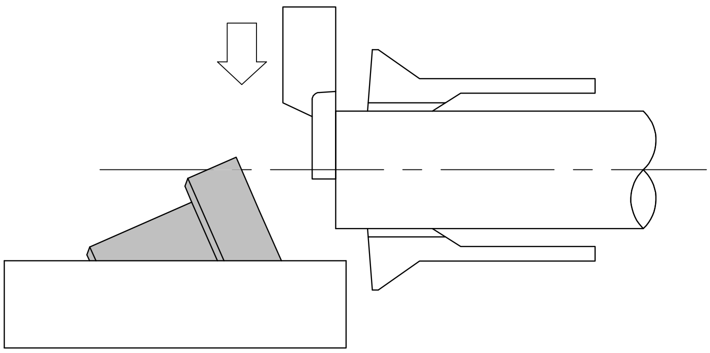


Preparation	Front Turning	**Cut-off End**

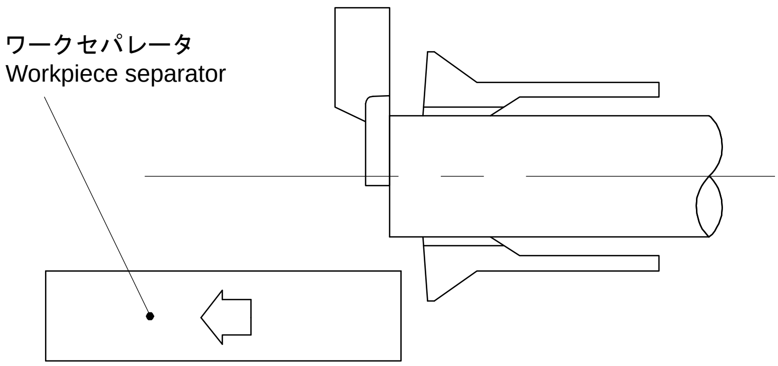
M32.. The workpiece separator moves forward. The program for product collection may differ from this depending on the specification of the machine type to be used. Refer to the Instruction Manual provided with the machine.



G01 X–3.0 F0.015........................ Move the tool up to X–3.0.
The commands execute cut-off machining by feeding the tool to Point B.



M33.. The workpiece separator moves backward. The program for product collection may differ from this depending on the specification of the machine type to be used. Refer to the Instruction Manual provided with the machine.



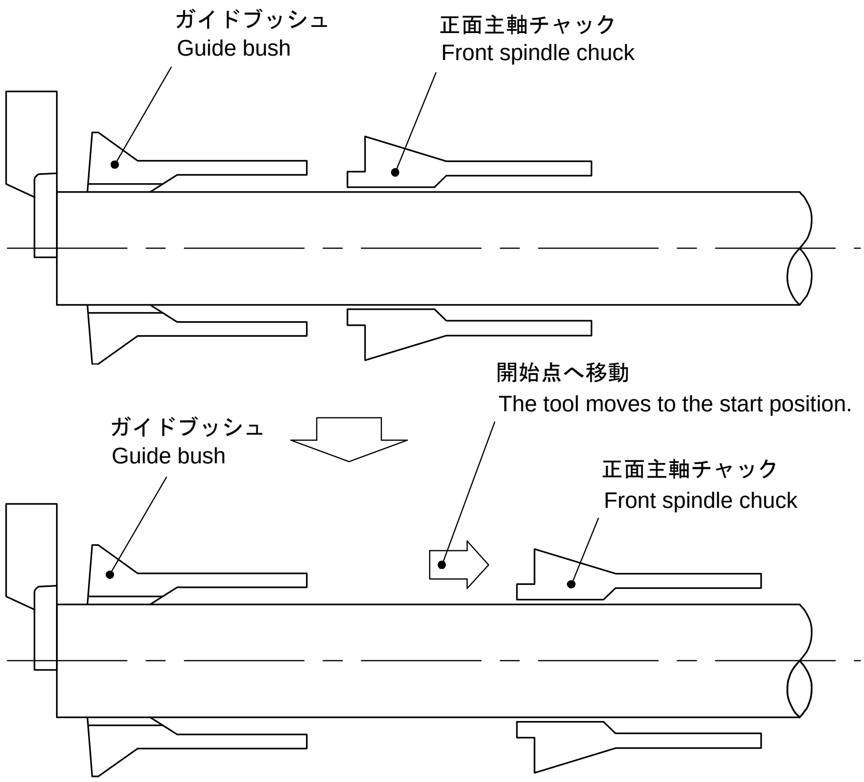
Preparation	Front Turning	**Cut-off End**

M05...Stop the front spindle.

M07...Open the front spindle chuck.

G04 U0.2..Dwell time

G00 X–3.0 Z0 T00........................Return the X and Z axes to the start position at a rapid feed rate. The Z coordinate must be the same value specified with G50Z☐☐ in the preparation process at the beginning of programming. For the X coordinate, specify the same value set at “Cut-off End (DIA)” in the machining data.
Since the tool position compensation function must be canceled at the end of a program, specify T00 here.



- The material does not move since the front spindle chuck is open.

M56...Count the number of products. The counter value increases by one only in continuous operation. The products machined in 1-cycle or 1-block operation are not counted.

M02...1 cycle stop (Control returns to the beginning of the program.)

M99...Program return. Control returns to the beginning of the program. (There is no need to enter M99 depending on the machine type.)

%..This is a stop code. The stop code is automatically entered to a program when it is created on the NC unit. The stop code is used when inputting or outputting programs.

(Blank page)

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	6	0	0						

7. Program Sample 2

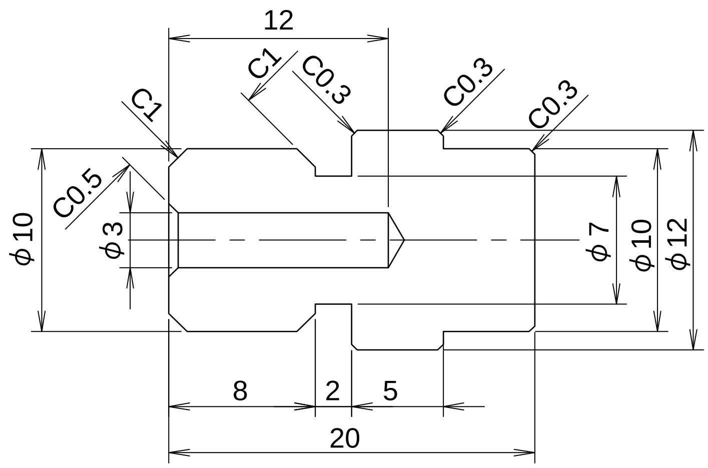
Code No.	C-CINCOM_IGB 4E1-0700	MFG No.	CINCOM	Issue Date	2012.6

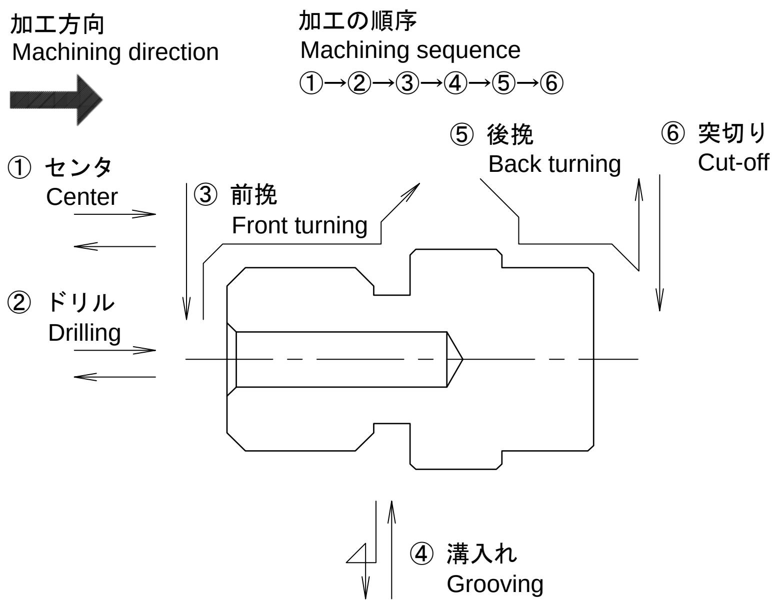
(Blank page)

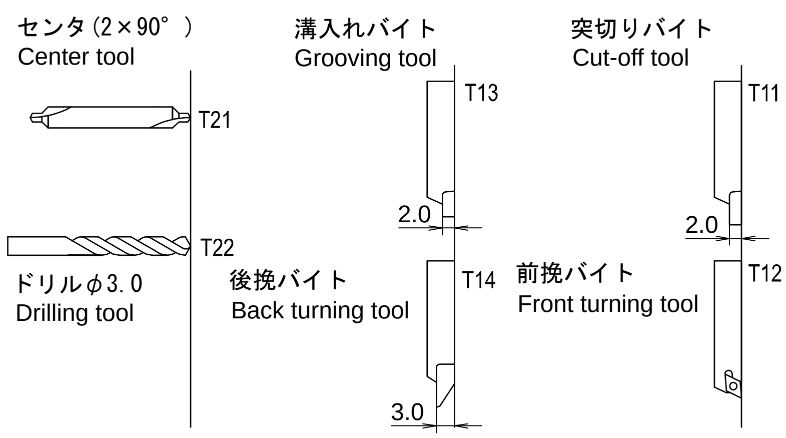
7.1 Machining Layout

This section explains programming steps using the machining layout shown below as an example.

Material: SUS303 (free-cutting stainless steel)
Material diameter: ø12.0 mm



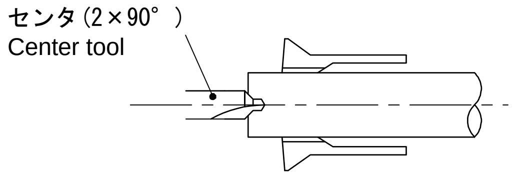




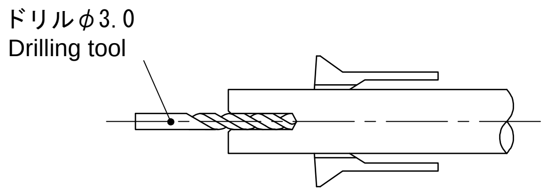
7.2 Machining Process Sequence

The sample workpiece is finished using the six kinds of representative tools shown below.

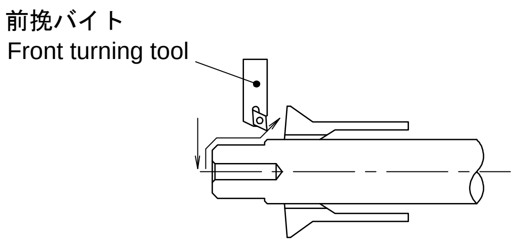
(1) Center drilling



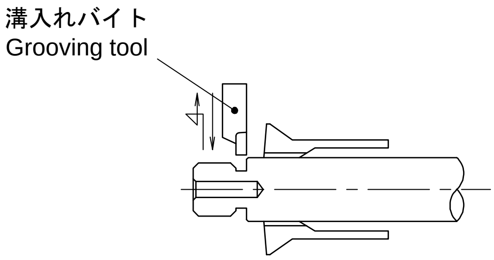
(2) Drilling



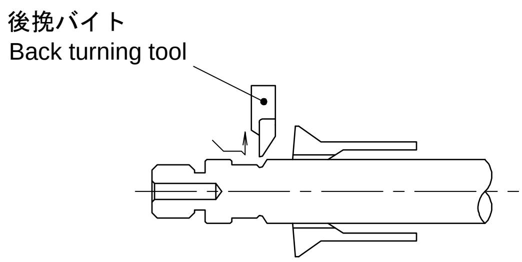
(3) Front turning



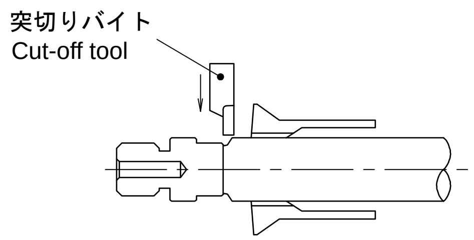
(4) Grooving



(5) Back turning



(6) Cut-off



7.3 Preparation Process

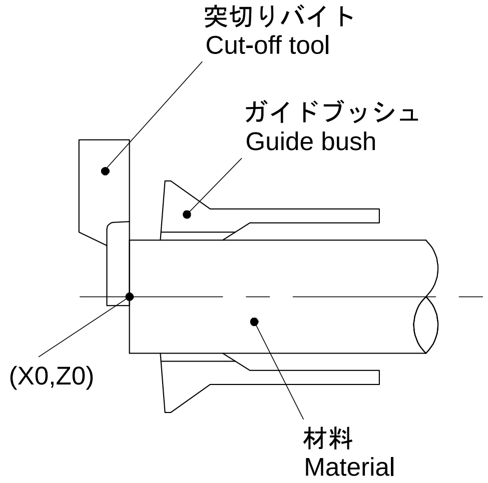
Preparation	Center Drilling	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

Key Point

In the preparation process, set the conditions required for the 1-cycle operation and start the operation. A program starts from the status the cut-off machining has been finished (start position).

Program

G50 Z0..Set the coordinate system taking the current position (start position) as Z0.



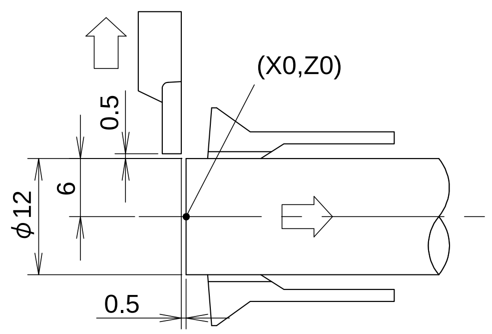
M06..Close the front spindle chuck.

G04 U0.2..Dwell time

M96(G26) ....Turn the spindle speed change detection function ON. (Refer to 8.8 for details.)

G99..Specify the feed per rotation mode.　Use this mode for normal turning operations (front turning, cut-off, etc.).

G00 X13.0 Z-0.5 ....Move the cut-off tool away from the material to start the next process. Specifying X13.0 (material diameter + 1.0 mm) moves the tool nose to the position 0.5 mm away from the material outer diameter. The Z–0.5 command also moves the tool nose to the position 0.5 mm away from the material end face in the longitudinal direction.



M03 S2300　(M03 S1=2300) ....Start the front spindle in the forward direction.

7.4 Center Drilling

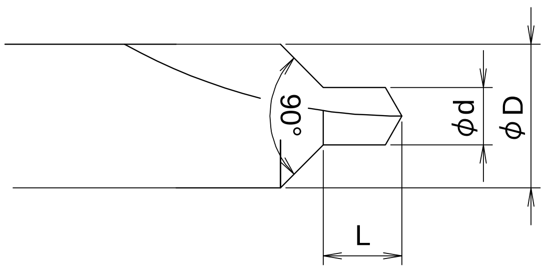
Preparation	**Center Drilling**	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

Key Point

On a lathe, center drilling is carried out without rotating a tool but a material rotates to machine a hole at the center on the end face of the material. Center drilling is effective to stable drilling operation.

A centering drill has the straight and tapered sections and their dimensions are specified by the JIS standards. Refer to the table below. In center drilling operation, use a drill that has a little smaller nominal diameter (ød) than the drill to be used in the next drilling operation. Feed the center drill to the depth where a tapered part is left at the mouth of the hole.

JIS Type A 90°

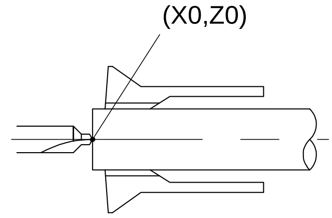


Nominal diameter d × 90°	D	L
1 × 90°	4	1.1
1.25 × 90°	5	1.4
1.6. × 90°	6.3	1.8
2 × 90°	8	2.2
2.5 × 90°	10	2.8
3.15 × 90°	11.2	3.6
4 × 90°	12.5	4.5
5 × 90°	16	5.6

* In addition to the centering drills specified by the JIS standards, many drills unique to the tool makers are also available. Refer to the catalogs of the tool makers for details.

Preparation	**Center Drilling**	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

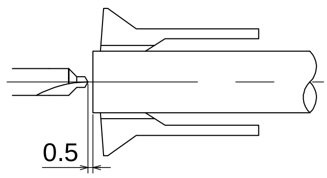
Specifying the tool number of a hole machining tool usually positions the tool to X0 even if G00 X0 is not specified. Specifying G00 Z0 aligns the centering drill nose with the center on the end face of the material as shown in the figure below. When specifying a tool number of a hole machining tool, confirm that the material does not extend from Z0. If it extends from Z0, the tool could interfere with the material.



Positioning pattern of a hole machining tool slightly varies according to the machine model.

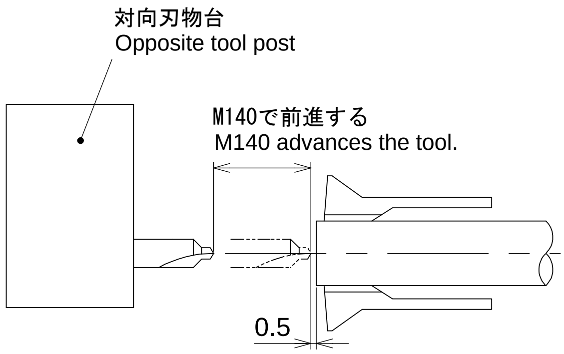
- Ordinary models

  Specify G00 Z–0.5. The tool nose moves to the position at the center of the material and 0.5 mm away from the material end face.



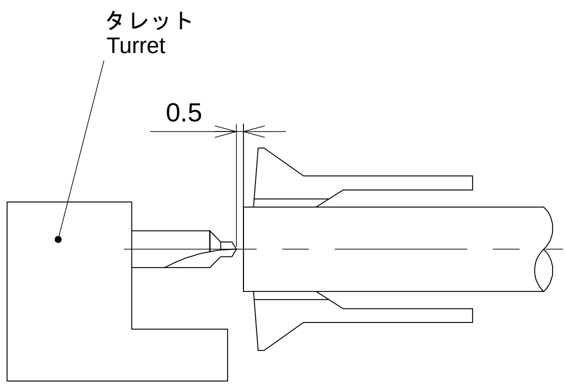
- Some L series machines

  For a hole machining tool mounted in the opposite tool post, specify M140 in the block next to the one where G00 Z–0.5 is specified. M140 moves the tool nose to the position 0.5 mm away from the material end face at the center of the material.



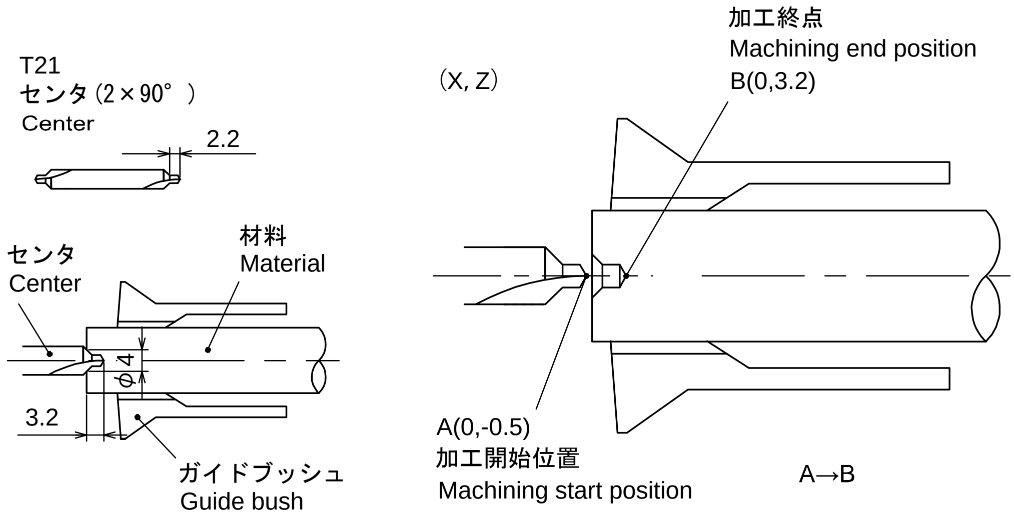
- M series machines

  For a hole machining tool mounted in the turret, specify G00 X0 Z–0.5. This moves the tool nose to the position 0.5 mm away from the material end face at the center of the material.



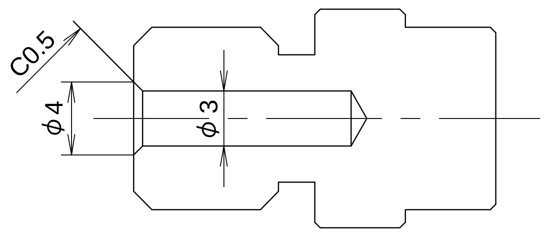
Preparation	**Center Drilling**	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

Sample



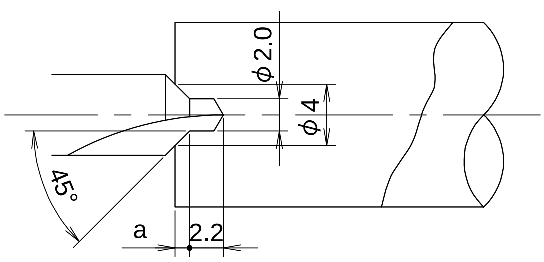
Calculating the coordinate values (depth of machining)

In this sample, C0.5 chamfer is required at the mouth of ø3.0 mm drilled hole. To achieve this, feed the centering drill to the depth that makes the diameter at the mouth of the hole ø4.0 mm.



To machine the diameter of the mouth to ø4.0 mm, it is necessary to feed the centering drill to Z = a + 2.2 mm position.

Since the diameter of the straight section of the centering drill to be used is ø2.0 mm, value “a” can be calculated as a = (4.0 – 2.0) / 2 = 1.0.



Accordingly, the coordinate value of Z where the centering drill must be fed is obtained as 1.0 + 2.2 = 3.2 mm.

Preparation	**Center Drilling**	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

Program

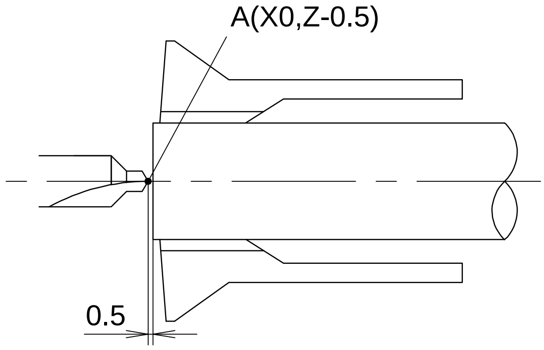
N0121..Enter the sequence number.

M03 S2300 (M03 S1=2300) ....Start the front spindle in the forward direction.

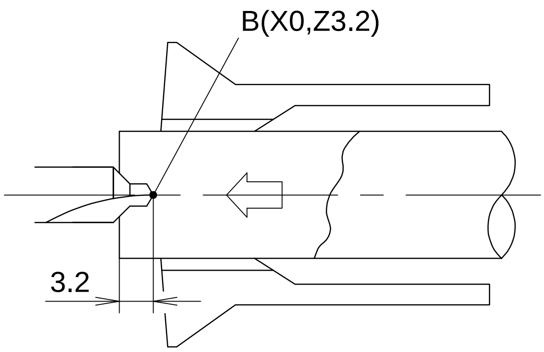
T2100..Select the centering drill T2100.

G00 Z-0.5 ....Execute positioning to the position 0.5 mm away from the end face of the material (approach point).

(M140) ....Move the opposite tool post forward. (only for some L series machines)

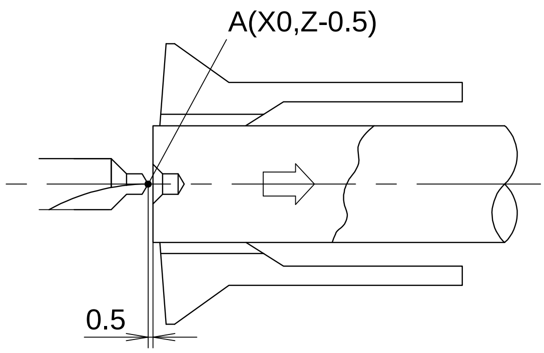


G01 Z3.2 F0.03 T21....................Execute cutting up to the depth of Z3.2. Specify the tool position compensation command T21.



G04 U0.2.......................................Specify the dwell function here to stabilize machining.

G00 Z-0.5 T00 ....Return the tool to the Z–0.5 position.
Specify T00 to cancel the tool position compensation function.



(M141) ....Move the opposite tool post backward. (only for some L series machines)

7.5 Drilling

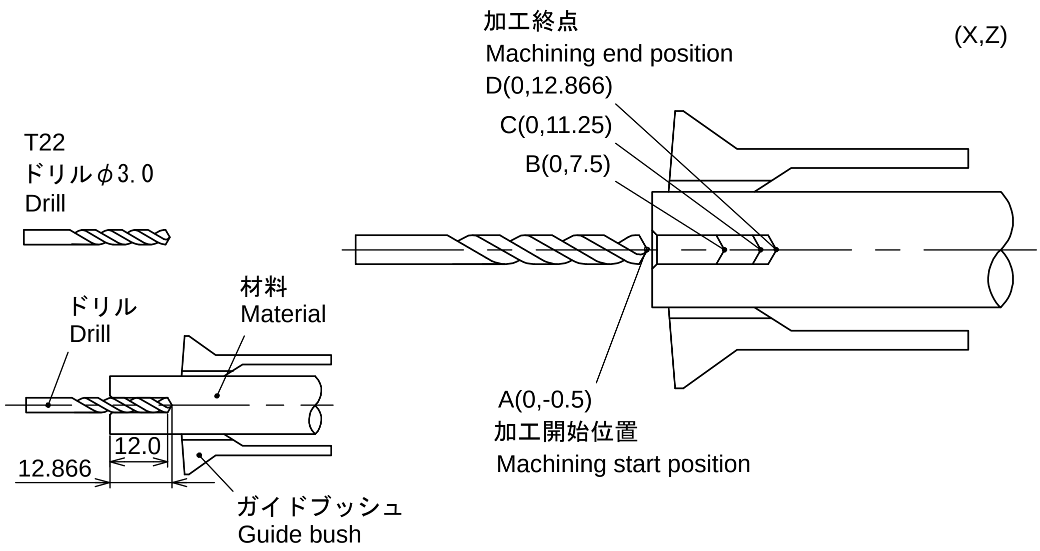
Preparation	Center Drilling	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

Key Point

Basically, drilling can be programmed in the same manner as center drilling. But drilling a deep hole requires programming different from center drilling program. When drilling a hole having the depth larger than two to three times the diameter, coolant cannot be supplied to the cutting point properly and chips cannot be expelled smoothly. Accordingly, it becomes necessary to retract a drill from the machined hole temporarily to supply coolant properly and expel chips from the drilled hole after feeding the drill to a certain depth. After that the drill should be fed into the machined hole again. Infeed and retraction of a drill must be repeated to machine a deep hole, and this type of operation is called stepping operation.

Preparation	Center Drilling	**Drilling**	Front Turning	Grooving	Back Turning	Cut-off End

Sample



A→B→A→Point 0.5 mm ahead B→C→A→Point 0.5 mm ahead C→D→A

When machining a 12.0 mm deep ø3 mm hole, the drill must be fed as deep as 12.866 mm with the front edge shape of the drill taken into consideration. Meanwhile, the allowable machining depth in one tool infeed operation is 7.5 mm when a hole is machined on SUS303 (free-cutting stainless steel) with a ø3 mm drill. Accordingly, machining a hole by feeding the drill as deep as 12.866 mm requires stepping operation.

How to determine the number of times infeed operations should be repeated and the depth of cut per infeed

Material	Tool	Depth of cut per infeed: ·n (mm) Infeed 1	Infeed 2 and later
Brass	Woodruff drill, Flat drill	Overall length	
Aluminum free-cutting steel	Twist drill	ℓ_1 = ød × 3.0	$\ell_n = \ell_{n-1}$ × 0.6～ød × 0.9 (mm)
Tool steel, Carbon steel Free-cutting stainless steel	Twist drill	ℓ_1 = ød × 2.5	$\ell_n = \ell_{n-1}$ × 0.5～ød × 0.9 (mm)
Stainless steel Difficult-to-cut material	Twist drill	ℓ_1 = ød × 2.0	$\ell_n = \ell_{n-1}$ × 0.4～ød × 0.9 (mm)

Permissible depth of cut per infeed differs depending on the material to be machined.

Since the material of the workpiece for the program sample is SUS303, find the values for free-cutting stainless steel in the table above.

Calculation of the depth of cut per infeed		Z coordinate value	
Infeed 1	ℓ_1=ø3 × 2.5＝7.5	Infeed 1	7.5
Infeed 2	$\ell_2=\ell_{2-1}$ × 0.5＝3.75	Infeed 2	$\ell_1+\ell_2$＝11.25
Infeed 3	$\ell_3=\ell_{3-1}$ × 0.5＝1.875	Infeed 3	$\ell_1+\ell_2+\ell_3$＝13.125

From the result of calculation above, it is found the depth reached in infeed 3 is 13.125 mm, which is larger than 12.866 mm or the depth to be machined. Therefore, the Z coordinate in infeed 3 should be 12.866.

Preparation	Center Drilling	**Drilling**	Front Turning	Grooving	Back Turning	Cut-off End

Program

N0222..Enter the sequence number.

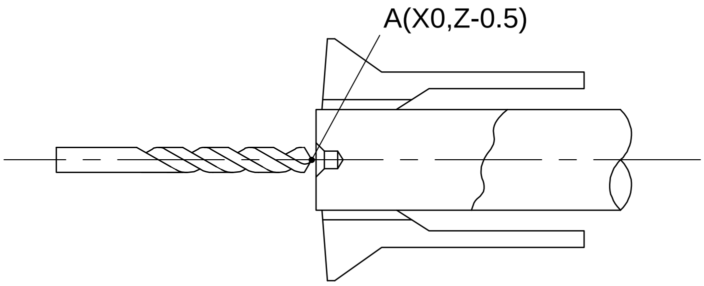
M03 S3100 (M03 S1=3100)..........Start the front spindle in the forward direction.

T2200..Select drill T2200.

G00 Z-0.5....................................Execute positioning of the tool to the position 0.5 mm away from the material end face in the longitudinal direction.

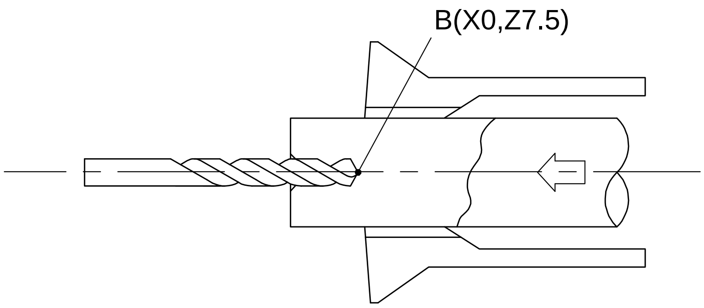
(M140)...Move the opposite tool post forward. (only for some L series machines)

The commands move the tool to Point A where machining starts (approach point).

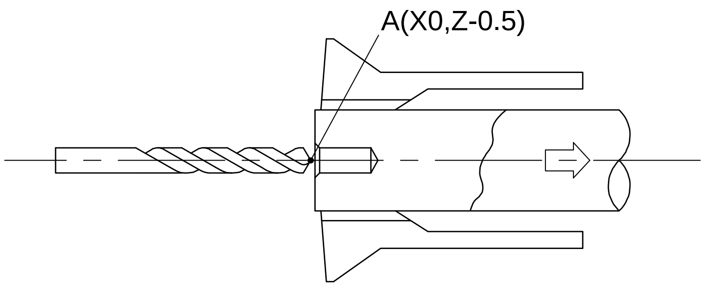


G01 Z7.5 F0.04 T22...................Execute the first infeed operation. Specify the tool position compensation command T22 in this line.

The commands execute drilling up to Point B (Z7.5 mm).



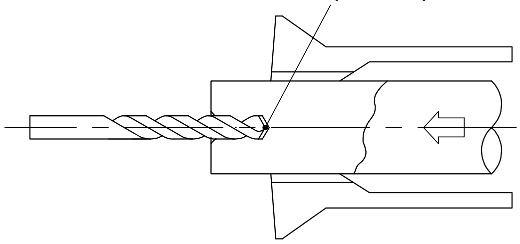
G00 Z-0.5....................................Return the tool to Point A (Z–0.5 mm).



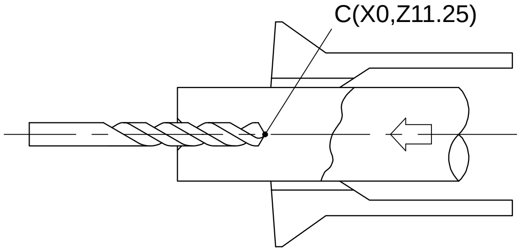
G04 U0.5.......................................Specify the dwell function (0.5 second dwell) to supply coolant to the drill.

Preparation	Center Drilling	**Drilling**	Front Turning	Grooving	Back Turning	Cut-off End

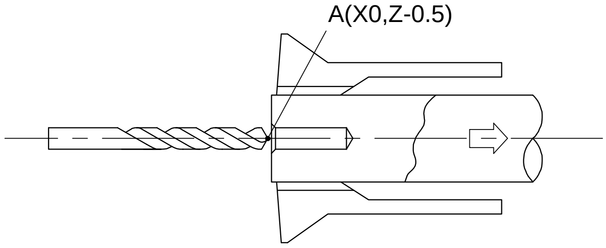
G00 Z7.0..Execute positioning to the start position for the second infeed operation. Specify the second infeed start position 0.5 mm above the position reached in the first infeed operation.



G01 Z11.25....................................Execute the second infeed operation. For this operation, the feed rate (F value), F0.04, specified for the first infeed operation is used since an F command is a modal command. The commands execute drilling up to Point C (Z11.25 mm).

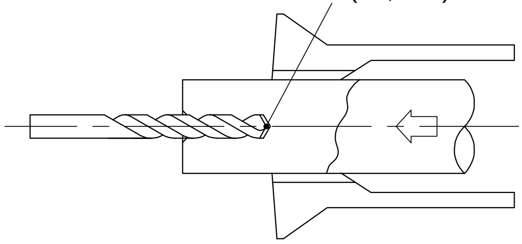


G00 Z−0.5 ....Return the tool to Point A or Z–0.5 mm position.



G04 U0.5..Coolant is applied to the drill for the dwell time (0.5 second).

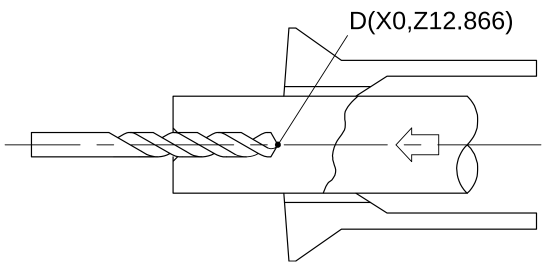
G00 Z10.75....................................Execute positioning to the start position for the third infeed operation. Specify the third infeed start position 0.5 mm above the position reached in the second infeed operation.



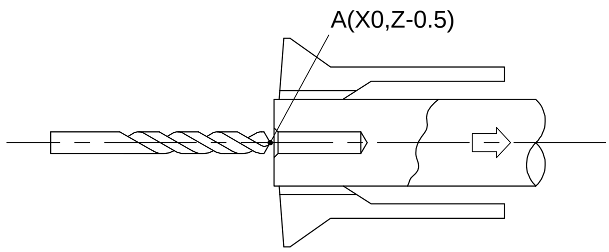
Preparation	Center Drilling	**Drilling**	Front Turning	Grooving	Back Turning	Cut-off End

`G01 Z12.866`.................................Execute the third infeed operation. For this operation, the feed rate (F value), F0.04, specified for the first infeed operation is used since an F command is a modal command.

Execute drilling up to point D (Z12.866 mm).



`G00 Z-0.5 T00` ....Return the tool to point A (Z–0.5 mm)
Specify T00 to cancel the tool position compensation function.

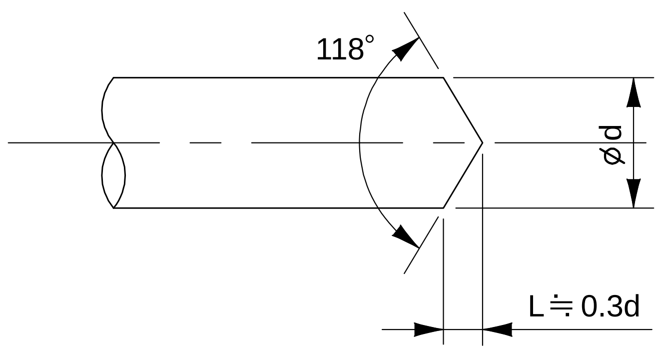


`(M141)` ....Move the opposite tool post backward. (only for some L series machines)

- Determine the drilling depth taking into consideration the drill tip shape. The dimension of drill tip can be calculated roughly using the expression of L ≈ 0.3 × d. For details, refer to the catalogs supplied by tool manufacturers.

L: Drill tip dimension

d: Drill diameter



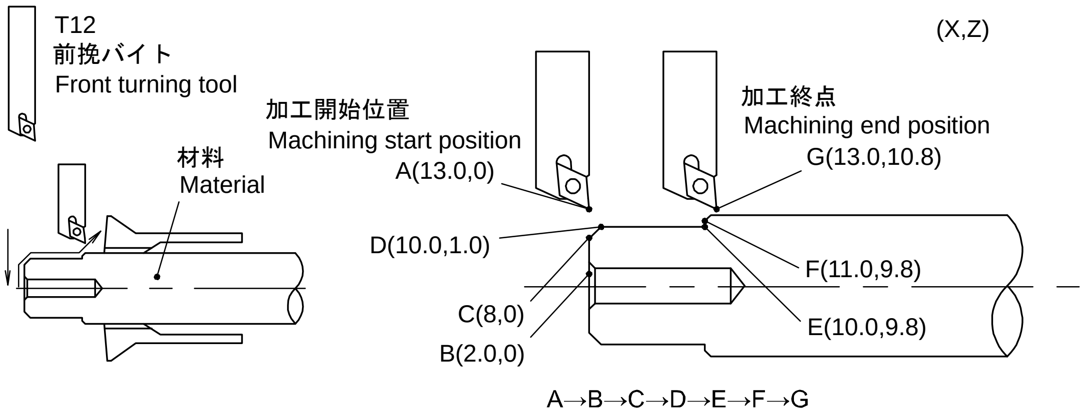
7.6 Front Turning

Preparation	Center Drilling	Drilling	**Front Turning**	Grooving	Back Turning	Cut-off End

Key Point

In front turning, a tool that has the cutting edge facing the direction the tool moves is used to perform outer diameter turning operation. Create the program meeting the shape to be machined.

Sample



Preparation	Center Drilling	Drilling	**Front Turning**	Grooving	Back Turning	Cut-off End

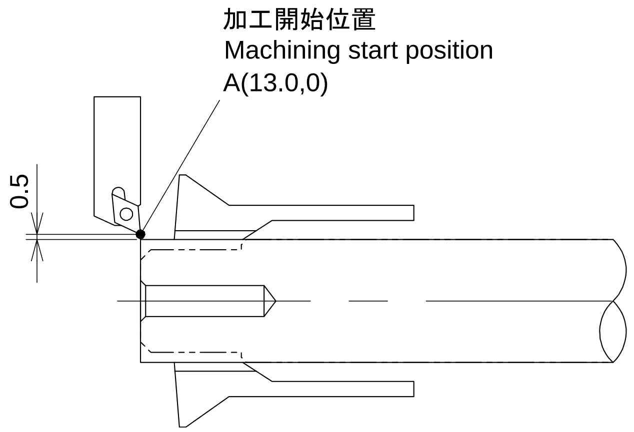
Program

`N0312`..Enter the sequence number.

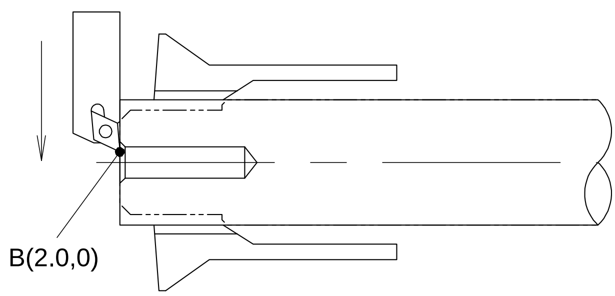
`M03 S3100 (M03 S1=3100)`........Start the front spindle in the forward direction.

`T1200`..Select the front turning tool T1200.

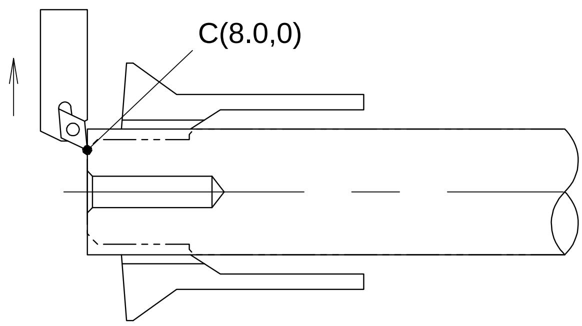
`G00 X13.0 Z0 T12`........................Execute positioning to X13.0, Z0, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate. Specify the tool position compensation command T21.
The commands move the tool to Point A where machining starts (approach point).



`G01 X2.0 F0.015`..........................Execute cutting up to X2.0.
The commands execute front turning to Point B to finish the end face.

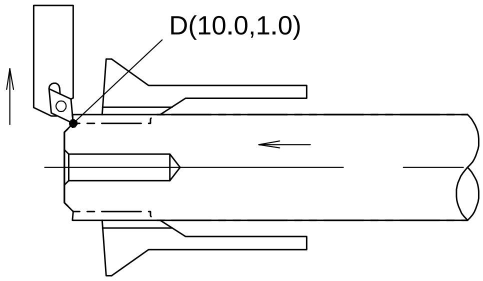


`G01 X8.0 F0.2`..............................Move the tool to X8.0 at a cutting feed rate.
The commands move the tool to Point C where chamfering starts. In this tool feed operation, the tool moves at a fast feed rate F0.2 since the material is not cut.

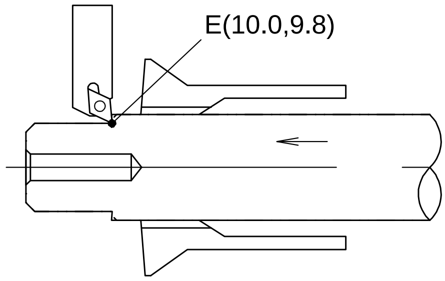


Preparation	Center Drilling	Drilling	**Front Turning**	Grooving	Back Turning	Cut-off End

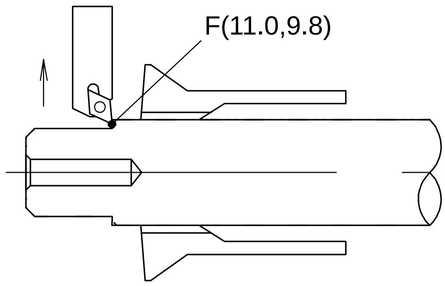
G01 X10.0 Z1.0 F0.015.............Execute cutting up to X10.0, Z1.0. The commands move the tool to Point D to chamfer the edge to C1.0.



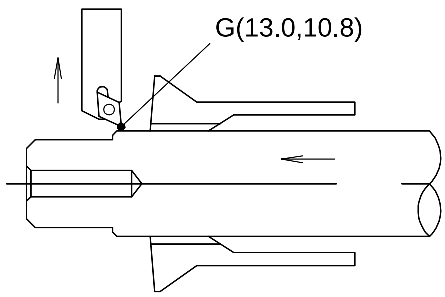
G01 Z9.8 F0.03............................Execute cutting up to Z9.8. The commands move the tool to Point E, leaving allowance 0.2 mm to be finished in the grooving process.



G01 X11.0 F0.05..........................Execute cutting up to X11.0. The commands move the tool to Point F where chamfering starts.



G01 X13.0 Z10.8 F0.015...........Execute cutting up to Z13.0, Z10.8, leaving the allowance to be finished n the next grooving process. The commands move the tool to Point G to chamfer the edge to C0.3 and further to move it away from the material.



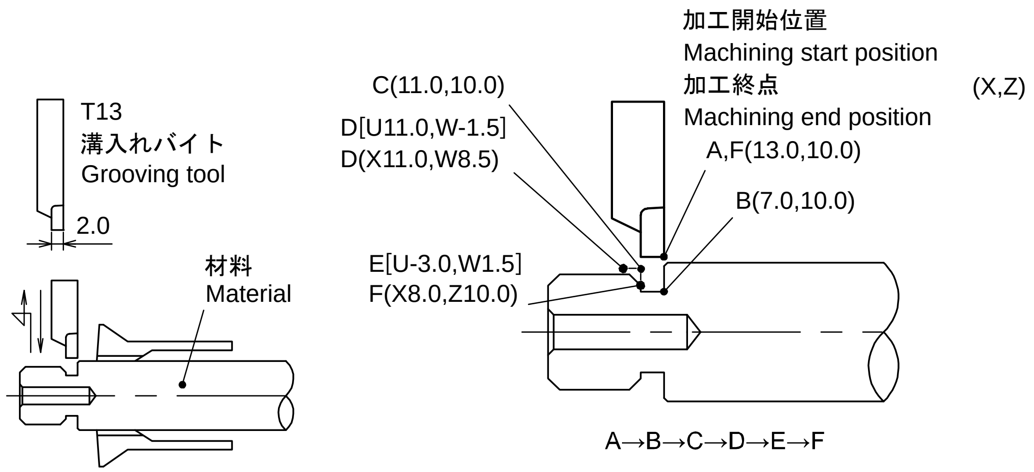
7.7 Grooving

Preparation	Center Drilling	Drilling	Front Turning	**Grooving**	Back Turning	Cut-off End

Key Point

A groove is machined. Generally, grooving is executed after finishing outer diameter machining (front/back turning).

Sample



[□□] indicates a relative value command.

(□□) indicates an absolute value command.

Preparation	Center Drilling	Drilling	Front Turning	**Grooving**	Back Turning	Cut-off End

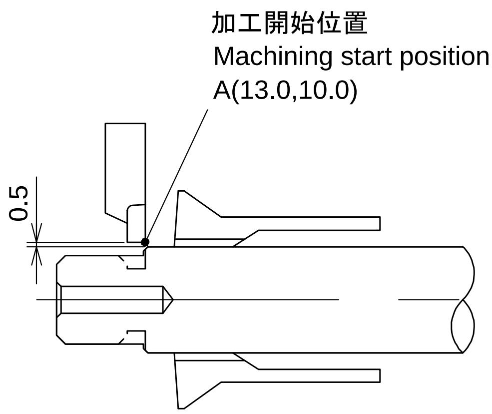
Program

N0413..Enter the sequence number.

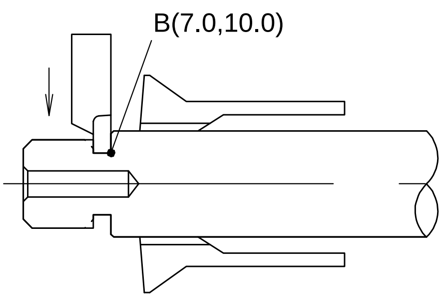
M03 S4500 (M03 S1=4500) ....Start the front spindle in the forward direction.

T1300..Select the grooving tool T1300.

G00 X13.0 Z10.0 T13................Execute positioning to X13.0, Z10.0, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate. Specify the tool position compensation command T13. The commands move the tool to Point A where machining starts (approach point).

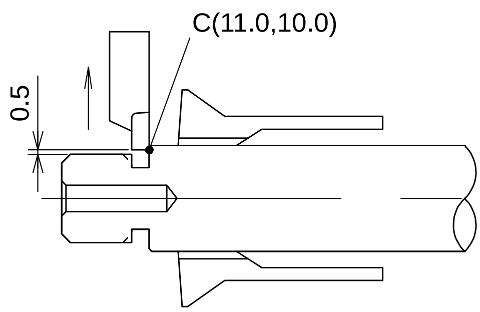


G01 X7.0 F0.015..........................Execute cutting up to X7.0. The commands execute grooving to Point B.



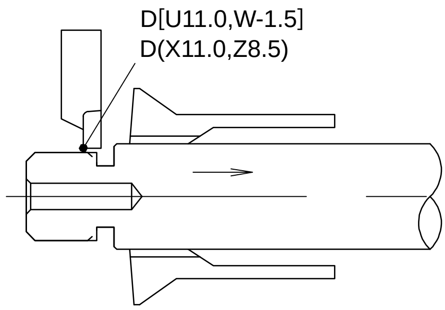
G04 U0.2..Specify the dwell function to stabilize the diameter of groove.

G01 X11.0 F0.2............................Return the tool to X11.0 at a cutting feed rate. The commands move the tool to Point C where chamfering starts.

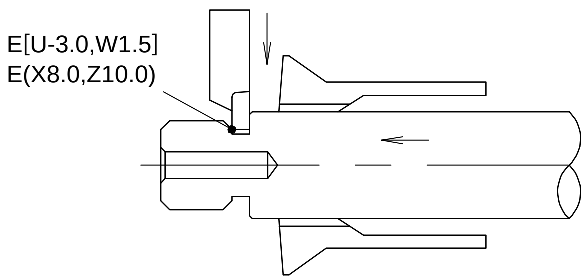


Preparation	Center Drilling	Drilling	Front Turning	Grooving	Back Turning	Cut-off End

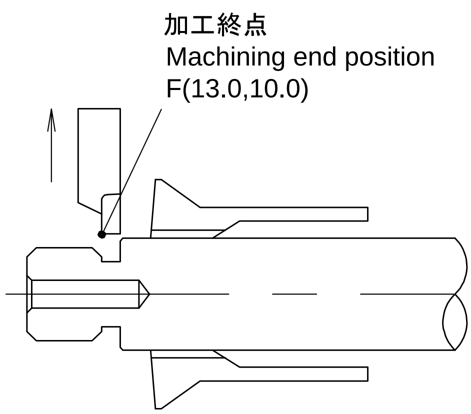
G00 W－1.5 (G00 Z8.5) .... Move the tool from X11.0, Z10.0 (Point C) by –1.5 mm (or up to Z8.5) along the Z axis at a rapid feed rate. In this programming, although both incremental and absolute value commands can be used, incremental programming will facilitate calculation.
The commands move the tool along the Z axis to point D where chamfering starts. The positioning point is determined with the grooving tool width 2 mm taken into account since chamfering is executed using the left cutting edge of the grooving tool.



G01 U－3.0 W1.5 F0.015 .... Execute chamfering by moving the tool from X11.0, Z8.5 by –3.0mm along the X axis and 1.5 mm along the Z axis (or up to X8.0, Z10.0).
(G01 X8.0 Z10.0 F0.015) The commands move the tool to Point E to chamfer the edge.



G01 X13.0 F0.2 .... Move the tool to X13.0. The commands move the tool to Point F to retract the tool nose from the material.

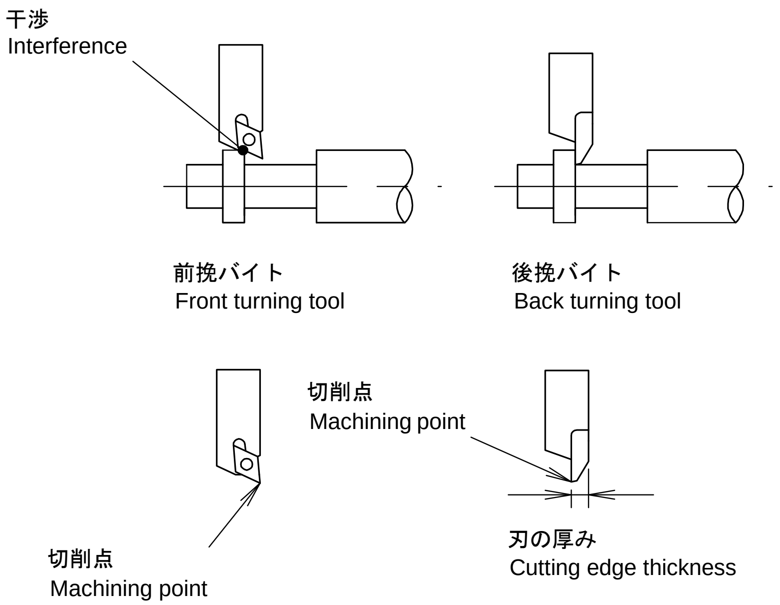


7.8 Back Turning

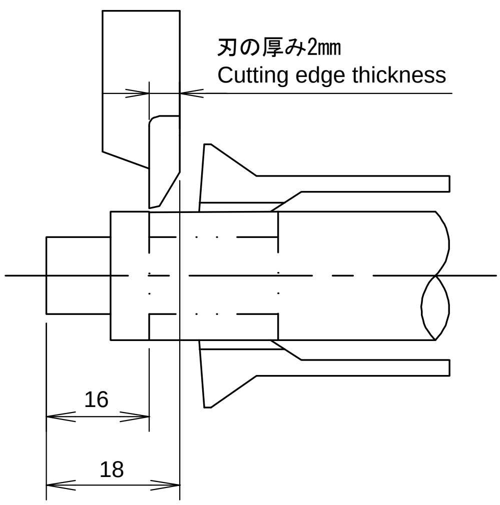
Preparation	Center Drilling	Drilling	Front Turning	Grooving	**Back Turning**	Cut-off End

Key Point

A back turning tool has the cutting edge at the opposite side of that of a front turning tool and can carry out machining at the back side of a workpiece, or the section beyond the collar.

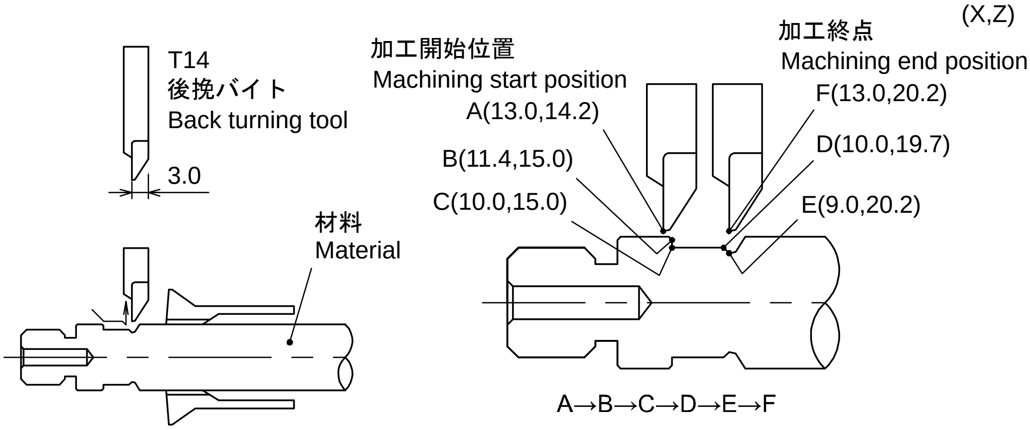


The cutting point of a back turning tool is shifted left by the cutting edge thickness from the cutting point of a front turning tool. Accordingly, if the cutting edge is 2 mm thick, the tool must be moved 2 mm more than the dimension specified on a drawing. To position the tool nose at a position 16.0 mm from the end face, for example, the Z coordinate to be specified in a program must be Z18.0. The function to shift the coordinate system is used in actual programming. For details of the coordinate system shift, refer to 8.5.



Preparation	Center Drilling	Drilling	Front Turning	Grooving	**Back Turning**	Cut-off End

Sample



Program

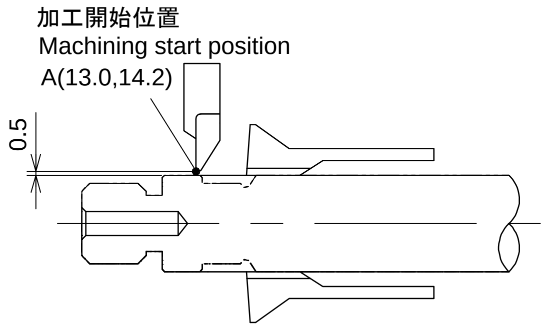
`N0514`..Enter the sequence number.

`M03 S3100 (M03 S1=3100)`........Start the front spindle in the forward direction.

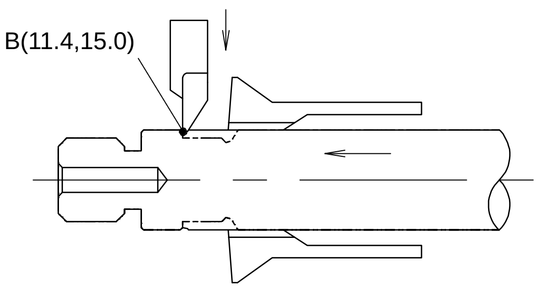
`T1400`..Select the back turning tool T1400.

`G50 W-3.0`....................................Specify the coordinate system shift (Z axis direction) command for the back turning tools.

`G00 X13.0 Z14.2 T14`..................Execute positioning to X13.0, Z14.2, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate. Specify the tool position compensation command T14. The commands move the tool to Point A where machining starts (approach point).

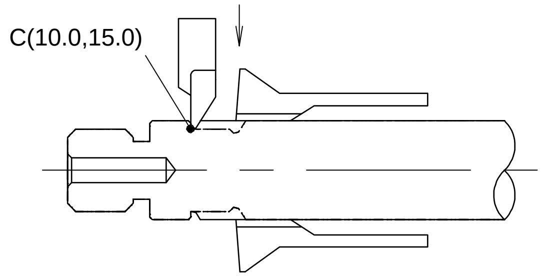


`G01 X11.4 Z15.0 F0.015`............Execute cutting up to X11.4, Z15.0. The commands move the tool to Point B to chamfer the edge.

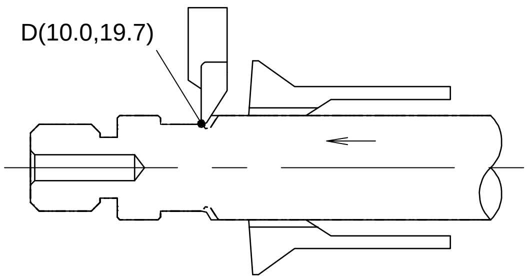


Preparation	Center Drilling	Drilling	Front Turning	Grooving	Back Turning	**Cut-off End**

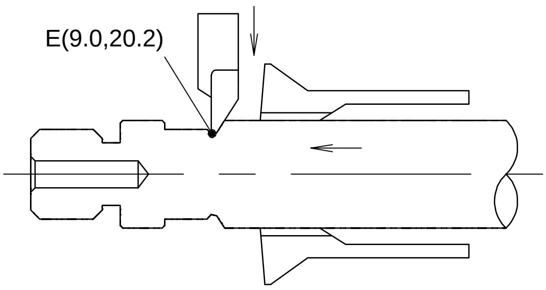
G01 X10.0 F0.015........................Execute cutting up to X10.0. The commands move the tool to Point C to execute outer diameter turning.



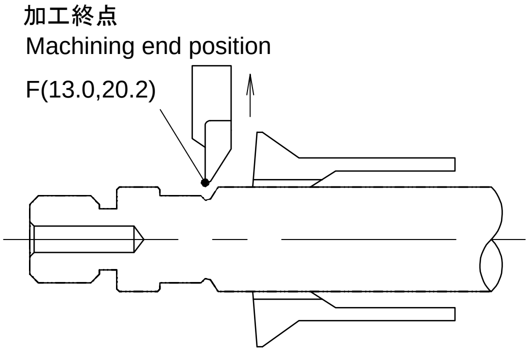
G01 Z19.7 F0.015........................Execute cutting up to Z19.7. The commands execute turning along the Z axis to Point D where chamfering starts.



G01 X9.0 Z20.2 F0.015.............Execute cutting up to X9.0, Z20.2. The commands execute cutting to Point E to chamfer the edge to C0.3 and further by 0.2 mm. This avoids a small part to be left uncut at the area where cut-off and chamfering overlap.



G00 X13.0 T00............................Move the tool to X13.0. The commands move the tool to Point F to retract the tool nose from the material. Specify T00 to cancel the tool position compensation function.



G50 W3.0.......................................Cancel the coordinate system shift (Z axis direction) for back turning tools.

7.9 Cut-off End Process

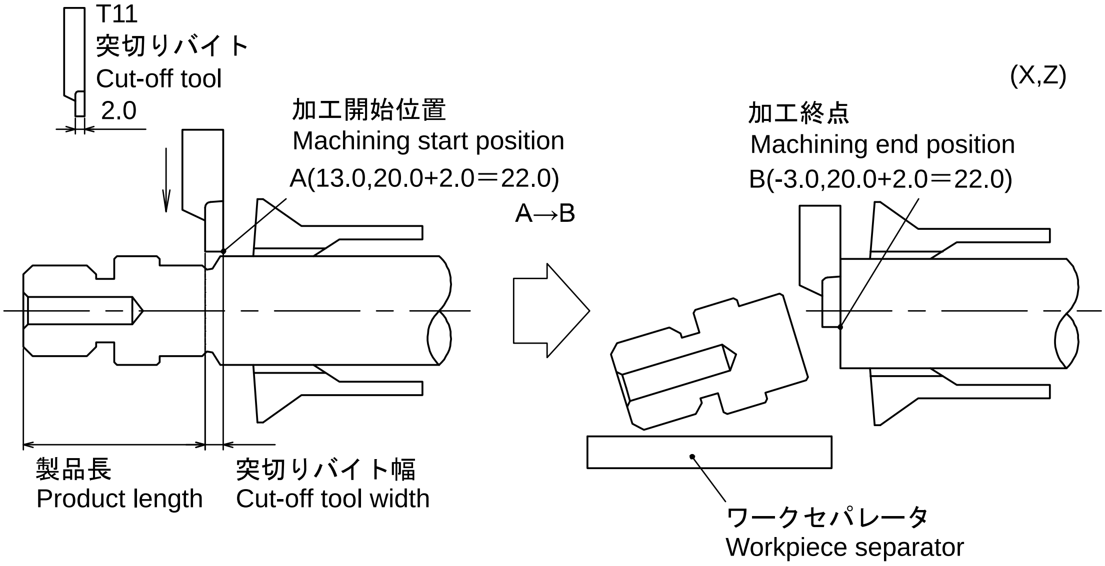
Preparation	Center Drilling	Drilling	Front Turning	Grooving	Back Turning	**Cut-off End**

Key Point

In the cut-off end process, set the cut-off commands and the commands required for ending the 1-cycle operation.

The cut-off machining separates the finished product from the material.

Sample



Program

N0611..Enter the sequence number.
M03 S3100 (M03 S1=3100)........Start the front spindle in the forward direction.
T1100..Select the cut-off tool T1100.
G00 X13.0 Z22.0 T11..................Execute positioning of the cut-off tool to cut off the product in the correct length (Product length + Cut-off tool width; Z20.0+2.0 = Z22.0). Specify the tool position compensation command T11. The commands move the tool to Point A where machining starts (approach point).
M32...The workpiece separator moves forward. The program for product collection may differ from this depending on the specification of the machine type to be used. Refer to the Instruction Manual provided with the machine.
G01 X-3.0 F0.015.........................Move the tool up to X-3.0. The commands execute cut-off machining by feeding the tool to Point B.
M33...The workpiece separator moves backward. The program for product collection may differ from this depending on the specification of the machine type to be used. Refer to the Instruction Manual provided with the machine.
M05...Stop the front spindle.
M07...Open the front spindle chuck.

Preparation	Center Drilling	Drilling	Front Turning	Grooving	**Back Turning**	Cut-off End

G04 U0.2..Dwell time

G00 X-3.0 Z0 T00........................Return the X and Z axes to the start position at a rapid feed rate. The Z coordinate must be the same value specified with G50 Z□□ in the preparation process at the beginning of programming. For the X coordinate, specify the same value set at “Cut-off End (DIA)” in the machining data.
Since the tool position compensation function must be canceled at the end of a program, specify T00 here.

M56..Count the number of products. The number of products is counted only during continuous operation.

M02..One-cycle stop is specified. Control returns to the beginning of the program.

M99..Program return. Control returns to the beginning of the program. (There is no need to enter M99 depending on the machine type.)

%..This is a stop code. The stop code is automatically entered to a program when it is created on the NC unit. The stop code is used when inputting or outputting programs.

Product Code	C	–	C	I	N	C	O	M	_	I	G	B			
Document Code	4	E	1	–	0	7	0	0							

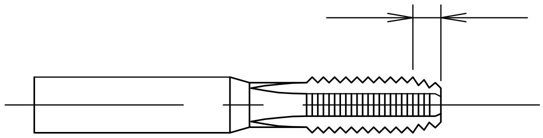
8. Actual Programming

Code No.	C-CINCOM_IGB 4E1-0800	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

8.1 Thread Cutting with a Tap

A tap is used to cut internal threads. The front end of a tap has a chamfered section so that it can engage with the material to be cut, and this chamfered section leaves incompletely threaded part in the material. Thread cutting with a tap is carried out after drilling a preparatory hole.



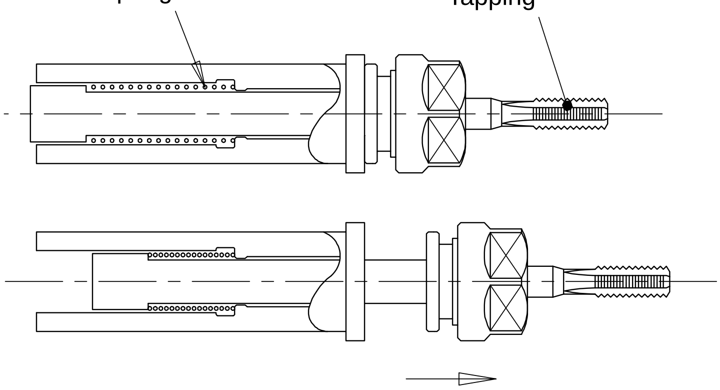
Thread cutting with a tap is largely classified into two types such as rigid tapping and floating tapping.

- Rigid tapping

In this type of tapping, the feature that the NC unit of the lathe completely synchronizes the feed of a tap with the rotational phase of a tap. The tap is held in a rigid sleeve like a drill.

- Floating tapping

Differing from rigid tapping, a dedicated tap sleeve with the floating function is used instead of a rigid sleeve so that threads can be cut with a tap even when the axis feed and tap rotation are not synchronized completely. A tap sleeve with the floating function has the mechanism that allows the sleeve itself to float in the longitudinal direction and also a key to prevent the sleeve from rotating together with a tap. This structure absorbs control errors to allow thread cutting even if axis feed and tap rotation is not controlled completely.



- The distance from the crest of one thread to the next is called pitch.
- The distance a tap advances in one turn of threads is called lead and with 1-flute threads (metric threads), a pitch equals a lead. With 2-flute threads, however, a lead is two times a pitch.
- Thread cutting start position with a tap must be set to provide the distance more than two times the lead to the material along the Z axis. This distance is required for the tap to reach the specified feed rate.
- Generally, the spindle speed is 300 to 1000 min^{-1} for the tapping. See <Chapter 9　Cutting Conditions>
- Drill the preparatory hole for tapping to the depth greater than the total of the length equivalent to the nominal tap diameter and the tapping length.
- If the nominal size of the thread is M4, for example, the depth of the preparatory hole is “thread cutting depth + 4 mm”.

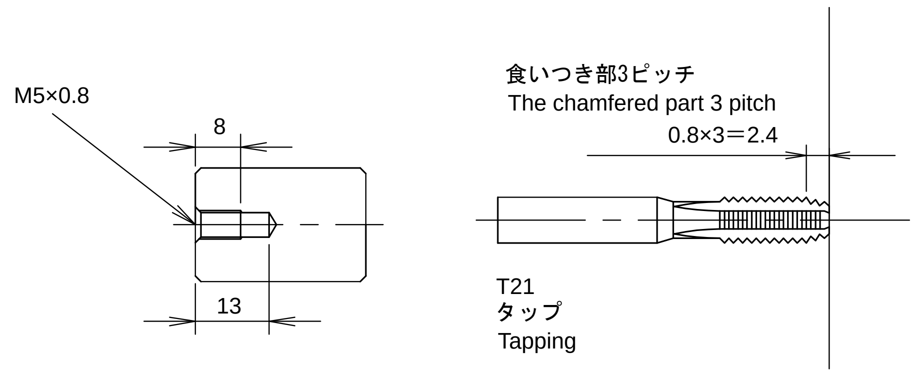
Command format

G32 X☐ Z☐ F☐.......Thread cutting is executed while the NC synchronizes spindle speed with axis feed. The commands cut threads to X☐ Z☐ from the current position at the feed rate F☐.

Program sample

The following explains the program sample that cuts threads with a tap using a floating type tap sleeve.

This sample assumes that the preparatory hole has been drilled.



N0521...Enter the sequence number.

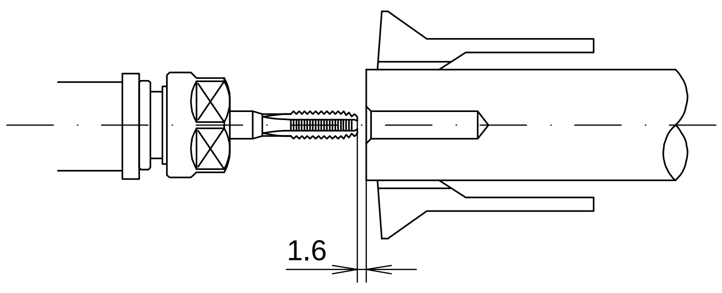
M03 S＝300...................................The spindle rotates forward.

T2100...Select the T21 tap.

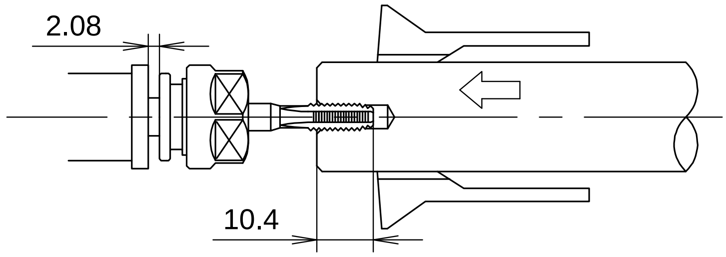
G04 U0.5.......................................Dwell time 0.5 second is entered for stabilizing the spindle speed.

G00 Z−1.6 ....The tool is 1.6 mm (0.8 × 2) away in the longitudinal direction.

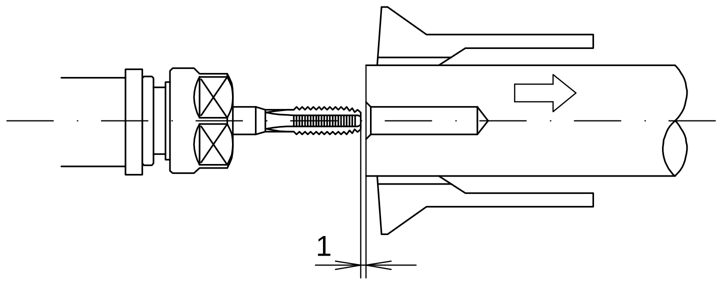
(M140) ....Move the opposite tool post forward. (only for some L series machines; specification of this M code is not necessary since tapping is usually executed after drilling)



G32 Z8.32 F0.64 T21..................Feed rate is set at a value 0.8 to 0.95 times the lead. In this sample program, 0.64 mm/rev (= 0.8 × 0.8) is selected. The depth of tapping is calculated by adding 3 leads (0.8 × 3 = 2.4 mm) at the chamfered part to the depth specified in the drawing; 8.0 + 2.4 = 10.4 mm. The depth to be specified in the program is also set at a value 0.8 to 0.95 times the calculated depth. In this sample program, 8.32 mm (= 10.4 × 0.8) is selected. Specify the tool position compensation command T21.



Z-1.0 F0.8 M04 T00....................Return the tool at the feed rate equivalent to the lead by rotating the spindle in the reverse direction. Specify T00 to cancel the tool position compensation function.



G04 U2.0..Dwell time is entered for the tap holder to wait until the tap is extracted completely. Generally, specify about 2.0 seconds as the dwell time.

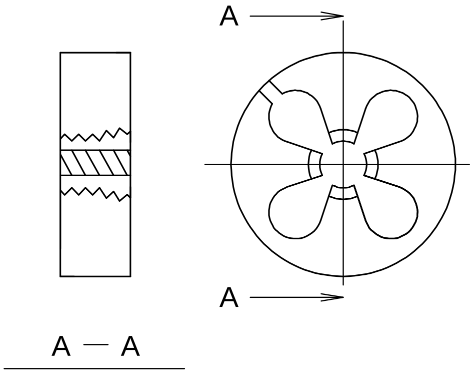
M03 S3100 (M03 S1=3100) ....Do not forget to rotate the spindle forward when starting the next machining process.

G04 U1.0..Enter dwell time for stabilizing the spindle speed.

(M141) ....Move the opposite tool post backward. (only for some L series machines)

8.2 Thread Cutting with a Die

A die is used to cut external threads. The front end of a die is chamfered so that it can engage with the material to be cut, and this chamfered portion leaves incompletely threaded part in the material. A die must be first adjusted to the outer diameter of the threads to be cut. Compared with tread cutting with a threading tool, thread cutting with a die is subject to high cutting resistance. This is because a die machines the height of thread ridge at a time. Also, a thread machined by a die may have a torn surface.

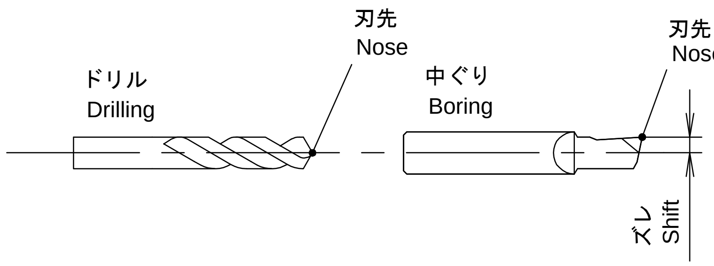


Thread cutting with a die can be programmed in the same manner as that with a tap. For details of programming, refer to the section explaining thread cutting with a tap.

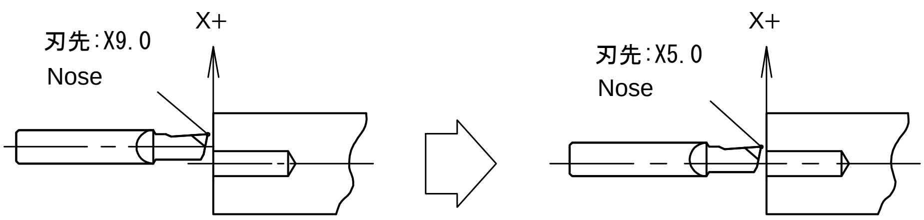
8.3 Boring

Boring process expands the inner diameter of a machined hole to the diameter specified on a drawing.

The tool nose of a boring tool is shifted in the X+ side in reference to the tool nose of a drill.



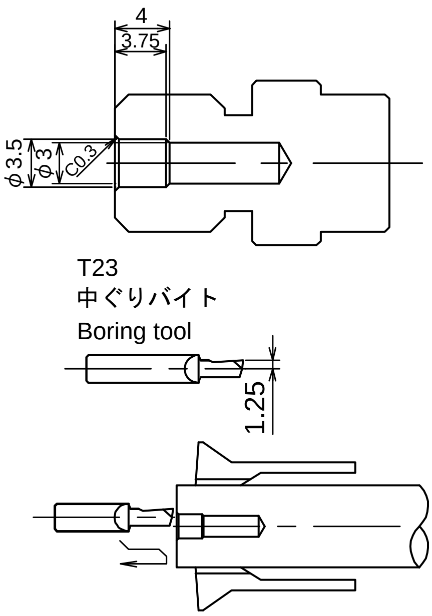
When a boring tool having the shift amount of 2.0 mm is used, for example, it is necessary to calculate the positioning point taking into consideration the shift amount; to position this boring tool at X5.0, the X coordinate to be specified is X1.0, which is calculated by subtracting 2.0 mm × 2 (diametric command) from 5.0 mm. To simplify the programming, the coordinate system shift function can be used. For details of the coordinate system function, refer to <8.5 Coordinate System Shift>.

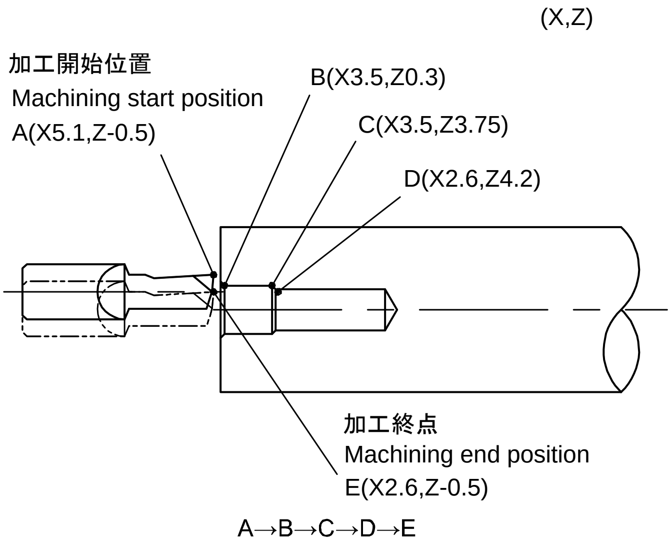


When X5.0 is specified
When X1.0 is specified

- Use the boring tool meeting the inner diameter to be machined.
  Usually, a tool smaller than the diameter of the machined prepared hole is used.
- Mount a boring tool by setting the tool nose in the X+ direction.

Sample





Program

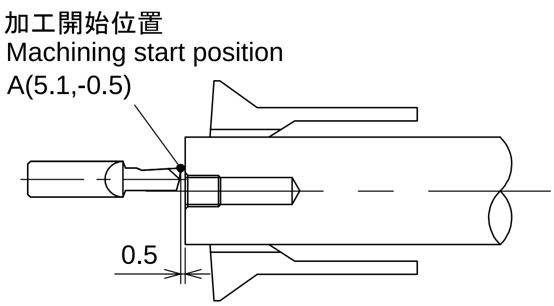
N0523... Enter the sequence number.

M03 S2700 (M03 S1=2700).......... Start the spindle in the forward direction.

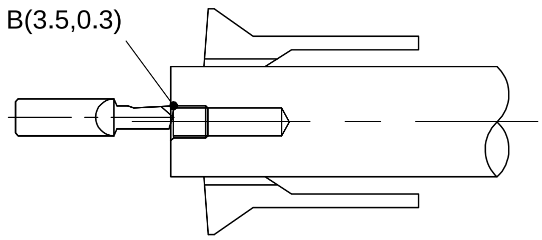
T2300... Select the boring tool T2300.

S50 U2.5....................................... Specify the coordinate system shift (X axis direction) command for the boring tool.

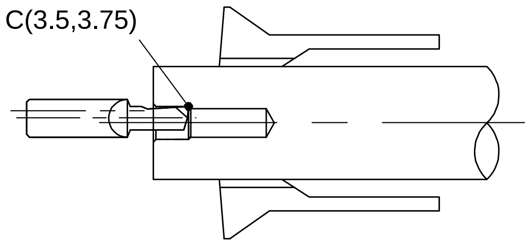
G00 X5.1 Z-0.5 T23................... Execute positioning to X5.1, Z–0.5, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate. Determine the X coordinate of the positioning point taking into consideration the chamfering to be executed next. The commands move the tool to Point A where machining starts (approach point).



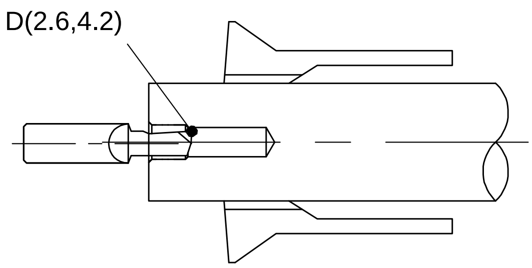
G01 X3.5 Z0.3 F0.01..................Execute cutting up to X3.5, Z0.3. The commands move the tool to Point B to chamfer the edge to C0.3.



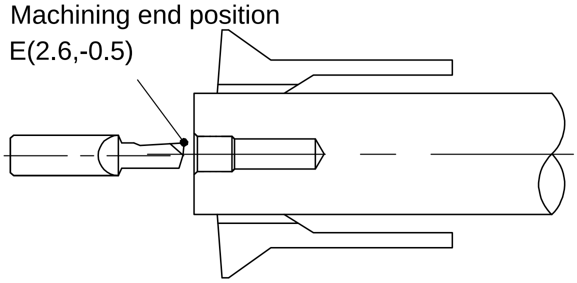
G01 Z3.75 F0.015........................Execute cutting up to Z3.75. The commands move the tool to Point C where chamfering starts.



G01 X2.6 Z4.2 F0.01..................Execute cutting up to X2.6, Z4.2. The commands move the tool to Point D where the tool nose is apart from the material while executing chamfering.



G01 Z-0.5 F0.5 T00..................Execute positioning to Z–0.5 at a rapid feed rate. The commands move the tool to the position where the boring tool exits the material.

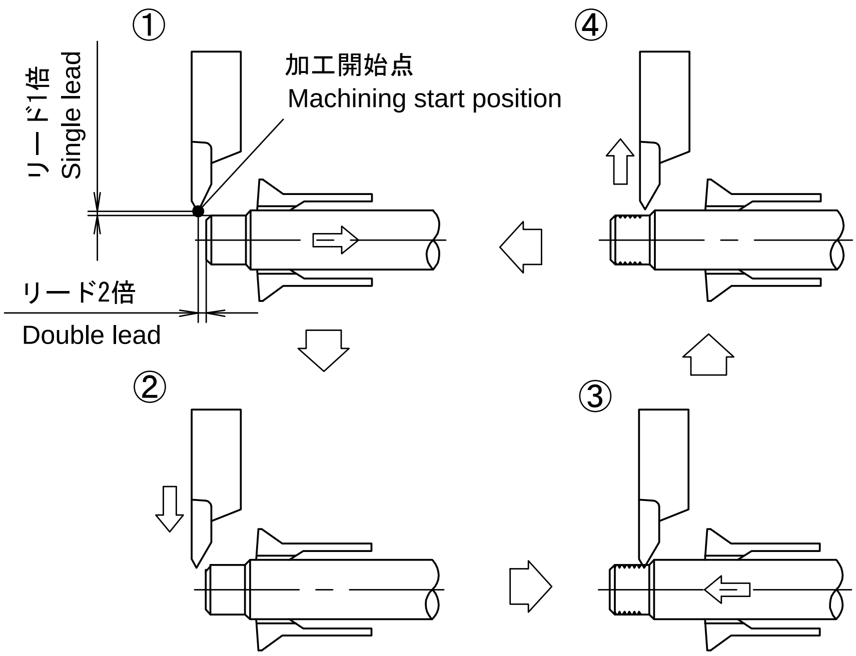


G50 U-2.5.....................................Cancel the coordinate system shift (X axis direction) for the boring tools.

8.4 Thread Cutting

In thread cutting process, external and internal threads are machined with a thread cutting tool. A thread cutting tool has the same profile as the root of thread shape. The figure below shows the sequence external threads are cut.

Steps (1) – (4) are repeated to cut thread.



(1) Start position of thread cutting → (2) Positioning at a rapid feed rate (X axis direction) → (3) Thread cutting → (4) Tool retraction while cutting thread → (1) Return to the start position of thread cutting at a rapid feed rate

- The distance from the crest of one thread to the next is called pitch.
- The distance a tap advances in one turn of threads is called lead and with 1-flute threads (metric threads), a pitch equals a lead. With 2-flute threads, however, a lead is two times a pitch.
- Generally, the spindle speed is 500 to 2000 min^{-1} for the thread cutting.
- The start position of thread cutting is determined by providing the clearance to the material by more than double-lead distance in the Z axis direction and single-lead distance in the X axis direction. To shorten the distance in the Z axis direction, selecting a lower spindle speed sometimes makes it possible.
- If the tool nose of a thread cutting tool is shifted, use the coordinate system shift function.
- For details of the coordinate system shift function, refer to <8.5 Coordinate System Shift>.
- The spindle speed N for thread cutting is limited as shown below.

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

N: Spindle speed in thread cutting

L: Thread lead

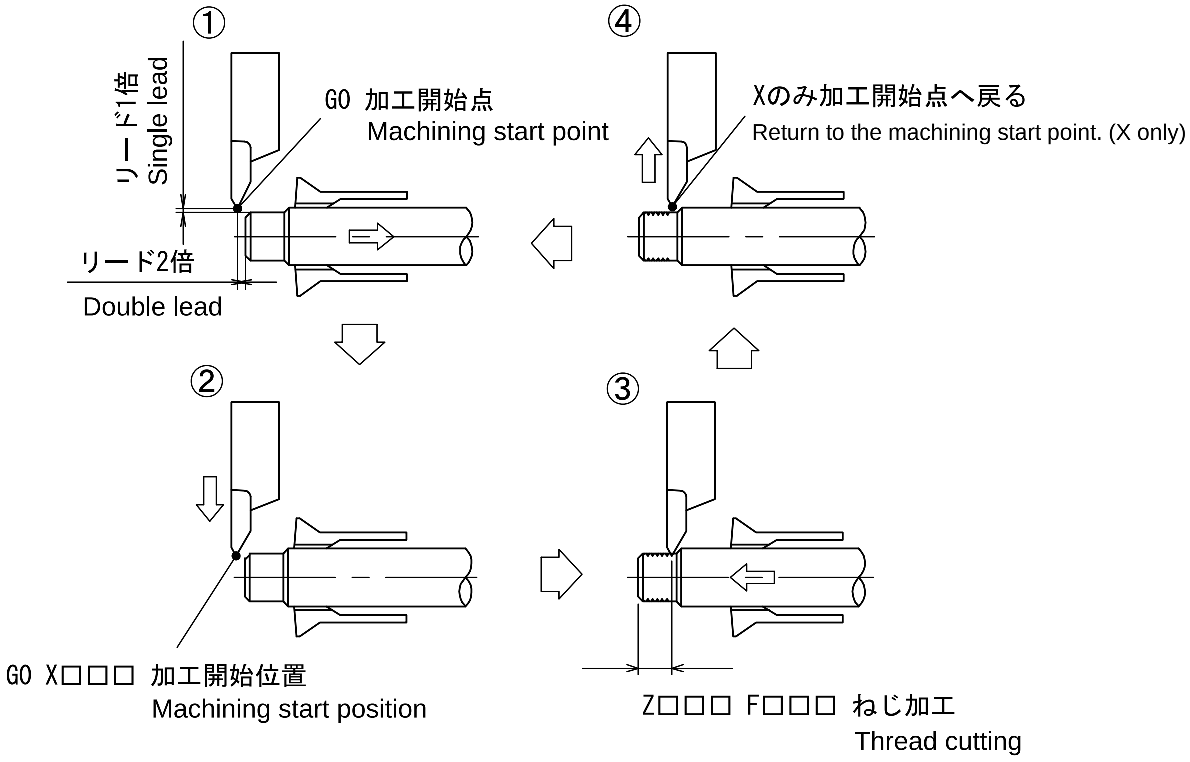
Fmax: Maximum feed rate of the machine

Command format

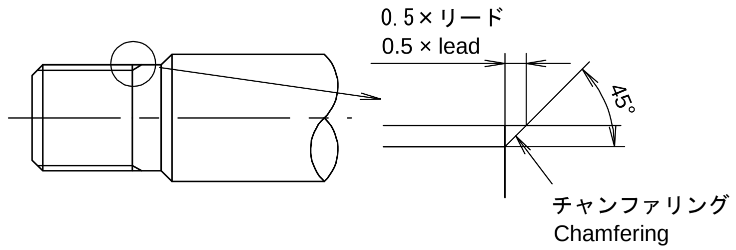
G92 Thread cutting canned cycle

G92 X□ Z□ F□.......A series of thread cutting operation (cycle), from (1) to (4) in the figure below, can be specified by one block (line) of commands.

X□ : Infeed position
Z□ : Thread cutting end point
F□ : Feed rate (thread lead)



- The chamfer at the thread tail is machined in the shape specified by the parameter. Usually, the length and angle of chamfer is set as 0.5 × lead and 45°. For details, refer to the <Instruction Manual provided by the NC unit manufacturer>.



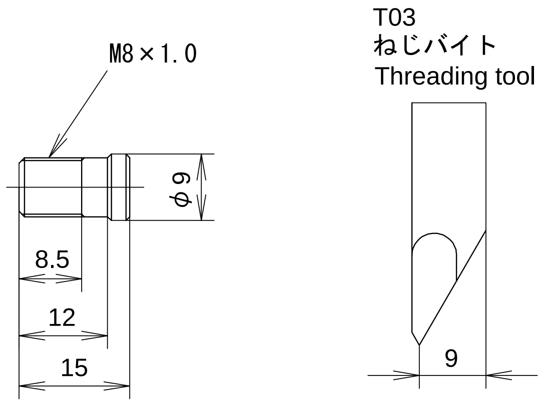
- Value of Z□ is the distance including the chamfer at the thread tail.

8.4.1 Parallel thread cutting

External thread cutting using G92 (thread cutting canned cycle)

Program sample

The program is explained assuming the front turning has been completed.



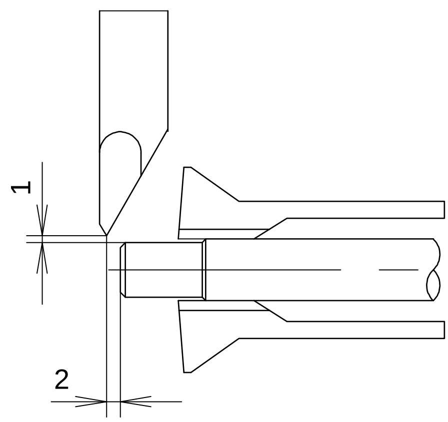
N0503 .... Enter the sequence number.

M03 S1500 (M03 S1=1500) .... Start the spindle in the forward direction.

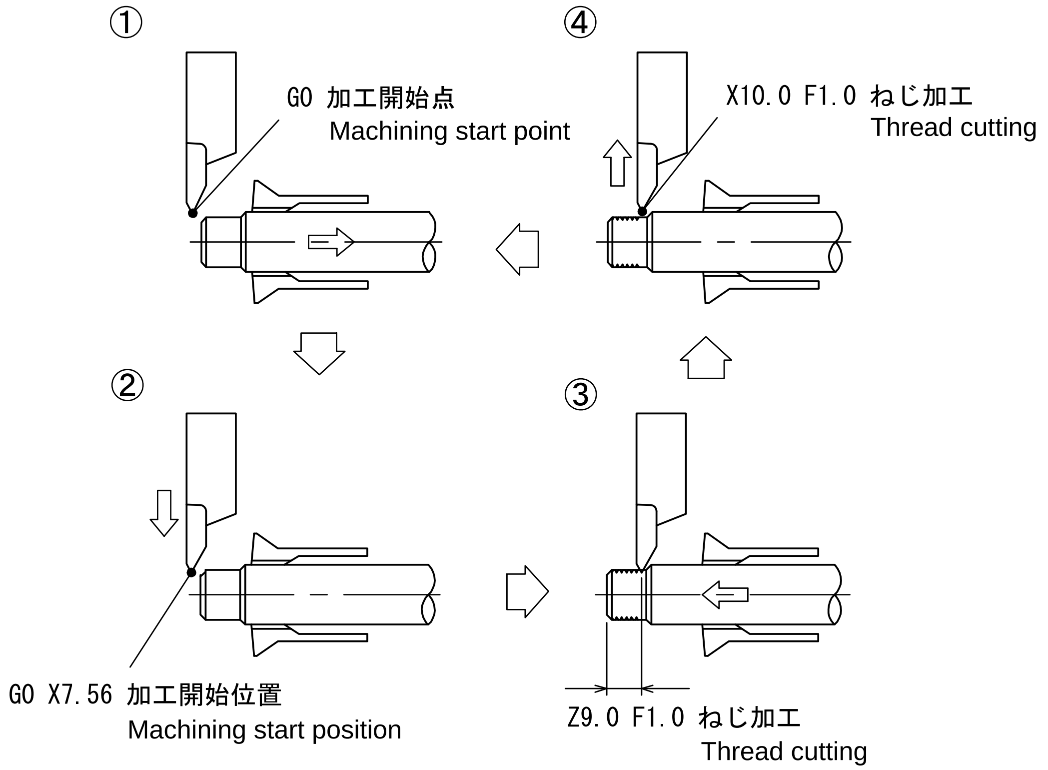
T0300 .... Select the thread cutting tool T0300.

G50 W-9.0 .... Specify the coordinate system shift (Z axis direction) command for the thread cutting tool.

G00 X10.0 Z-2.0 T03 .... Execute positioning to the thread cutting start position at a rapid feed rate. Allow the clearance to the material by more than double-lead distance in the Z axis direction and single-lead distance in the X axis direction.



G92 X7.56 Z9.0 F1.0..................First infeed
Calculate the X value as explained in <9.3 Thread Cutting Count>.
X = 8.0－(0.22 × 2) = 7.56
For the Z value, include the chamfer (0.5 × lead).
Z = 8.5 + (1.0 × 0.5) = 9.0
F is equivalent to lead.
With these commands, one cycle of thread cutting from (1) to (4) is executed.



X7.26...Second infeed
G92, Z9.0 and F1.0 are stored in the NC unit as modal commands and thread cutting cycle is repeated at the second infeed position by simply specifying the X value.
Calculate the X value as explained in <9.3 Thread Cutting Count>.
X = 7.56 – (0.15 × 2) = 7.26

X7.04...Third infeed.　X = 7.26 – (0.11 × 2) = 7.04

X6.86...Fourth infeed.　X=7.04 – (0.09 × 2) = 6.86

X6.76...Fifth infeed.　X=6.86 – (0.05 × 2) = 6.76

X6.7...Sixth infeed.　X=6.76 – (0.03 × 2) = 6.7

X6.7...Execute the thread cutting cycle at the same X position two times. This is called zero-cut.

G00 X10.0.....................................Return the tool to the thread cutting start position.

G50 W9.0.......................................Cancel the coordinate system shift (Z axis direction) for the thread cutting tool.

- Cutting resistance causes the workpiece to move away from a tool during the thread cutting process.　As a result, some parts remain uncut.　Zero-cut is a cutting method that minimizes uncut parts by cutting a workpiece with the same infeed amount twice.

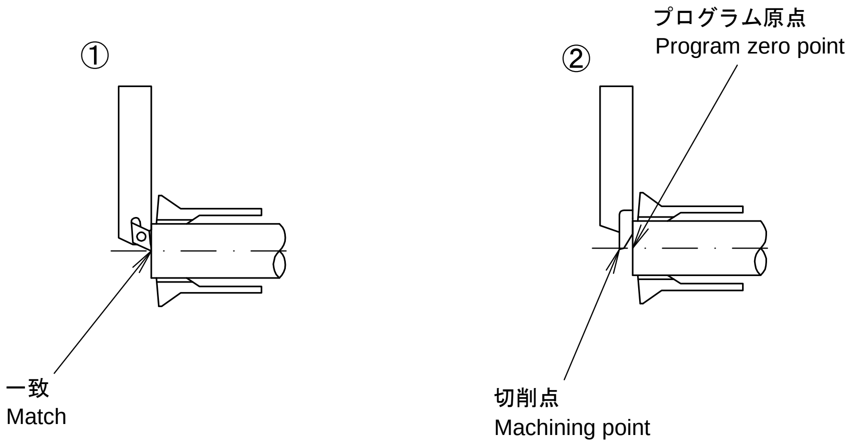
8.5 Coordinate System Shift

With a boring tool and a back turning tool, the cutting point is not aligned with the program zero point (center of rotation on the end face of a material) even when G0 X0 Z0 is specified. Such a state is called the cutting point is shifted and the distance of shift is referred to as the shift amount. If the cutting point is shifted, the dimensions specified on a drawing cannot be directly used in programming, making programming complex work. To solve such a problem, the feature called coordinate system shift is used, which allows programming using the dimensions specified on a drawing even with tools having shifted cutting point.

Command format

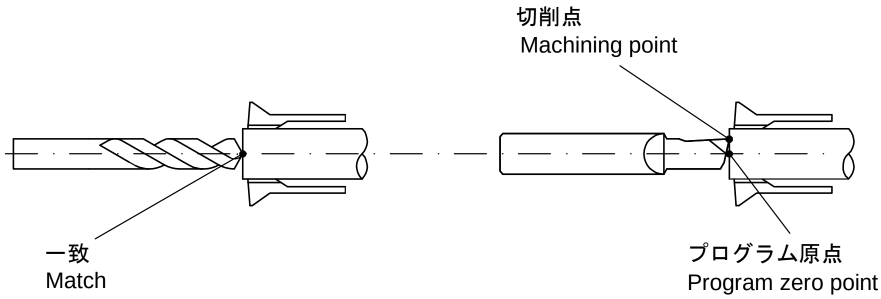
Coordinate shift in the Z axis direction

G50 W□□



Coordinate shift in the X axis direction

G50 U□□

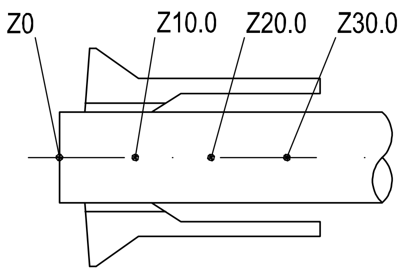


The commands G50 W□□ and G50 U□□ indicate “to add the specified value to the current coordinate system”. Specification of G50 W–20.0, for example, adds –20.0 mm to the Z coordinate values specified in the current coordinate system. The processing to change the coordinate system is called coordinate system shift.

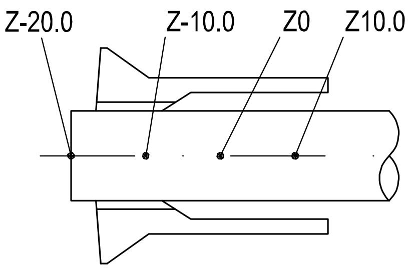
The G50 W□□ and G50 U□□ commands do not call material and tool movements when they are specified because they simply change the coordinate system. When an axis move command (G0, G1, etc.) is specified after the specification of G50, the axis move command is executed on the changed coordinate system.

Program

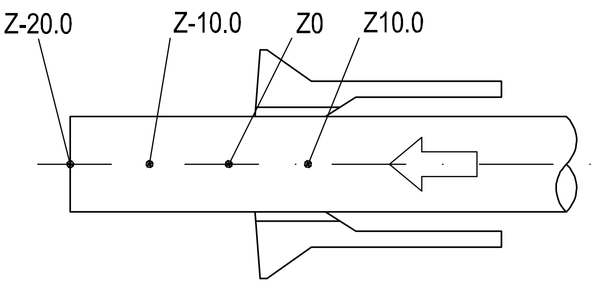
G00 Z0..Executes positioning to Z0.



G50 W-20.0....................................Shifts the coordinate system by –20.0 in the Z axis direction. The commands do not call axis movements.



G00 Z0..Executes positioning to Z0 in the new coordinate system. These commands move the material 20.0 mm in the Z+ direction.

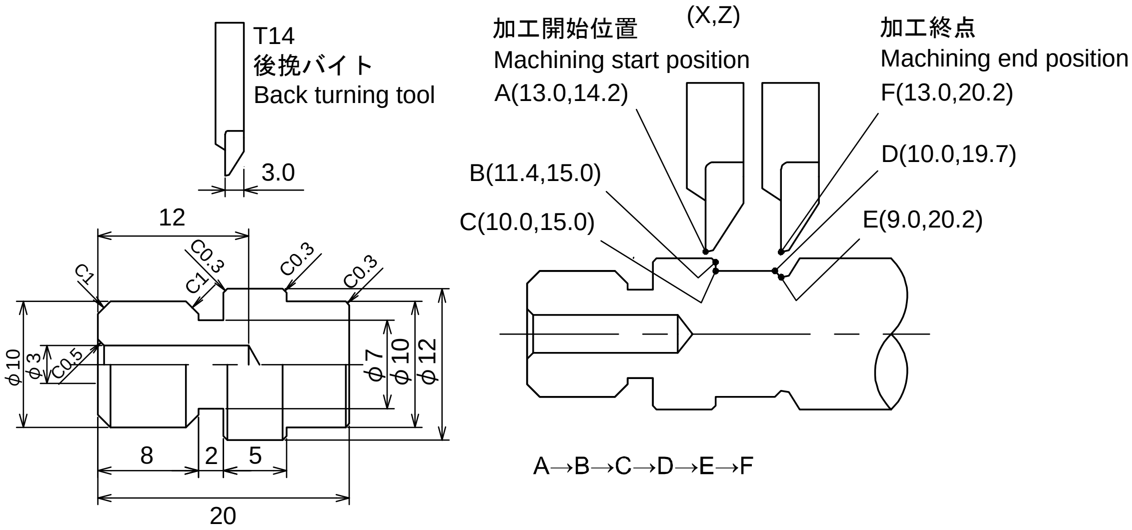


- After finishing machining on a shifted coordinate system, the coordinate system must be shifted again to cancel the shifted amount to restore the original coordinate system (coordinate system shift cancel).
- Do not use the coordinate system shift command for cut-off machining when the machine is equipped with a back spindle.
  If the command is used, the back spindle is shifted by the coordinate system shift amount during cut-off machining process.

8.5.1 Coordinate system shift in the Z axis direction

In back turning operation, the coordinate system should be shifted in the Z axis direction to simplify programming since the cutting point of a back turning tool is shifted in the Z axis direction.

Sample



Program

N0514..Enter the sequence number.

M03 S3100 (M03 S1=3100)........The spindle rotates forward.

T1400..Select the back turning tool T1400.

G50 W-3.0.....................................**Specify the coordinate system shift (Z axis direction) command for the back turning tools.**

G00 X13.0 Z14.2 T14.................Execute positioning to X13.0, Z14.2, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate.
The commands move the tool to Point A where machining starts (approach point).

G01 X11.4 Z15.0 F0.015...........Execute cutting up to X11.4, Z15.0.
The commands move the tool to Point B to chamfer the edge.

G01 X10.0 F0.015.......................Execute cutting up to X10.0. The commands move the tool to Point C to cut the material to the ø10.0 mm position.

G01 Z19.7 F0.03.......................Execute cutting up to Z19.7. The commands move the tool to Point D where chamfering starts.

G01 X9.0 Z20.2 F0.015.............Execute cutting up to X9.0, Z20.2. The commands executes cutting to Point E to chamfer the edge to C0.3 and further by 0.2 mm.

G00 X13.0....................................Execute positioning to Z13.0. The commands move the tool to Point F to move it away from the material.

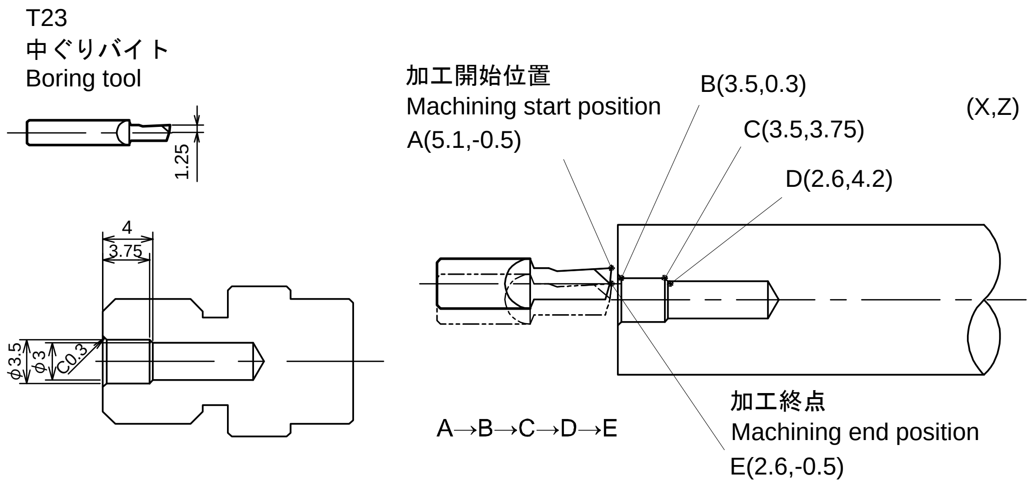
G50 W3.0......................................**Cancel the coordinate system shift (Z axis direction) for back turning tools.**

8.5.2 Coordinate system shift in the X axis direction

In boring operation, the coordinate system should be shifted in the X axis direction to simplify programming since the cutting point of a boring tool is shifted in the X axis direction.

- For the coordinate system shift command in the X axis direction, specify two times the cutting point shift amount since X axis command values are specified in diametric values.

Sample



Program

N0523 .... Enter the sequence number.

M03 S2700 (M03 S1=2700) .... Start the spindle in the forward direction.

T2300 .... Select the boring tool T2300.

S50 U2.5 .... **Specify the coordinate system shift (X axis direction) command for the boring tool.**

G00 X5.1 Z–0.5 T23 .... Execute positioning to X5.1, Z–0.5, where 0.5 mm clearance is left between the material and the tool nose, at a rapid feed rate. Calculate the X coordinate taking into consideration the next chamfering process.
The commands move the tool to Point A where machining starts (approach point).

G01 X3.5 Z0.3 F0.01 .... Execute cutting up to X3.5, Z0.3.
The commands move the tool to Point B to chamfer the edge to C0.3.

G01 Z3.75 F0.015 .... Execute cutting up to Z3.75. The commands move the tool to Point C where chamfering starts.

G01 X2.6 Z4.2 F0.01 .... Execute cutting up to X2.6, Z4.2. The commands move the tool to Point D where the tool nose is apart from the material while executing chamfering.

G01 Z–0.5 F0.5 .... Execute positioning to Z–0.5. The commands move the tool to Point E where the boring tool exits the material.

G50 U–2.5 .... **Cancel the coordinate system shift (X axis direction) for the boring tools.**

8.6 Subprograms

When a program includes fixed programs or programs that appear repeatedly, programming can be made simple by preparing such fixed programs and repeatedly used programs as subprograms beforehand.

Command format

NC unit produced by FANUC

$$\texttt{M98 P}\underbrace{\square\square\square\square}_{\text{Repetition count (Default: 1)}}\ \underbrace{\square\square\square\square}_{\text{Subprogram number}}$$

(With 10T or R04)

$$\texttt{M98 P}\underbrace{\square\square\square\square}_{\text{Subprogram number}}\ \texttt{L}\underbrace{\square\square\square}_{\text{Repetition count (Default: 1)}}$$

NC unit produced by MITSUBISHI Electric

$$\texttt{M98 P}\underbrace{\square\square\square\square}_{\text{Subprogram number (Default: Calling-program number)}}\ \texttt{H}\underbrace{\square\square\square\square}_{\text{Sequence number in the subprogram to be called (Default: First-sequence number)}}\ \texttt{L}\underbrace{\square\square\square\square}_{\text{Repetition count (Default: 1)}}$$

- The call count is 1 when the repetition count is omitted.
- Once a subprogram is called, it can be repeated up to 9,999 times. (Note 1)
- The number of nesting levels for calling subprograms from the main program varies depending on the device type.
- The subprogram call command can be used with any axis control group of the main program.
- Subprograms are called from each axis control group of the main program. Consequently, subprograms called from the axis control $1 of the main program, work as programs for the $1 axis control group.
- M99 must always be specified at the end of a subprogram.
- At the end of a subprogram, M02 must not be specified.

Program

Main program

```
O0018

M98 P0200
G00 X11.0 Z15.0

M99
M02
%
```

Calling the subprogram of program number O0200.

Subprogram

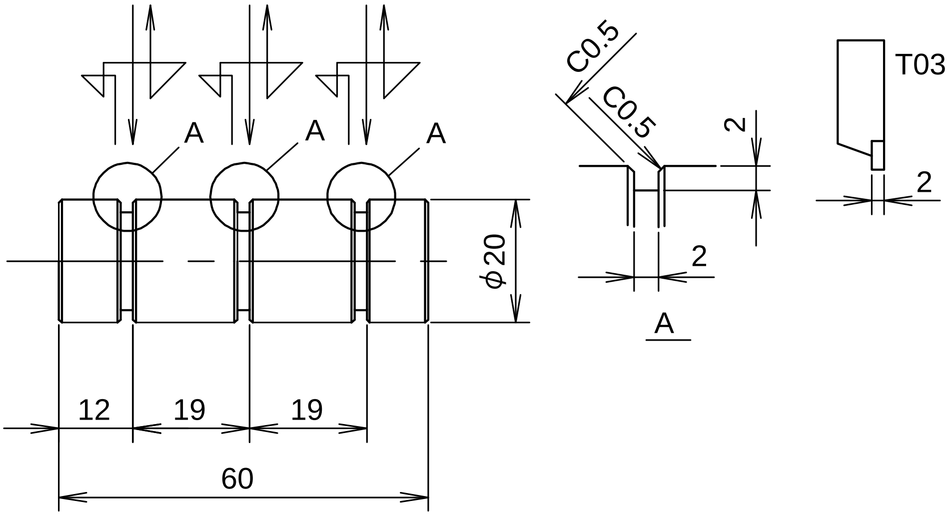
```
O0200

M99
%
```

Control returns to the main program.

Note 1: The R04 permits up to 9 repetitions for one call. Since the size of total program group to be executed is limited, the subprogram repetition count may be limited depending on the program length.

8.6.1 Example 1 - How a subprogram is used



Three grooves to be machined with T03 have the same shape meaning that the same program can be used for machining them. Programming can be simplified by registering the grooving program as a subprogram. In a subprogram, Z axis commands are expressed by W commands (relative commands) so that they can be used for grooving independent of the Z position of grooves.

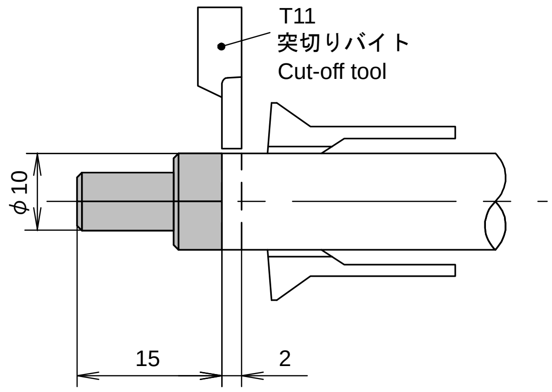
Program

```
Main program
.....
.....
M03 S1800(M03 S1=1800)
N0203 T0300
G00 X21.0 Z12.0 T03
M98 P0002
G00 X21.0 Z31.0
M98 P0002
G00 X21.0 Z50.0
M98 P0002
.....
.....
```

```
Subprogram
O0002
G00 X21.0
G01 X16.0 F0.01
X21.0 F0.2
W-1.0
X19.0 W1.0 F0.01
X21.0 F0.2
W1.0
X19.0 W-1.0 F0.01
X21.0 F0.2
M99
```

8.6.2 Example 2 - How a subprogram is used

Stepping machining is sometimes used in grooving and cut-off machining to remove chips. Programming of a stepping machining can be simplified by using a subprogram.



The sample program below makes a cut-off machining program using stepping machining which returns a tool 0.1 mm after cutting 1 mm in diameter. A subprogram is repeated 14 times since the tool cuts off the material starting from X11.1 up to X–3.0. With the NC unit of Mitsubishi Electric, a subprogram can be stated following the main program since an H argument can be used so that the program is handled as one program.

Program

NC unit produced by FANUC

```
Main program
.....
.....
N0611
M03 S3100
T1100
G00 X11.1 Z17.0 T11
M32
M98 P0003 L14
M33
M05
M07
G00 X-3.0 Z0 T00
M56
M02
M99
%
```

```
Subprogram
O0003
G01 U-1.1 F0.01
G01 U0.1 F0.2
M99
```

NC unit produced by MITSUBISHI Electric

```
Main program
.....
.....
N0611
M03 S3100
T1100
G00 X11.1 Z17.0 T11
M32
M98 H0003 L14
M33
M05
M07
G00 X-3.0 Z0 T00
M56
M02
M99

N0003
G01 U-1.1 F0.01
G01 U0.1 F0.2
M99
%
```

Subprogram

8.7 Bar Loader

8.7.1 Operating the bar loader (M54 and M55)

Use the M54 and M55 commands to operate the bar loader.

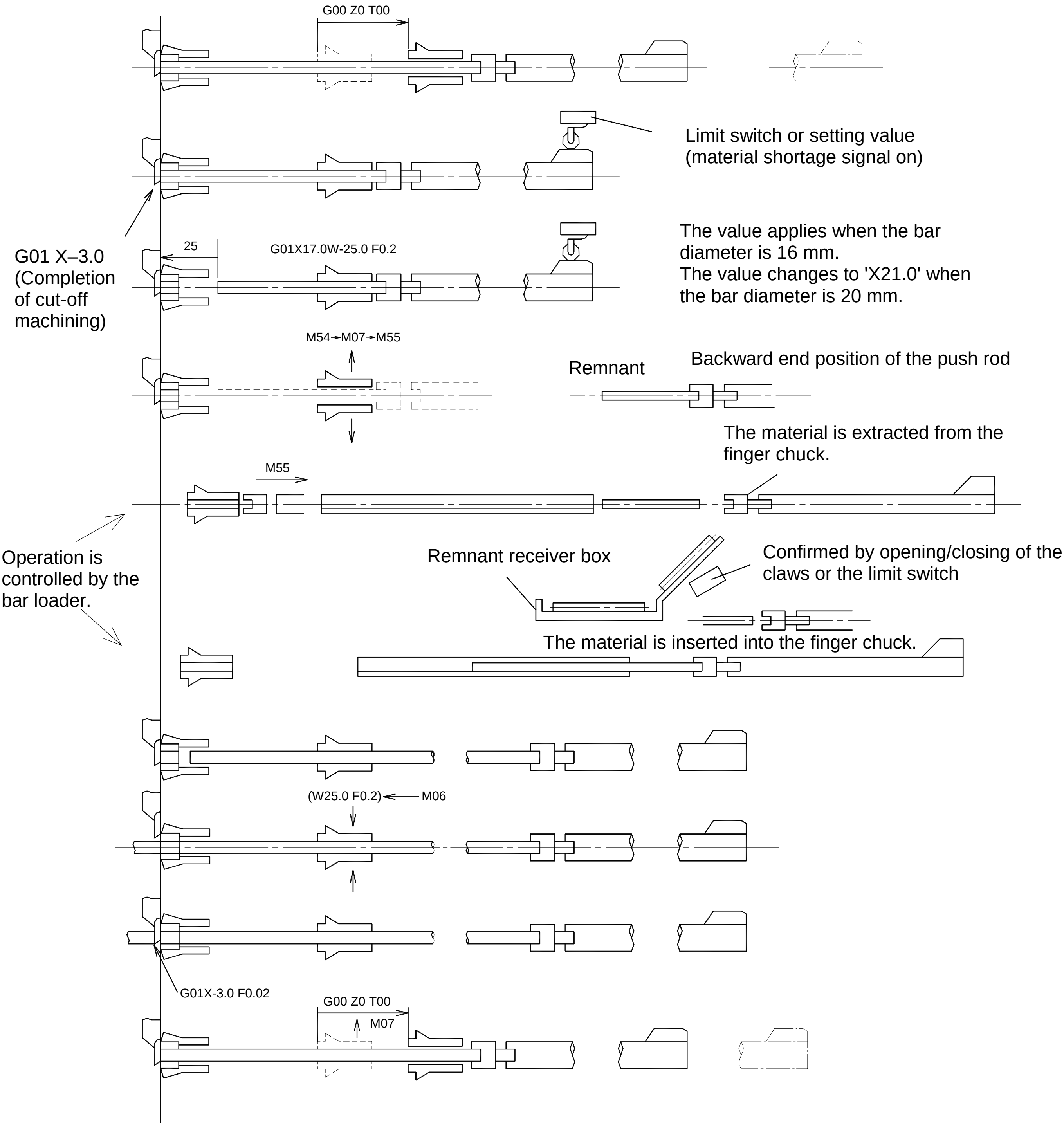
Command format

M54..Bar loader torque OFF

M55..Bar loader operation start

Generally, material replacement operation is executed in the same sequence for almost all machine models. However, programming differs according to the machine model.

Operating diagram



8.7.2 Material replacement program For M series, C series, A20 and A32

The commands enclosed by M08 and M09 are executed when the bar replacement signal is input from the bar loader. Specify a block skip code "/" at the head of the blocks between M08 and M09.

Command format

M08...Material replacement signal check

M09...End of the Material replacement program.

Program sample

Preparation process
M09...**End of the Material replacement program.**
↓
Machining process
↓
T1100...Selection of the cut-off tool T11.
G00 X17.0 Z22.0 T11..................Positioning to cut off the product in the correct length. Specify the tool position compensation command T11.
M32...The workpiece separator moves forward.
G01 X-3.0 F0.015 ....Cut-off machining is specified.
M33...The workpiece separator moves backward.
M08...**Material replacement signal check**
M08...**Material replacement signal check**
/ (   ) ....Program for deburring the outer diameter of materials
/ G01 X17.0 W-25.0 F0.2..........Retracting the cut-off tool to the material outer diameter position simultaneously with the material extraction from the guide bushing
/ M53...Coolant OFF
/ M05...Main spindle stop
/ M54...Bar loader machining torque OFF
/ M07...The spindle chuck opens.
/ M55...Bar loader start (material replacement)
/ M06...The spindle chuck closes.
/ G4 U2.0.....................................The delay of bar loader torque selection is prevented.
/ M52...Coolant ON
/ M03 S1=1000..............................Spindle start in the forward direction
/ G01 W25.0 F0.2........................The material is inserted into the guide bushing.
/ X-3.0 F0.02..............................Cut-off of the material front end
M09...**End of the Material replacement program.**
M05...The main spindle stops.
M07...The spindle chuck opens.
G00 X-3.0 Z0 T00 ....Returning the tool to the start position
M56...Count the number of products. The number of products is counted only during continuous operation.
M02...1 cycle stop
M99...Program return. Control returns to the beginning of the program.
%

Notes

- Create a program as a process for deburring the tip of remnant if necessary.
- The start and end points of material change program must be equal in the program. If the end point does not match the start point, an alarm may occur.
- All blocks preceded by "/" can be collectively handled as a subprogram by specifying /M98 P□□□□.

8.7.3 Material replacement program For B12, BL series and R04/07

A material replacement program created as a subprogram is executed in response to the execution of M109. M109 is executed when the material replacement command is input from the bar loader.

Command format

M09 End of the Material replacement program.

M109 Q□□□□

Q: Material change program number (numerics only)

Program sample

Preparation process

M09	Material replacement program	
	O7000	Material replacement program
Machining process	( )	Program for deburring the outer diameter of materials
	G01 X17.0 W-25.0 F0.2	Retracting the cut-off tool to the material outer diameter position simultaneously with the material extraction from the guide bushing
Cutting-off process	M53	Coolant OFF
M32	M05	Main spindle stop
G01X-3.0 F0.04	M54	Bar loader machining torque OFF
M33	M07	Open spindle chuck.
M109 Q7000	M55	Bar loader start (material replacement)
M05	M06	Close spindle chuck.
M07	G04 U2.0	Prevention of delay in bar loader torque selection
G00 X-3.0 Z0 T00	M52	Coolant ON
M56	M03 S1000	Spindle start in the forward direction
M02	G01 W25.0 F0.2	Inserting the material into the guide bushing
M99	G01 X-3.0 F0.02	Cutting off the material front end
%	M99	End of the material replacement program
	%	

Notes

- Create a program as a process for deburring the tip of remnant if necessary.
- The program number must be 4 digits or less. Numbers 9000s cannot be used.
- The start and end points of material change program must be equal in the program. If the end point does not match the start point, an alarm may occur.
- The material change program cannot call a subprogram.
- The contents of the material change program is not displayed on the screen during execution.
- The material replacement program cannot be executed in blocks.

8.7.4 Material replacement program
For A12 and A16

/M109 specified between M08 and M09 is executed when the material replacement signal is input from the bar loader. The material replacement program prepared as a subprogram is executed in response to the execution of M109.

Command format

M08 Material replacement signal check
M09 End of the Material replacement program.
/M109 Q□□□□
Q: Material change program number (numerics only)

Program sample

Preparation process

Main program	Material replacement program	
M09		
↓	O7000	Material replacement program
Machining process	( )	Program for deburring the outer diameter of materials
↓	G01 X17.0 W-25.0 F0.2	Retracting the cut-off tool to the material outer diameter position simultaneously with the material extraction from the guide bushing
Cutting-off process	M53	Coolant OFF
M32	M05	Main spindle stop
G01X-3.0 F0.04	M54	Bar loader machining torque OFF
M33	M07	Open spindle chuck.
M08	M55	Bar loader start (material replacement)
M08	M06	Close spindle chuck.
/M109 Q7000	G04 U2.0	Prevention of delay in bar loader torque selection
M09	M52	Coolant ON
M05	M03 S1000	Spindle start in the forward direction
M07	G01 W25.0 F0.2	Inserting the material into the guide bushing
G00 X-3.0 Z0 T00	G01 X-3.0 F0.02	Cutting off the material front end
M56	M99	End of the material replacement program
M02	%	
M99		
%		

Notes

- Create a program as a process for deburring the tip of remnant if necessary.
- The program number must be 4 digits or less. Numbers 9000s cannot be used.
- The start and end points of material change program must be equal in the program. If the end point does not match the start point, an alarm may occur.
- The material change program cannot call a subprogram.
- The contents of the material change program is not displayed on the screen during execution.
- The material replacement program cannot be executed in blocks.

8.8 Spindle Speed Change Detection Function

The spindle speed change detection function monitors the spindle speed. The function automatically stops the machine when it detects the speed fluctuation exceeding a specified limit. The function assists in preventing overload operation.

Command format

NC unit produced by FANUC

G25...Spindle speed change detection function OFF

G26...Spindle speed change detection function ON

NC unit produced by MITSUBISHI Electric

M97...Spindle speed change detection function OFF

M96...Spindle speed change detection function ON

Notes

- The spindle speed change detection function is ON when the power is turned on.
- For the program that uses M97 or G25, always specify M96 or G26 in the preparation process in the program.
- The command must be specified in a block independently. (Do not specify other commands in the same block.)

8.9 Constant Surface Speed Control Function

When a tool is fed into the workpiece along the X axis (radial direction) in such as cut-off process, the diameter being cut changes and, as the result, the surface speed (relative speed between the material and the tool) changes. When the surface speed control command is specified, the spindle speed is controlled as the tool position changes in the X axis direction to maintain the surface speed constant.

Command format

G50 S□□□□

In the constant surface speed control mode, spindle speed is controlled within the range of spindle speeds specified by Q and S.

- S: Maximum spindle speed (min^{-1}) for clamping

G96 S□□□□: Start of constant surface speed control

- S: Surface speed (m/min)

(On A20, G96 S1=□□□□ is specified.)

G97: End of constant surface speed control

Notes

- To use the constant surface speed control function, turn off the spindle speed change detection function by specifying M97 or G25.
- If an S value is not specified when G96 is selected, the S value of the previously specified G96 is effective.
- The maximum spindle speed specified by G50 S□□□□ is effective only while G96 (constant surface speed control) is enabled.
- Specify Q if the spindle speed is likely to become too low.
- Do not execute the tool change command (T) while constant surface speed control is enabled. When a tool change is needed, execute the tool change command after the cancel command (G97).
- For a program that uses the constant surface speed control, be sure to specify G97 in the program of the preparation process.

Program

Preparation process

G50 Z0..Set the coordinate system taking the current position (start position) as Z0.
M06...Close the front spindle chuck.
M09...**End of the Material replacement program.**
G97...**Constant surface speed control OFF**
M96 (G26)......................................**Spindle speed change detection function ON**
G99...The feed per rotation is specified.
G00 X13.0 Z-0.5 ....Move the cut-off tool away from the material to move it to the next process.
M03 S2300 (M03 S1=2300)........Start the front spindle in the forward direction.

↓

Machining process

↓

Cut-off end process
N0611
M03 S2600 (M03 S1=2600)..........Start the front spindle in the forward direction.
T1100...Selection of the cut-off tool T11.
G00 X13.0 Z22.0 T11..................Execute positioning of the tool to cut the product to the correct length.
M97 (G25)......................................**Spindle speed change detection function OFF**
G50 S5000......................................**Maximum spindle speed for clamping (5000 min^{-1})**
G96 S100 (G96 S1=100)..............**Constant surface speed control ON (surface speed: 100 m/min)**
M32...The workpiece separator moves forward.
G01 X-3.0 F0.015 ....The commands specify cut-off operation. Since the constant surface speed control mode is selected, the spindle speed is controlled meeting the X axis position so that the surface speed is maintained at 100 m/min. If the spindle speed reaches 5000 min^{-1}, the spindle speed does not increase exceeding this clamp speed.
M33...The workpiece separator moves backward.

↓

Material replacement program (program differs according to the machine model.)

↓

M05...Stop the front spindle.
M07...Open the front spindle chuck.
G00 X-3.0 Z0 T00 ....Return the tool to the start position and cancel the tool position compensation function.
G97...**End of constant surface speed control OFF**
M96 (G26)......................................**Spindle speed change detection function ON**
M56...Count the number of products. The number of products is counted only during continuous operation.
M02...One-cycle stop is specified. Control returns to the beginning of the program.
M99...Program return. Control returns to the beginning of the program.
%...Stop code. The stop code is automatically entered.

8.10 Front Spindle Indexing Function (M28, M20)

The front spindle can be indexed at 1° intervals. For some machine models, indexing intervals will be 15°.

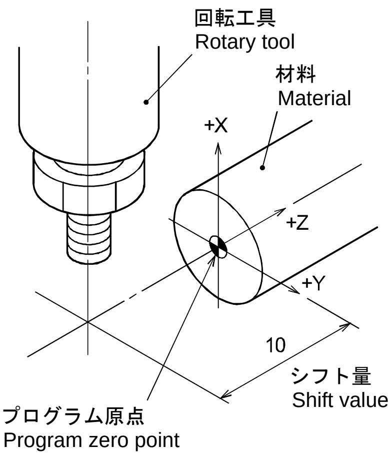
Command format

M28 S□□□..................................Front spindle indexing command

M20...Front spindle indexing cancel command

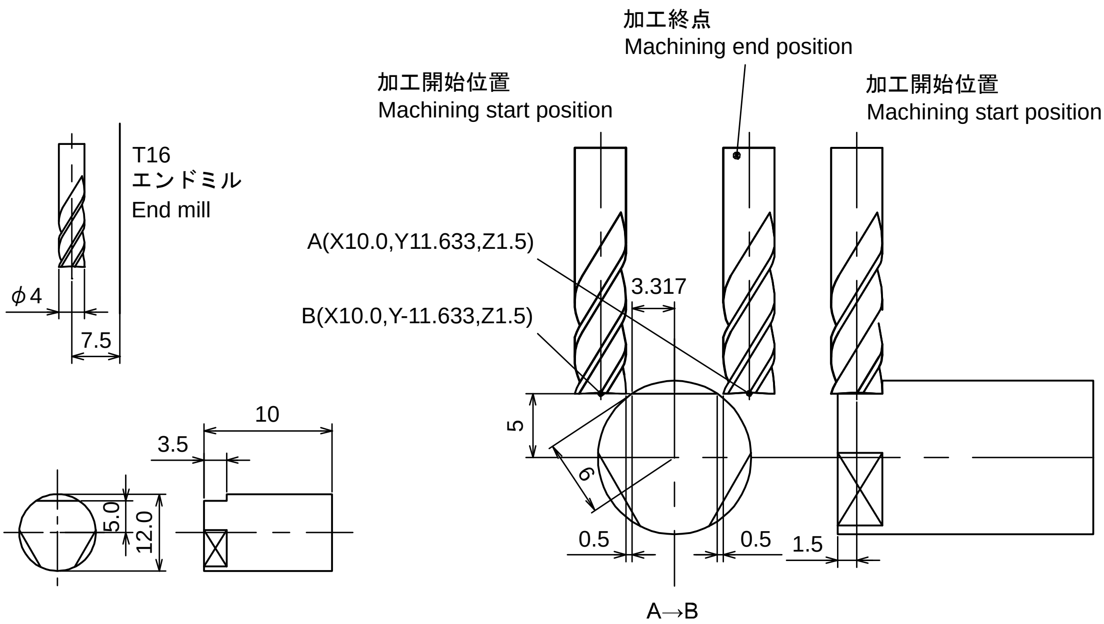
- Specify the indexing angle by S□□□ with an integer in the range of 0 to 3591. Note that a minus value cannot be specified.
- The front spindle is indexed in the forward rotation direction.
- Before indexing the front spindle, stop it once by specifying M5.
- To return to the turning process, cancel the spindle indexing function by specifying the M20 command, and then rotate the front spindle by executing M03 S□□□□ (M03 S1 =□□□□). Some machine can rotate the front spindle without executing the M20 command. For details, refer to the Instruction Manual of the machine.

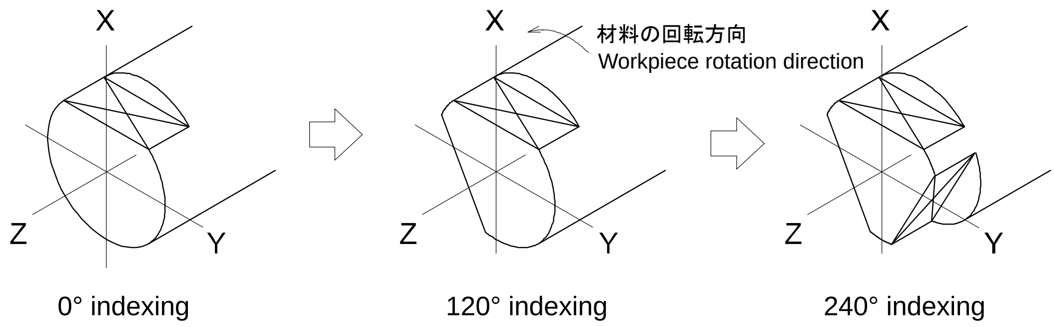
8.11 Secondary Machining (Cross Machining)



- In the secondary machining, three axes (X, Y and Z) are controlled. The program zero point is set at the same program zero point in turning operation, that is, it is set at the center of rotation on the end face of the material (X0, Y0, Z0).
- The plus sign indicates the upward movement of the X axis, leftward movement of the Z axis and the front side movement of the Y axis.
- A Y coordinate is usually specified in diameter. However, some models use a radial value. For details, refer to the <Instruction Manual of the Corresponding Machine Model>.
- Since a rotary tool is shifted in the Z axis direction, shift the coordinate system by specifying G50 W□□ when creating a program. If the shift amount is 10, as shown in the figure above, specify G50 –10.0 to shift the coordinate system for programming.

Sample





Program

```
N1608
M05..................................Front spindle stop
04 U0.5..............................Dwell time
M80 S1100 (M58 S3=1100)..............Rotary tool start in the forward direction
G98..................................Feed per minute mode
T1600................................Selecting an end mill T1600
G50 W-7.5............................Coordinate system shift in the Z axis direction
M28 S0...............................Spindle indexing command
G00 X13.0 Y-11.633 Z1.5 T16..........Positioning of the Y and Z axes at a rapid feed rate
G00 X10.0............................Positioning of the X axis to Point A at a rapid feed rate (to avoid
                                     interference between the end mill and the material)
G01 Y11.633 F110.....................First machining at point B
M28 S120.............................Spindle indexing command
G00 X13.0............................Positioning of the X axis at a rapid feed rate
G00 Y-11.633.........................Positioning of the Y axis at a rapid feed rate
G00 X10.0............................Positioning of the X axis to Point A at a rapid feed rate
G01 Y11.633..........................Second machining at Point B
M28 S240.............................Spindle indexing command
G00 X13.0............................Positioning of the X axis at a rapid feed rate
G00 Y-11.633.........................Positioning of the Y axis at a rapid feed rate
G00 X10.0............................Positioning of the X axis to Point A at a rapid feed rate
G01 Y11.633..........................Third machining at Point B
G00 X13.0............................Positioning of the X axis at a rapid feed rate
M82 (M60)............................Rotary tool stop
G4 U0.5..............................Dwell time
M20..................................Spindle indexing cancel command
G50 W7.5.............................Canceling the coordinate system shift in the Z axis direction
G99..................................Feed per rotation mode
M03 S3000 (M03 S1=3000)..............Start the main spindle in the forward direction.
```

8.12 Machining Data

- Bar Stock O.D.

  Enter the outer diameter of the material to be machined.

  The outer diameter data is used in automatic movement in tool setting operation such as [DIA], [CORE] and [Positioning Pont].

- Tool Positioning Point (DIA)   Outside Diameter of Material +

  Enter the value two times the clearance between the material outer diameter and the position where the nose of the selected tool is positioned.

  The positioning point data is used in positioning operation when a tool is selected in a program together with the material outer diameter data. It is also used when executing the positioning point function on the Preparation screen.

- Cut-Off   Tool

  This is the tool number where a cut-off tool is mounted.

  The cut-off tool number is used in automatic movement of the start position function.

- Cut-Off   Speed (mm^{-1})

  This is the spindle speed used for cutting the material end face in such as cut-off operation called on the Preparation screen.

- Cut-Off   Feed (mm/rev)

  This is the feed rate used for cutting the material end face in such as cut-off operation called on the Preparation screen.

- Cut-Off   End (DIA mm)

  This is the X coordinate of the material end face cutting in such as cut-off operation called on the Preparation screen.

  It is also used as the start position of the X axis for starting a program.

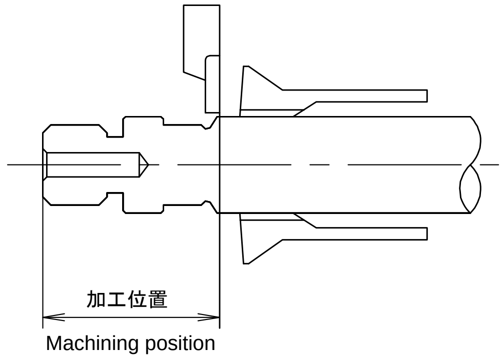
- Machining Length

The machining length data is used for determining spindle position (Z1) of the start position.

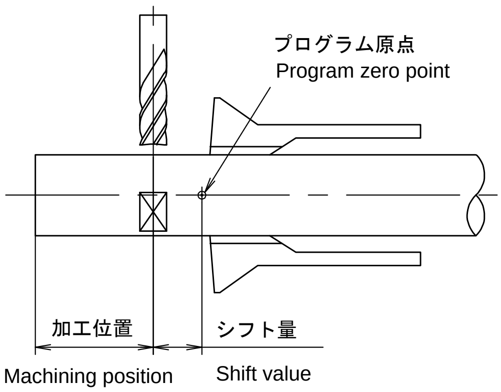
Enter the maximum spindle move distance necessary for machining a workpiece.

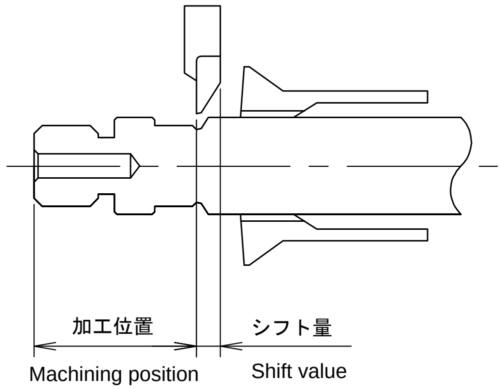
Usually, a value 1 mm added to value (1) or value (2), whichever larger, is entered.

(1) Machining position of cut-off operation



(2) Machining position of such as secondary machining, back turning and thread cutting to be carried out by a shifted tool + Shift amount





- Pieces/1Chuck

Enter the number of products to be collected after the execution of a program.

Use this data when multiple products are machined in one chucking. Usually, “1” is entered.

Machining Length × (Pieces/1Chuck) = Spindle position (Z1) of the start position

The data to be entered may slightly differ according the machine model.

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	8	0	0						

9. Cutting Conditions

Code No.	C-CINCOM_IGB 4E1-0900	MFG No.	CINCOM	Issue Date	2012.6

(Blank page)

9.1 Cutting Conditions

This section shows standard cutting conditions. In actual operation, specify cutting conditions in consideration of the material of workpieces to be machined, the shapes of tools to be used, and the materials of the tools.

The cutting speeds shown in the tables below are determined on the assumption that a rotary guide bushing is used. If a fixed guide bushing is used, set 50 to 70 % of the cutting speeds in the tables.

9.2 Cutting Speed and Feed Rate

As a rule, a cutting speed must be obtained in accordance with the outer diameter of the material. Exceptionally set a cutting speed higher than the standard for the outer diameter of the material in the following case: the infeed amount is large, and the difference between the finishing diameter and the material diameter is great.

9.2.1 Cutting speed for outer diameter turning (front turning, back turning, etc.) (carbide tools)

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 (Equivalent to or lower than SUS303F)	100 to 180
Aluminum	200 to 400		
Free cutting steel	150 to 250	Stainless steel (Equivalent to or lower than SUS304F)	60 to 150
Carbon tool steel	120 to 200		
Structural carbon steel	80 to 180	Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	30 to 80

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

V: Cutting speed (m/min)
D: Material outer diameter (mm)
N: Spindle speed (min^{-1})

9.2.2 Cutting feed rate for outer diameter turning (front turning, back turning, etc.) (carbide tool)

A feed rate varies depending on the infeed amount, the required surface roughness, the precision, and the expected tool life as well as the material to be machined.

A: Dimensional tolerance: Within ±5 μm, Surface roughness: Within 3S, Infeed amount: 4 mm or more

B: Dimensional tolerance: Within ±10 μm, Surface roughness: Within 6S, Infeed amount: 2 to 4 mm

C Dimensional tolerance: Within ±10 μm, Surface roughness: Without 6S, Infeed amount: 2 mm or less

mm/rev

Material	Feed rate in diametric direction feed (X)			Feed rate in 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
Structural carbon steel	0.015	0.025	0.05	0.03	0.05	0.1
Alloy tool steel	0.01	0.02	0.03	0.03	0.05	0.1
Free cutting stainless steel (Equivalent to or lower than SUS303F)	0.015	0.025	0.05	0.03	0.05	0.1
Stainless steel (Equivalent to or lower than SUS304F)	0.01	0.02	0.03	0.03	0.05	0.1
Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	0.008	0.015	0.02	0.02	0.04	0.08

9.2.3 Cutting speed for inner diameter turning (boring) (carbide tools)

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	50 to 90
Aluminum	100 to 200	(Equivalent to or lower than SUS303F)	
Free cutting steel	80 to 120	Stainless steel	30 to 80
Carbon tool steel	60 to 100	(Equivalent to or lower than SUS304F)	
Structural carbon steel	40 to 90	Heavy cutting stainless steel	15 to 40
		(Equivalent to or higher than SUS304F)	

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

V: Cutting speed (m/min)
D: Inner machining diameter (mm)
N: Spindle speed (min^{-1})

9.2.4 Cutting feed rate for inner diameter turning (boring) (carbide tools)

The cutting conditions for boring vary largely depending on the rigidity of the tool and the tool holder.

The appropriate diametrical feed rate is about the half of the longitudinal feed rate.

A: Dimensional tolerance: Within ±5 μm, Surface roughness: Within 3S, Infeed amount: 4 mm or more

B: Dimensional tolerance: Within ±10 μm, Surface roughness: Within 6S, Infeed amount: 2 to 4 mm

C Dimensional tolerance: Within ±10 μm, Surface roughness: Without 6S, Infeed amount: 2 mm or less

mm/rev

Material	Feed rate in 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
Structural carbon steel	0.015	0.03	0.05
Alloy tool steel	0.015	0.03	0.05
Free cutting stainless steel (Equivalent to or lower than SUS303F)	0.015	0.03	0.05
Stainless steel (Equivalent to or lower than SUS304F)	0.015	0.025	0.04
Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	0.01	0.02	0.03

9.2.5 Reaming

The cutting conditions for reaming vary largely depending on the reamer.

Material	Cutting speed (m/min)	Feed rate (mm/rev)		
		ø1 to ø2	ø2 to ø5	ø5 to
Brass	10 to 20	0.07 to 0.25	0.15 to 0.5	0.25 to 1.0
Phosphor bronze	10 to 20	0.03 to 0.15	0.1 to 0.3	0.3 to 0.5
Aluminum	20 to 30	0.08 to 0.25	0.15 to 0.5	0.25 to 1.0
Free cutting steel	10 to 20	0.06 to 0.25	0.15 to 0.5	0.25 to 1.0
Carbon tool steel	10 to 20	0.05 to 0.15	0.1 to 0.3	0.1 to 0.5
Structural carbon steel	10 to 20	0.03 to 0.15	0.05 to 0.3	0.1 to 0.5
Alloy tool steel	6 to 12	0.03 to 0.1	0.05 to 0.2	0.1 to 0.3
Free cutting stainless steel (Equivalent to or lower than SUS303F)	10 to 20	0.03 to 0.12	0.05 to 0.2	0.1 to 0.4
Stainless steel (Equivalent to or lower than SUS304F)	10 to 20	0.02 to 0.1	0.05 to 0.2	0.1 to 0.3
Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	6 to 12	0.02 to 0.08	0.04 to 0.15	0.08 to 0.25

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

V: Cutting speed (m/min)
D: Outer diameter of reamer (mm)
N: Spindle speed (min^{-1})

9.2.6 Thread cutting speed with a tap or die

	Non-ferrous material	Ferrous material
M1 to M3	600 to 1000 min^{-1}	500 to 800 min^{-1}
M4 to M6	400 to 800 min^{-1}	300 to 600 min^{-1}

9.2.7 Cutting speed for a hole machining tool (centering drill, drill) (high-speed tool steel)

m/min

Material	ø1 to ø2	ø2 to ø5	ø5 to ø12
Brass	40 to 60	60 to 80	80 to 100
Aluminum	40 to 60	60 to 80	80 to 100
Free cutting steel	30 to 40	40 to 60	60 to 80
Carbon tool steel	20 to 30	30 to 50	50 to 60
Structural carbon steel	15 to 20	20 to 30	30 to 35
Alloy tool steel	8 to 12	12 to 15	15 to 20
Free cutting stainless steel (Equivalent to or lower than SUS303F)	20 to 30	30 to 40	40 to 50
Stainless steel (Equivalent to or lower than SUS304F)	To 10	10 to 15	15 to 20
Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	To 8	8 to 10	10 to 15

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

V: Cutting speed (m/min)
D: Inner diameter of a hole (mm)
N: Spindle speed (min^{-1})

9.2.8 Cutting feed rate with a hole machining tool (centering drill, drill) (high-speed tool steel)

mm/rev

Material	ø1 to ø2	ø2 to ø5	ø5 to ø12
Brass	0.03 to 0.06	0.06 to 0.12	0.12 to 0.25
Aluminum	0.03 to 0.06	0.06 to 0.12	0.12 to 0.25
Free cutting steel	0.025 to 0.05	0.05 to 0.10	0.1 to 0.2
Carbon tool steel	0.02 to 0.04	0.04 to 0.08	0.08 to 0.16
Structural carbon steel	0.04 to 0.05	0.04 to 0.08	0.08 to 0.16
Alloy tool steel	0.015 to 0.03	0.03 to 0.06	0.06 to 0.12
Free cutting stainless steel (Equivalent to or lower than SUS303F)	0.02 to 0.04	0.04 to 0.08	0.08 to 0.16
Stainless steel (Equivalent to or lower than SUS304F)	0.015 to 0.03	0.03 to 0.06	0.06 to 0.12
Heavy cutting stainless steel (Equivalent to or higher than SUS304F)	0.01 to 0.02	0.02 to 0.04	0.04 to 0.08

Use the following formula when converting a cutting feed rate from “mm/rev” value to “mm/min” value.

$$Fm = Fr \times N$$

Fm: Feed rate (mm/min)
Fr: Feed rate (mm/rev)
N: Spindle speed (min^{-1})

9.3 Thread Cutting Count

The required infeed depth (radius value) in each infeed step can be found in the column of the thread pitch value of the thread to be cut.

9.3.1 Non-ferrous material

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
Thread cutting count	Infeed 1	0.22	0.22	0.25	0.25	0.32	0.35	0.35	0.35	0.40	0.40
	Infeed 2	0.07	0.18	0.20	0.22	0.22	0.25	0.23	0.27	0.35	0.36
	Infeed 3	0.03	0.05	0.07	0.13	0.14	0.16	0.20	0.22	0.25	0.30
	Infeed 4				0.05	0.08	0.10	0.15	0.15	0.20	0.25
	Infeed 5					0.05	0.06	0.09	0.12	0.15	0.20
	Infeed 6						0.05	0.06	0.08	0.12	0.15
	Infeed 7							0.05	0.06	0.07	0.12
	Infeed 8								0.05	0.05	0.07
	Infeed 9									0.03	0.05
	Infeed 10										0.03
	Infeed 11										0.02

9.3.2 Ferrous material

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
Thread cutting count	Infeed 1	0.18	0.22	0.22	0.22	0.22	0.25	0.28	0.30	0.32	0.35
	Infeed 2	0.08	0.13	0.14	0.15	0.16	0.18	0.18	0.21	0.22	0.23
	Infeed 3	0.04	0.07	0.10	0.11	0.12	0.14	0.12	0.16	0.18	0.19
	Infeed 4	0.02	0.03	0.04	0.09	0.08	0.09	0.09	0.11	0.13	0.16
	Infeed 5			0.02	0.05	0.08	0.08	0.08	0.09	0.12	0.15
	Infeed 6				0.03	0.07	0.07	0.07	0.08	0.11	0.13
	Infeed 7					0.05	0.06	0.07	0.08	0.10	0.11
	Infeed 8					0.03	0.05	0.06	0.07	0.08	0.11
	Infeed 9						0.03	0.06	0.05	0.08	0.09
	Infeed 10						0.02	0.05	0.05	0.07	0.08
	Infeed 11							0.04	0.05	0.06	0.07
	Infeed 12							0.03	0.03	0.06	0.06
	Infeed 13								0.02	0.04	0.06
	Infeed 14									0.03	0.05
	Infeed 15									0.02	0.05
	Infeed 16										0.04
	Infeed 17										0.02

9.4 Cutting Condition Tables for the Secondary Machining Process

9.4.1 End milling (carbide tools)

The cutting conditions in the table below assume the depth of cut is equivalent to the end mill diameter. If the depth of cut is larger than the end mill diameter, cutting conditions must be adjusted.

Material	Diameter mm	Cutting speed m/min	Infeed amount mm	Feed rate mm/cutting edge
Non-ferrous material	ø2	30 to 100	2.0	0.02 to 0.03
Brass	ø3	30 to 100	3.0	0.03 to 0.04
Aluminum	ø4	30 to 100	4.0	0.04 to 0.05
etc.	ø5, ø6	30 to 100	5.0 to 6.0	0.05 to 0.08
Ferrous free-cutting material	ø2	30 to 80	2.0	0.015
(e.g., free cutting steel)	ø3	30 to 80	3.0	0.02
	ø4	30 to 80	4.0	0.03
	ø5, ø6	30 to 80	5.0 to 6.0	0.03 to 0.06
Ferrous material	ø2	20 to 60	2.0	0.015 to 0.02
Carbon tool steel	ø3	20 to 60	3.0	0.015 to 0.02
Structural carbon steel	ø4	20 to 60	4.0	0.025 to 0.03
etc.	ø5, ø6	20 to 60	5.0 to 6.0	0.04 to 0.06
Alloy tool steel	ø2	15 to 50	2.0	0.008
Stainless steel	ø3	15 to 50	3.0	0.015
etc.	ø4	15 to 50	4.0	0.025
	ø5, ø6	15 to 50	5.0 to 6.0	0.03 to 0.04

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

V: Cutting speed (m/min)
D: End mill diameter (mm)
N: Spindle speed (min^{-1})

Calculate the feed rate (mm/min) from the feed rate value in mm/cutting edge unit.

$$Fm = (\text{Feed rate mm/cutting edge}) \times \text{Number of cutting edges} \times N$$

Fm: Feed rate (mm/min)
N: Spindle speed (min^{-1})

9.4.2 Slitting cutter (high-speed tool steel)

The conditions for slitting vary largely depending on the drive motor power and the deceleration rate.

Material	Width mm	Cutting speed m/min	Infeed amount mm	Feed rate mm/cutting edge
Non-ferrous material	0.5	40 to 100	To 6.0	0.008
Brass	1	40 to 100	To 6.0	0.006
Aluminum	2	40 to 100	To 6.0	0.005
etc.	3	40 to 100	To 6.0	0.004
Ferrous free-cutting material	0.5	40 to 80	To 4.0	0.0035
(e.g., free cutting steel)	1	40 to 80	To 4.0	0.003
	2	40 to 80	To 4.0	0.003
	3	40 to 80	To 4.0	0.002
Ferrous material	0.5	30 to 70	To 3.0	0.0025
Carbon tool steel	1	30 to 70	To 3.0	0.002
Structural carbon steel	2	30 to 70	To 3.0	0.002
etc.	3	30 to 70	To 3.0	0.0015
Alloy tool steel	0.5	20 to 50	To 2.5	0.002
Stainless steel	1	20 to 50	To 2.5	0.002
etc.	2	20 to 50	To 2.5	0.001
	3	20 to 50	To 2.5	0.0005

Calculate the spindle speed from the cutting speed.

$$N = \frac{V \times 1000}{3.14 \times D}$$

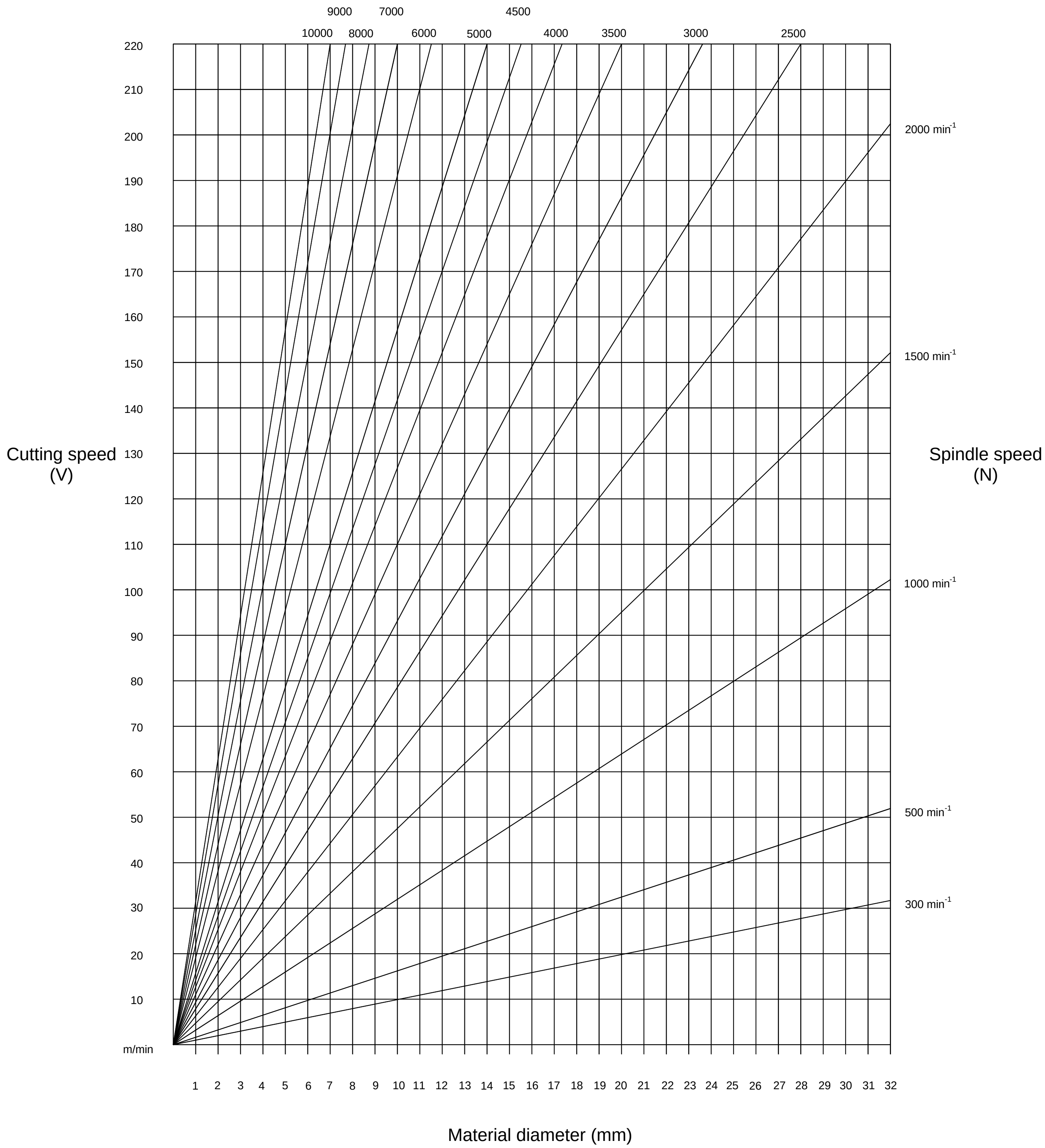
V: Cutting speed (m/min)
D: Slitting cutter diameter (mm)
N: Spindle speed (min^{-1})

Calculate the feed rate (mm/min) from the feed rate value in mm/cutting edge unit.

$$Fm = (\text{Feed rate mm/cutting edge}) \times \text{Number of cutting edges} \times N$$

Fm: Feed rate (mm/min)
N: Spindle speed (min^{-1})

9.5 Quick Reference Table for Cutting Speed



Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	0	9	0	0						

10. Terminology

(Blank page)

Term	Description
One cycle stop	A program is automatically executed for one cycle, and then it stops.
3-spindle	A tool post for mounting drilling tools, or tools. The origin of this term is that three tools are generally mounted.
C axis	A control axis for rotating a workpiece. C axis facilitates the secondary machining and special machining.
GB	Guide Bushing
LCD	Liquid Crystal Display
MDI	Manual Data Input. This method enters words directly to the machine instead of entering them through a program.
MDI operation	Automatic operation of a program entered by the MDI method
GBL	This stands for Guide-Bushing Less.
NMT	Network Machine Tool
NOP	Network Option
OS	Operating System
OT	Overtravel. Status in which the machine has exceeded the operation range.
PCMCIA card	Personal Computer Memory Card International Association. A PCMCIA card is generally called a PC card which is used as a card-type extension device (e.g., storage or interface) with a portable personal computer.
PLC constant	A constant used by PLC (ladder) software.
Safe mode	The safe mode activates the system (Windows) with the minimum system driver. If the previous activation has failed or a registry has been damaged, the system is automatically activated in the safe mode. This mode is used mainly for system repair.
TI axis	A turret indexing axis for selecting a turret tool.
Windows	One of operating systems (basic software) of personal computers which provides environment for multitask operation and multi-window operation.
Contact	A degree of contact between two faces. Good contact indicates a high degree of contact.
Absolute command	An absolute value command which enables the move to a determined position regardless of the current position.
Alkart (Cincom) net	Alkart net permits connection to the computer installed at Citizen Machinery Miyano Co., Ltd. through the Internet and provides various types of services.

Term	Description
Profile material machining	Machining of a square or hexagonal material other than a round material
Phase shift	A shift in the direction in which various workpieces under the secondary machining are rotating or in which workpieces under back machining or front machining are rotating.
Incremental command	An incremental value command which enables the move by a determined distance from the current position.
Inching	Operation for replacing the material on the bar loader when a profile material is used. The bar loader carries the material to insert it into the spindle chuck of the machine. When the tip of the material arrives at the end of the chuck, the bar loader pushes the material several times. Inching is this repetitive pushing.
Indexing	Indexing of the turret tool post
Inverted ring	An inverted ring is used for securing a bearing and so on in the axial direction by making use of a groove machined on a shaft or in a hole.
Extension chucking device	A chucking device having a longer overhang than the standard chucking device.
Apron	A common name of a part of the machine cover which surrounds the machine. The purpose of the apron is to collect oil.
Circular interpolation	Moving to the command-specified position through circular machining while keeping the feed rate constant.
End of block	A semicolon (;) entered as a line feed character at the end of a program line. Line feed is permitted on the screen.
End mill	An end mill is used for milling the outer circumference or end face of a cylinder with a rotary cutting tool having a cutter.
Override	A percentage of the feed rate. An override value is generally set with a dial. When 100% is set, the machine operates at the feed rate specified by the command. When 0% is set, the machine does not operate.
Push rod	A part of the bar loader which pushes a material.
Optional stop	When a program is automatically operated, this function stops the program with the block specified with M01 in the program. Generally, this function is enabled with the button on the operation panel.
Offset	An amount of compensation for tool wear. Generally, an amount of tool mounting compensation can also be interpreted as offset in a broad sense.
Operating system	The most fundamental program for operating a computer. This software controls and manages computer resources (e.g., memory, disks, and peripheral equipment) to provide useful environment to users.
Orientation	Indexing of the spindle.
Cursor	A mark indicating the input position on the screen. When a character is entered from the keyboard, it is input to the position where the cursor is sitting.
Free cutting steel	A steel material including 0.10 to 0.25% sulfur to make it easy to cut.
Start position	A position where machining starts. Generally, the spindle moves back to the start position when cut-off machining is completed. The start position is identical to the end position in the program.
Start position operation	Operation for moving the tool post and headstock to the start position. This operation is executed with the preparation function.

Term	Description
Regenerative brake resistor	When a brake is applied to a motor, energy is generated. A regenerative brake resistor is used for converting the energy into heat and radiating the heat.
Guide sleeve	A guide sleeve is used to support a pipe or knock-out rod of a long workpiece device and knock-out device.
Guide pipe	Generally, a guide pipe is used to secure and support a workpiece stock pipe of a long workpiece device.
Guide bushing device	A supporting device for preventing material deflection near the cutting point whenever a long size workpiece having a small diameter is machined. An automatic unit equipped with a guide bushing is called an automatic Swiss-type unit.
Burr	Rough and irregular protrusion at the edges of a machined workpiece.
Machining reference point	A reference point for programming. Generally, the center of the end face of a workpiece is defined as a reference point.
Machining data	A collection of data for machining. Machining data is stored and managed in the same manner as for the machining program.
Machining layout	Generally, machining layout is expressed with a diagram explaining the processes for machining a workpiece and the order in which the processes are performed.
Galling	Status of parts which rub against each other and whose surfaces are torn or fused.
Custom program	A program arranged by a user. Under NC, a custom program is used for creating a screen specific to a machine manufacturer.
Custom variable	A variable used in users' programs. Custom variables are used with the machine manufacturer's macros.
Virtual tool nose	A tool nose position on the supposition that a tool nose is not round when it is actually round.
Y-wrench	A Y-shaped wrench
Flammable material	A material which is likely to burn. It burns with sparks generated while it is being cut. A flammable material consists of magnesium, titanium, zirconium, or sodium alloy.
Interference	Status in which a mobile section of the machine, workpiece, and tool run into each other.
Keyboard	A device for entering characters.
Machine coordinate	A coordinate as reference to the machine. The machine coordinate is invariable.
Cap connector	A wiring part for securing cables while closing them tightly. Product name.
Coolant	Liquid coolant.
Gang tool	A tool post for mounting tools or a tool. The origin of this term is that tools are placed like the teeth of a comb.
Clamp	To secure the machine or a tool so that a command value (e.g., speed) does not exceed a specified value.
(Turret) clamp	To secure a turret after it is turned in the indexing direction. Generally, the turret is finally positioned with the coupling.

Term	Description
Clamp block	Generally, a part of a workpiece chucking device which chucks a workpiece is called a clamp block.
Cross machining	Machining of a workpiece in the direction at the right angle (cross) to the longitudinal (Z) direction of the workpiece. Cross machining can also refer to machining with an axis cross.
Axis control group	Grouping of multiple tool posts which operate independently of each other. The tool posts are grouped with Z axis as reference.
Zero point return	Action for return to the reference point of the machine operation system.
Built-up edge	A material which is fused onto a tool nose by heat or vibration generated by machining workpieces.
Return position	A position where a tool post and the headstock separate from each other. Generally, the tool post moves into the zero point return direction.
Stationary guide bushing device	A guide bushing device which does not have a rotary mechanism. A stationary guide bushing is used with a material having a relatively small diameter.
Canned cycles for drilling	A function which executes a machining program for positioning, drilling, boring, or tapping with the 1-block command.
Collet chuck	A collet chuck is used to mainly chuck a round bar material. It has several cut lines in the axial direction and chucks a material while pushing the tapered part on the outer circumference into the axial direction.
Circuit protector	A device which shuts down the power when overcurrent is detected.
Servo amplifier parameter	A parameter for a servo motor. It is registered in the amplifier.
Servo parameter	A parameter for a servo motor. It is registered in the NC unit.
Servo motor	A motor for controlling a position.
Thermal protector	A device which detects an overload caused by overcurrent and shuts down the power.
Differential rotary tool function	A function which controls operation by superimposing the rotary tool speed on the spindle speed. This function is used for tapping the center of a workpiece with the rotary tool while the workpiece chucked on the spindle is being rotated.
Saddle	A part which moves the table, tool post, or feed change mechanism along the guide face. The saddle of a lathe is an H-shaped part where the carriage is mounted on the bed.
Coordinate system shift	To shift a work coordinate by a specified amount in a machining program. This function is used when a tool is shifted from the standard position.
Sequence	Sequential procedure
Axis cross	A function which exchanges axes, belonging to axis control groups, between the groups. Generally, this function is specified in the macro.
Axis parameter	A parameter for axes (spindle excluded) controlled by the machine.
Microindicator	A type of a dial gauge which is graduated in divisions of 1 µm or less.
Automatic operation	Continuous program execution in a cycle or in a block.
Shank size	A dimension of a square of the tool part to be clamped.
Shank diameter	A diameter of the cylindrical-tool part to be gripped by the chuck.

Term	Description
Junction head	A lubrication piping part which branches an oil flow.
Weight	Weight provided to push a material into a single-bar feeder.
Constant surface speed control	A function which keeps the surface speed (the relative speed of a workpiece and tool) constant. This function changes the spindle speed by reading the tool position.
Slide surface	Generic name of a surface in contact with a slipping slide.
Spindle	An axis (spindle) for rotating the material to be machined. Generally, a spindle indicates the main spindle.
Spindle speed change detection	A function which monitors the spindle speed and issues an alarm when the spindle speed is out of a specified range for a period of time.
Spindle C axis function	A C axis function for the spindle. A program can be specified with the C address.
Headstock	A table on which the spindle is mounted. When the spindle is of mobile type, the headstock has a moving axis.
Spindle indexing function	A function which indexes the spindle at specified intervals.
Lubrication oil level detection	A function which monitors the amount of remaining lubrication oil and issues an alarm when the lubrication oil level becomes equal to or lower than a specified level.
Momentary power failure	Status in which the power momentarily lowers or shuts down.
Core height	A shift from the center point of workpiece rotation to the cutting point.
Super-option	A function which gives a point to each function so that software/option functions can be used in combination for each program.
Scan disk	A system tool (software) which detects and corrects hard disk errors.
Rake face	A front face of a tool in the cutting direction. Chips flows onto the rake face.
Screen saver	Software for protecting the computer screen from burn when the keyboard and mouse are not operated for a period of time.
Station	A location for mounting tooling. For a turret, each face is called a station.
Stepping (in drilling)	Drilling conditions becomes worse as a drill goes more deeply into the hole during deep hole machining. Stepping is a machining method to perform infeed operation several times for a hole.
Splash guard	A cover for preventing coolant and chips from scattering.
Space key	A key for entering a space.
Thrust	A longitudinal direction of an axis. A diametrical direction is called a radial direction.
Sleeve holder	A tooling which secures sleeves for drilling, boring, and tapping.
Slitting cutter	A disk cutting tool used to machine a relatively narrow and deep groove in the outer circumference or end face of a material.

Term	Description
Slitting spindle	A rotary tool (tooling) for mounting a slitting cutter.
Slitting	Self-explanatory
Throwaway tip	A replaceable part of the nose of a throwaway tool whose shank and nose are separated.
Control axis	An axis controlled by a numeric control (NC) unit.
Coolant flow rate detection	A function which monitors the coolant flow rate and issues an alarm when the flow rate becomes equal to or lower than the setting.
Coolant level detection	A function which monitors the amount of remaining coolant and issues an alarm when the coolant level becomes equal to or lower than a specified level.
Absolute/incremental input	Absolute input means that an input value is used as is. Incremental input means that an input value is used added to the original value.
Zero-cut	A method of cutting a workpiece with the same infeed amount twice during thread cutting. This method minimizes parts which are left uncut because of cutting resistance.
Preselection (of tool)	Some machines are equipped with more than one tool post. Selecting of the next tool in advance while a tool on an another tool post is machining a workpiece.
Turning	Basic operation of a lathe which turns the outer circumference of a workpiece.
Center support	Supporting of the tip of a relatively long workpiece while the workpiece is being machined. The center support prevents the workpiece from vibration.
Lathe	A machine tool which rotates the bar material to be machined and turns the material by pushing the tool to the material.
Speed clamp	A speed clamp value is determined for each axis (motor) by maintaining the feed rate within a specified range.
Stock gripper device	A temporary chucking device used when a material is re-gripped.
Queuing point, tool queuing point	A point at which a tool stands by for a workpiece. A tool is replaced through this point.
Opposite tool post	A tool post which is opposite to the spindle, as is the case for the L series. This tool post is used mainly for drilling.
Die	An external threading tool.
Dial gauge	Instrument consisting of a gear mechanism which magnifies microscopic displacement and measures length and displacement precisely. Generally, a dial gauge graduated in 1/100 mm divisions is used. A high-precision dial gauge is graduated in 1/1000 mm divisions.
Multi-axis control group program queuing	Queuing of multiple axis control groups in order to maintain synchronization with each other.
Tap	An internal threading tool.
Bars on the shelf	Materials on the stock shelf of a bar loader.

Term	Description
Tab key	A key which moves the cursor to the column skipping a specified number of positions. The cursor moves to the tab setting position at a time when the tab key is pressed unlike when multiple spaces are entered.
Dowel	A protrusion left at the center of the cut-off face of a workpiece when cut-off machining is performed.
Turret	A cylindrical (prismatic) tool post for mounting a tool. This tool post is indexing for selecting a tool.
Carbon steel	Steel whose carbon content is high. (Example: S45C)
Setup	Overall preparation for actual machining.
Chasing	Thread cutting.
Chasing function	A control function of an NC unit which can perform thread cutting.
Chip conveyor	Machining workpieces leaves chips in the machine. A chip conveyor is a device to carry out the chips from the machine.
Chamfering	Machining to round edges and corners.
Carbide tool	A tool made of cemented carbide (e.g., WC, Tic, and CBN).
Superimpose	Axes which were operating with different coordinates as reference can be operated with the coordinate of a basic axis. The superimpose function synchronizes the axes with each other to enable the operation.
Long sleeve	A type of a sleeve or rotary tool which is longer than the standard size in order to grip a short tool.
Linear interpolation	Linear moving to the command-specified position while the feed rate is kept constant.
Earth current	Electric leakage.
Tooling	A generic name of holders for various types of cutting tools.
Tool setting	A function which adjusts a tool mounting error.
Tool set data	An error adjustment value set by the tool setting function.
Tool presetter	A device for setting a tool (which is to be mounted on the turret) onto the holder outside the machine.
Tool holder	Tooling for holding various types of cutting tools.
Tool layout pattern	Information about tool mounting positions at a holder. This information is equivalent to tool pitch information.
Cut-off	A process in which a workpiece is cut to the product length.
Cut-off tool	A tool used in the workpiece cut-off process.
Cut-off tool breakage detection	A function which detects a cut-off tool breakage by a workpiece left without being cut off.
Rating	Most suitable output, voltage, current, and speed that are determined for electric devices.
Distributor	Generally referring to a power distributor. A part for branching a coolant flow can also be interpreted as a distributor.

Term	Description
Default	Initial setting.
Taper	Status in which two faces form an angle. Taper in turning a workpiece indicates a conical shape.
Deflag	A system tool (software) which optimizes the hard disk by relocating fragmented data in the hard disk in order to speed up file access.
Electromagnetic switch	An electromagnetic contactor equipped with a thermal relay (overcurrent detector).
Electromagnetic contactor	A contactor using electromagnetic force. This contactor controls the turn-on/turn-off of motors and so on.
Door lock	A function which locks the machine door during operation. The door is locked generally when the start button is pressed.
Dwell	A function which stops program execution for specified time. This function is specified with the G04 command.
Synchronization	Controlling of more than one axis (e.g., the spindle and tool spindle, or moving axes) at the same time.
Synchronous rotary guide bushing device	A guide bushing device which can rotate synchronously with the spindle.
Synchronous tap	A function which performs tapping while synchronizing the spindle or rotary tool with a moving axis (X or Z axis).
Simultaneous thread cutting	Performing simultaneous thread cutting with multiple tool posts, or program for simultaneous thread cutting with multiple tool posts.
Dog	A part for operating a limit switch.
Tommy hole	A hole into which a bar tool on the outer circumference of a nut and so on is inserted.
Trouble guidance	A service which permits users to access the troubleshooting database at Citizen Machinery Miyano Co., Ltd. If a failure occurs, search the database for the cause of the failure and action to be taken.
Draw bar	A part for drawing a guide bushing.
Boring	Turning for an inner diameter with a tool.
Long workpiece device	A device for machining a long workpiece which exceeds the Z axis stroke. This device is used for preventing the tip of a workpiece from vibration.
Thread cutting canned cycle	One of canned cycles for turning. This function enables the specification of a shape with only a single block whereas a shape is generally specified with several blocks. The function is useful for simplifying a machining program.
Network option	Optional software which can be purchased through the Internet.
Network machine tool	A machine tool which enables data exchange through the Internet. The machine is called NMT.
Knock-out	Pushing-out of a workpiece chucked by the back spindle in order to eject the workpiece.

Term	Description
Non-guide bushing	A lathe which is not equipped with a guide bushing device.
Hard disk	Large-capacity storage of a computer. The Cincom system software and machining programs are stored in the hard disk.
Tool bit	An alias of a cutting tool.
Byte	One byte consists of eight bits. One byte can express 256 (= 2^8) statuses.
Center of tool	A tool nose position for a workpiece. The infeed direction is called a diameter, and the center of tool is in the direction at the right angle to the infeed direction. Generally, the center of tool indicates the difference between the centers of the tool and workpiece.
Tool nose radius compensation function	When a tool having a round nose is used and the R value of the tool nose is set, this function automatically calculates an error and makes compensation for the error.
Background	A process which operates in non-interactive mode under a multitask operating system whereas a process called "foreground" operates in interactive mode. Generally, the background has a lower process priority level than the foreground when it is set.
Backlash	A play of a transmission device such as a gear.
Tool post	A generic name of posts (gang tool post, 3-spindle tool post, and turret) for mounting tools.
Last part	A function which executes the last program to perform back machining in order to prevent workpieces without back machining from remaining in the machine.
Balance cut	Machining of both sides of a material with two tools.
Woodruff drill	A drill made by the following method: a half of the finished shaft diameter, which is equivalent to the drilling diameter, is machined flat, and a cutter is attached on it. The cutting force of this drill is weak, but the drill is accurate to perform drilling.
Fretting	Status of the surface of a turned workpiece.
Pick-off	A workpiece which is chucked on the main spindle passes to the back spindle.
Flat drill	A drill made of a round steel which has a flat tool edge.
Abnormal surface	Status in which a machined workpiece has a tear or stripped pattern on its surface because of poor fretting.
Firmware	Software installed in hardware.
Format	Initializing of disks. All data is erased from a disk when the disk is formatted.
Multiple canned cycle	A function which executes a prepared canned cycle with the 1-block command. Canned cycles include a finish machining cycle and longitudinal rough machining cycle.
Milling	Machining of a flat with a rotary tool such as an end mill.
Program check	A function which checks programs with the machine. The on-machine check function enables users to correct a program by operating the handle in the forward direction and/or reverse direction.
Program	A sequence of process methods written by combining commands.

Term	Description
Program zero point	Work zero point used when a machining program is executed. Generally, the end face of a workpiece is defined as the work zero point.
Block	Program commands written on a line.
Block skip	A function which disables a block specified with a slash (/) in a program when the program is automatically operated. Generally, this function is enabled with the button on the operation panel.
Discharge resistor unit	A drive unit generates energy. A discharge resistor unit is provided for discharging the energy, converting it into heat, and radiating the heat.
Interpolate	Moving of multiple axes to the command-specified position while keeping the feed rate constant.
Maintenance	Work for maintaining the machine in good condition.
Compensation number	When an amount of compensation (offset) is given to a tool in a machining program, a number with which the amount of compensation is registered is called a offset number.
Bobbin	A part of the chuck mechanism that transmits force from the stationary part to the rotary part of the spindle in order to open/close the collet chuck.
Hob	A cylindrical tool for gear cutting. A hob is used for machining spur gears and helical gears.
Polygon	A shape consisting of multiple sides. Polygon machining shapes a workpiece into a square or hexagonal product with a cutter having several tool noses. The workpiece and cutter rotate at a constant rate during polygon machining.
Feed per rotation	A method of feeding an axis proportionally to the spindle speed. For the feed rate, specify the move distance per rotation.
Microindicator	A type of a dial gauge which is graduated in divisions of 1µm or less.
Feed per minute	A method of feeding an axis proportionally to time. For the feed rate, specify the move distance per minute.
Macro	A command code or program which consists of machining programs in combination so that it can be specified in a block.
Milling interpolation	Interpolation for profile control by converting programmed commands into linear-axis move and rotary axis move in an orthogonal coordinate system (e.g., X-Y plane).
Tear	Status of the surface of a machined workpiece that appears to be torn.
Main breaker	A device for turning on the machine. The main breaker can shut down the power.
Chamfering	An operation to break sharp edges or corners.
Modal	An object which is enabled once it is specified. For G codes, a specified G code is enabled until another G code of the same group is specified.
Morse taper	A standard taper having a determined angle which is used with taper shanks of tools.
Model No.	A symbol indicating the machine version and model. A red sticker indicating the model number is attached on the front side of the machine. (Example: 2M3)

Term	Description
Nominal diameter	A hole diameter (applicable material diameter) of a chuck or guide bushing.
Last program	A program which is enclosed between G999; and N999;. Generally, back machining is written in the last program. When the machine is equipped with the Last Part function, the function executes the last program.
Thread lead	A dimension in the axial direction when a full turn is given to a thread.
Rego chuck	A chuck used with a tool such as a drill or tap. Rego chuck is a trademark of Regofix Co., Ltd. A Rego chuck has many cut lines on the outer circumference. In comparison with a collet chuck, the Rego chuck chucks a tool in a wide range for the nominal diameter.
Rotary guide bushing device	A guide bushing device equipped with a rotary function.
Workpiece	A product produced through the complete machining processes from a bar or blank.
Workpiece coordinate	A coordinate system used for executing a machining program. Generally, the end face of a workpiece is defined as the zero point, X axis is in the diametrical direction, and Z axis is in the longitudinal direction.
Workpiece separator	A device for collecting workpieces with which machining is completed.
Word	A program is written with words. A word consists of alphanumeric characters. It begins with an alphabetic character followed by several numeric characters.

Product Code	C	–	C	I	N	C	O	M	_	I	G	B		
Document Code	4	E	1	–	1	0	0	0						