Computerized Numerical Controller (CNC)

The Basics of CNC Machining Programming (for Machining center)

This course is intended for the persons who operate the machine tools such as machining center or milling machine with CNC (computer numerical control) equipment.

This course provides you with the basic knowledge required when working with the machining center and milling machine.

Click the Forward button in the upper-right of the screen to proceed the course.

This course describes the basic knowledge of CNC machining program and how to use the common instructions to the CNC control.

The goal of this course is to help you to be able to prepare the basic machining program with referring the manuals of your CNC or machine tool.

The contents of this course are as shown below.

Chapter 1 The basic knowledge for programming

The terms and commands you should know before learning CNC programming

Chapter 2 Positioning and interpolation

The basic instructions needed to move axes

Chapter 3 Coordinate system

Concept of coordinate system on machine tool

Test

Score to pass: 70% or more

Go to the next page	▶️	Go to the next page.
Back to the previous page	◀️	Back to the previous page.
Move to the desired page	TOC	"Table of Contents" will be displayed, enabling you to navigate to the desired page.
Exit the learning	✕	Exit the learning.

■ **Safety precautions**

We recommend using MITSUBISHI CNC software for training (NC Trainer2) to verify your CNC machining program operation.
NC Trainer2 allows you to check how your program runs within your PC environment but without actual machine.

In case you attempt to verify your program with machine tool, be educated about the operations and safety of your machine tool and be thoroughly familiar with your machine tool prior to using.

Note that the CNC instruction format may vary from one MTB to the next because they have to meet them to their own specific features of the machines.
Thus the CNC functions and command formatting shown in this course may NOT be identical to the ones used on your machine tool.
In this case, please give the first priority on the manual issued by your machine tool builder.

Chapter 1 The basic knowledge for programming

This chapter discusses the terms and commands you should know before learning CNC programming.

Machining center is a type of machine tool that rotates the tool, installs the workpiece (the stock to be machined) on the table, and performs the operation such as drilling or cutting.

Machining center is composed of the devices such as the table to fix the workpiece, the spindle motor to rotate the cutting tool, the servo motors to move the axes to the start position of machining, and the equipment to change tools automatically (ATC, Automatic Tool Changer) as shown below. Tools are stored in the tool magazine of ATC.
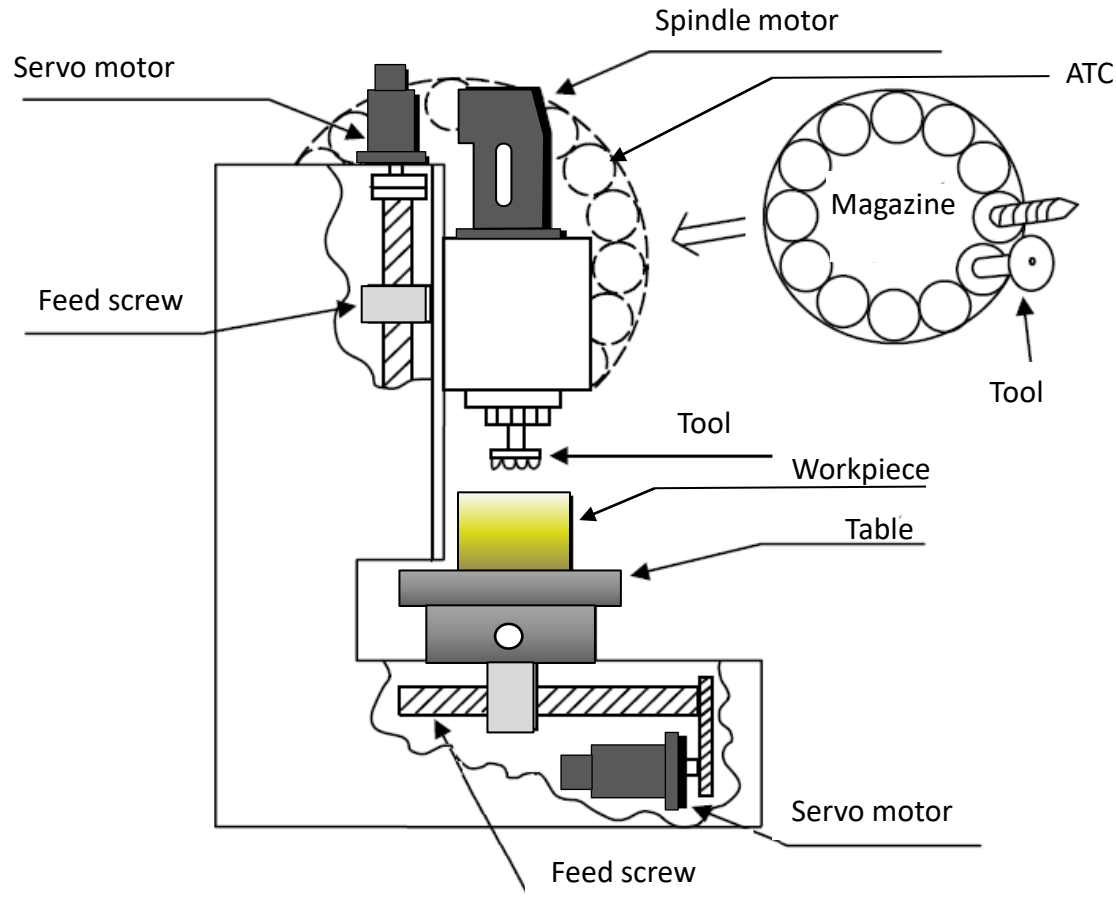
Machining center has the coordinate system to designate the positions for machining the workpiece.

For general machining center, the **right and left** direction is called **X axis**, **forward and backward** direction as **Y axis**, and **up and down** direction as **Z axis** as shown below.

Y axis
+ direction of motion

Z axis
+ direction of motion

Z axis (+)

Y axis (+)

X axis
- direction of motion

X axis
+ direction of motion

Y axis
- direction of motion

Z axis
- direction of motion

X axis (-)

X axis (+)

Y axis (-)

Z axis (-)
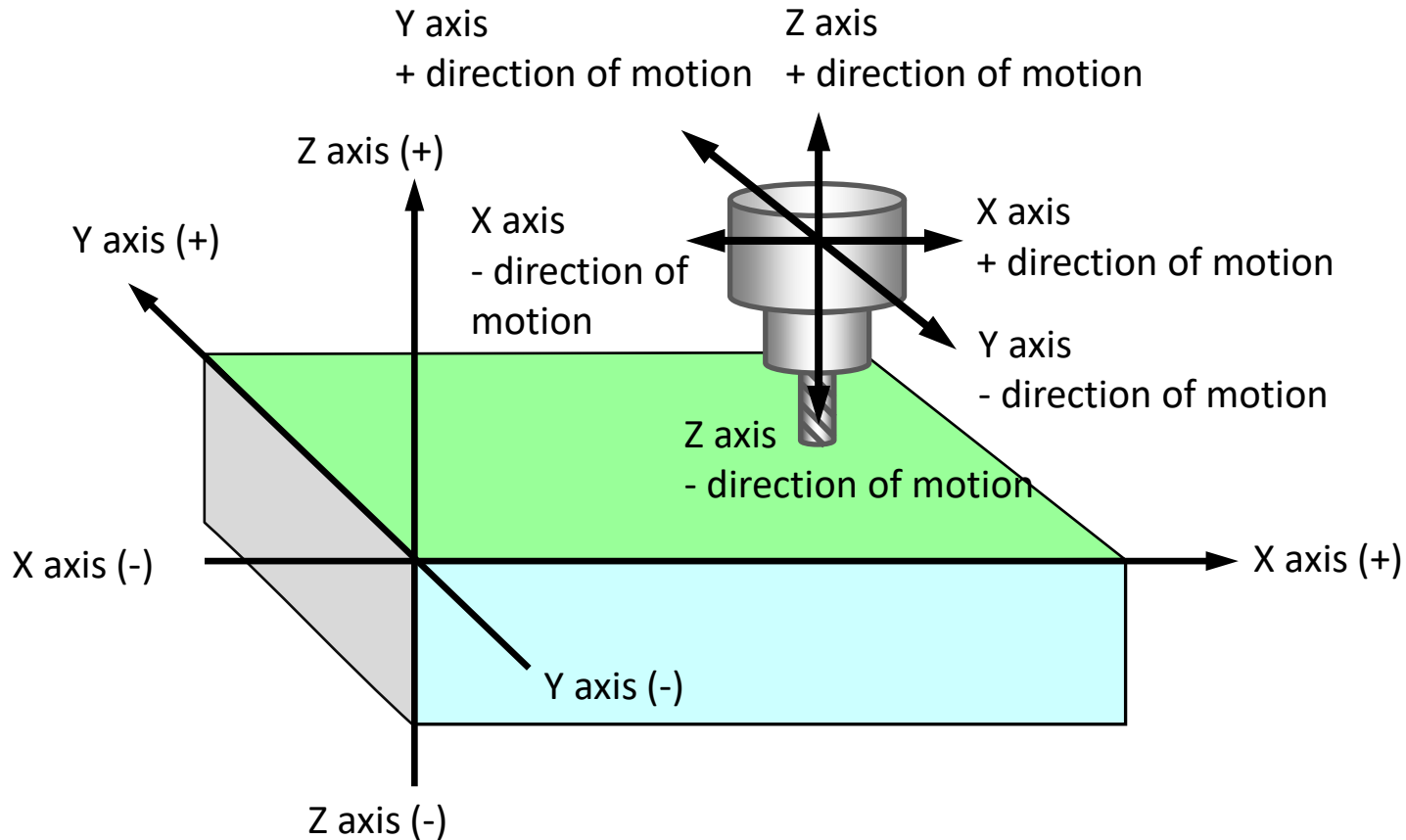
Machining program is a measure to tell the CNC how and in which order it makes the machine tool move.

The primary unit of CNC command that constitutes the machining program is made in the combination of an alphabetical letter and the numerical value.

This primary unit (alphabetical letter + numerical value) is called a "**word**".

The alphabetical letter at the head of a word is called an "**address**".

Word

G28

X-1.234

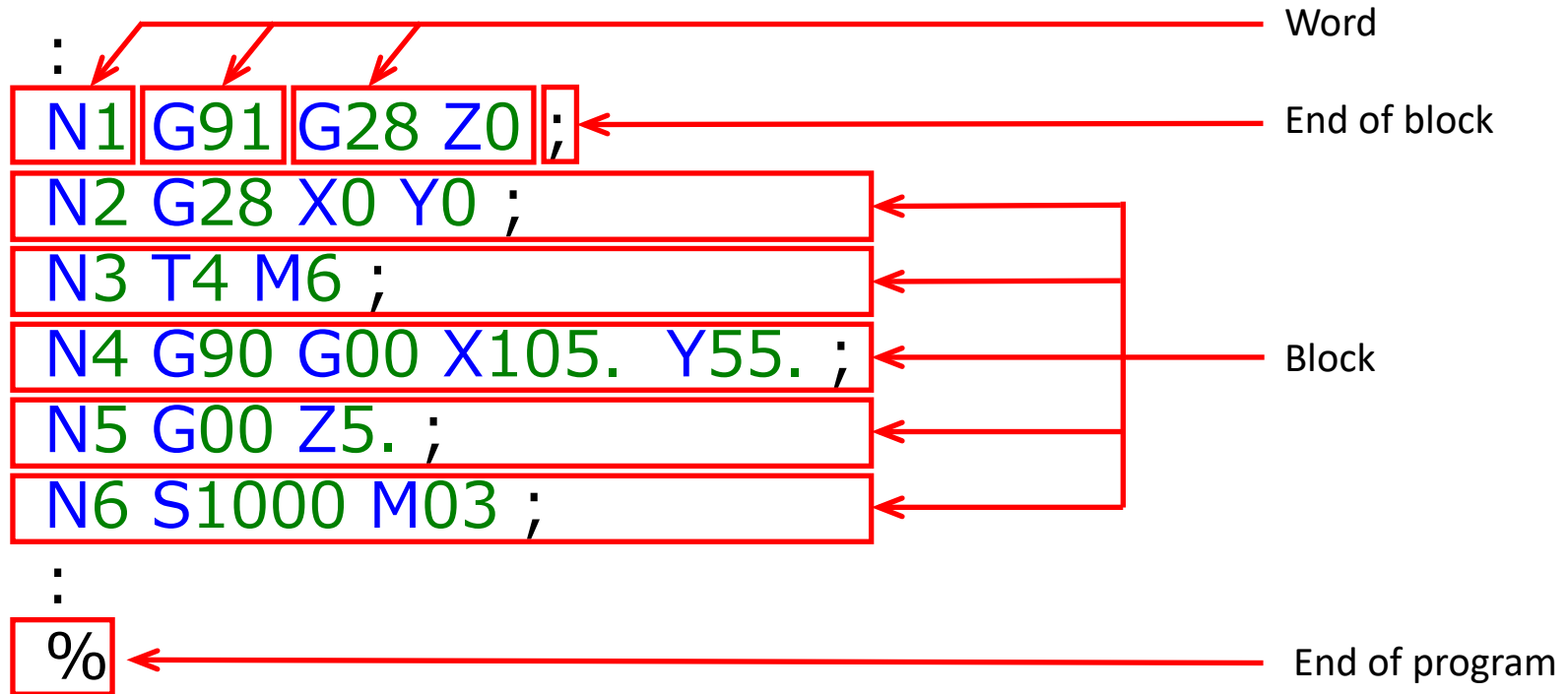Alphabet
(**Address**)   Numerical value
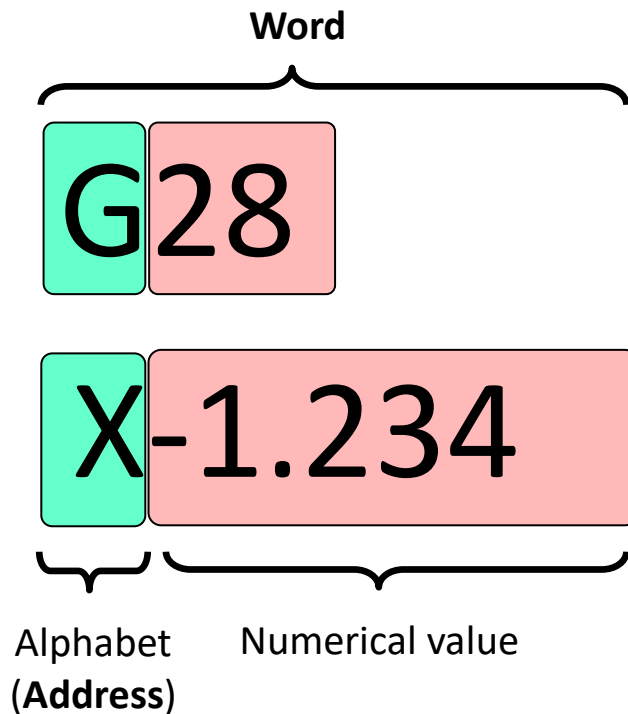
In the formatting of machining program, one or more "word" (command) constitute one line.
The sign of new line at the end of a line is shown as ";" on CNC display screen.
One line of machining program is called a "block".
Write "%" at the end of the machining program.

The machining program will be executed block by block.

:

N1 G91 G28 Z0 ; — Word

— End of block

N2 G28 X0 Y0 ;

N3 T4 M6 ;

N4 G90 G00 X105. Y55. ; — Block

N5 G00 Z5. ;

N6 S1000 M03 ;

:

% — End of program

Let's discuss the various "address" that composes each word prior to looking into the primary unit of machining program "word".

Address has the specific names and meanings as shown below.

Example of Address

Address	Applications
N	Block line identification
G	Preparatory-purpose command
X,Y,Z	Point designation to which the axis moves
F	Feedrate/speed designation of the axis
S	
T	Tool Designation
M	Miscellaneous function

Word

G28

X-1.234

Address   Numerical value
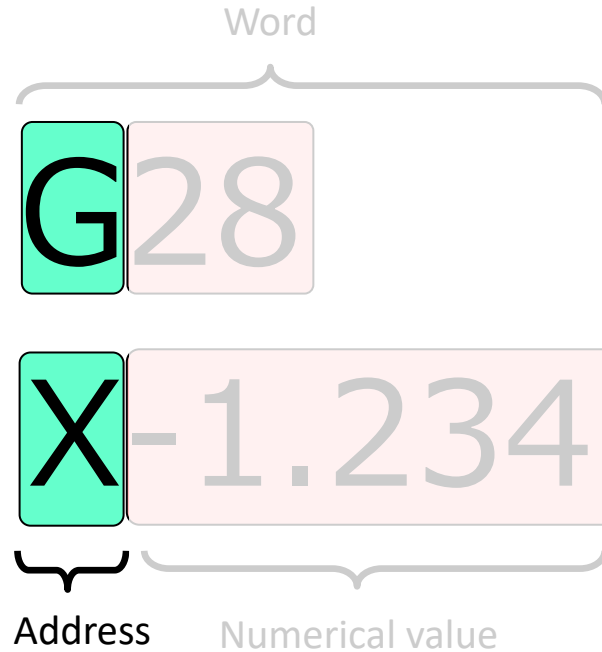
Next, let's discuss how to use the "word", the combination of address letter and the numerical value.

"N" is the address to be used to visually identify a certain machining content (block) by the programmer/operator.

"N" is called as "**Sequence number**" when accompanied by the numerical value.

This sequence number can be used to identify **the location of the executing block** of the program.

The display screen of the CNC control shows the status of the CNC operation.

When you run a program, the sequence number of currently running block will appear on the display.

Note that the sequence number on the screen is retained even after it is executed, so the latest sequence number the control ran will appear even while the control is running other blocks with no sequence numbers (N).

N1 G91 G28 Z0 ;
N2 G91 G28 X0 Y0 ;
N3 T4 M06 ;
N4 S2000 M03 ;
N5 G90 G00 Z50. ;
⋮
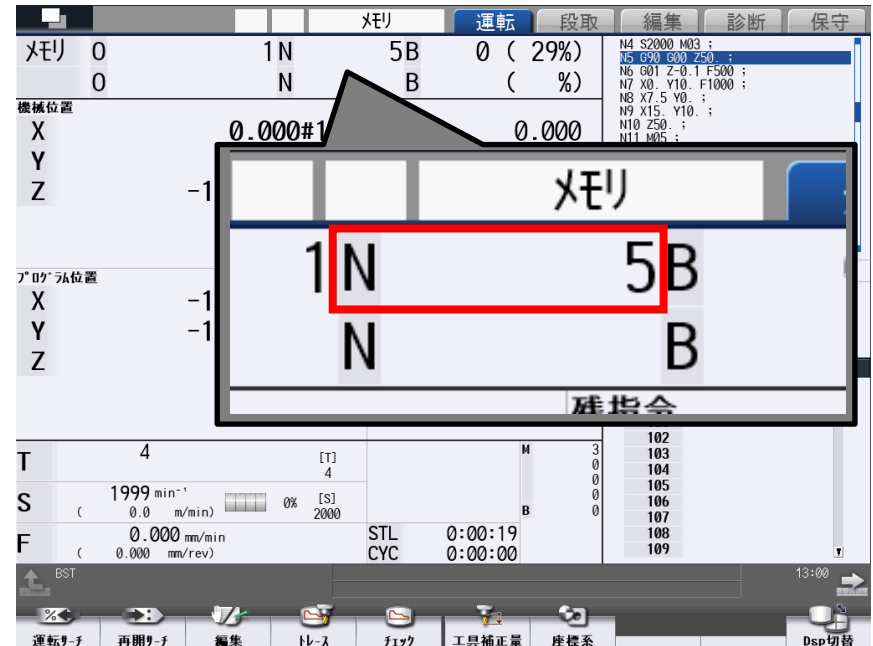
* You do not have to use the sequence number in every block.

"G" is the letter address to be used to tell the control preparatory-purpose command.

The word "G" accompanied by a numerical value is called preparatory function, G function, or G code(s).

There are the types of G codes as shown in table below, and they have group **for each function**.
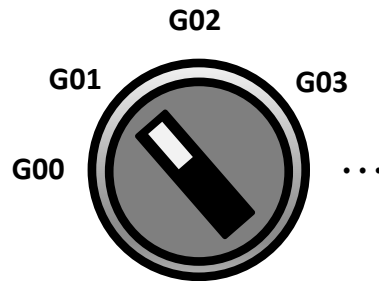
Example of G code

G code	Group	Description
G00	01	Positioning (Rapid motion command)
G01	01	Linear interpolation (Cutting feed command)
G02	01	Circular interpolation (Clockwise)
G03	01	Circular interpolation (Counterclockwise)
:	:	:
G04	00	Dwell
:	:	:
G17	02	Plane selection, X-Y
G18	02	Plane selection, Z-X
:	:	:
G90	03	Absolute mode
G91	03	Incremental mode
:	:	:

G codes can be roughly classified into two. Let's see this when using switch analogy.

Modal command

The G codes **other than group 00**, such like group 01 or 02, are to be used to restate the mode within the same group. These G codes are called **Modal command**.

You do not have to restate the modal command except when you need to change the modal.

Group01: Specify the preparatory function

G code	Group	Description
G00	01	Positioning (Rapid motion command)
G01	01	Linear interpolation (Cutting feed command)
G02	01	Circular interpolation (Clockwise)
G03	01	Circular interpolation (Counterclockwise)
:	:	:

Non-modal (unmodal) command

G code in **Group00** is only valid within one block. These G codes are called **Non-modal (unmodal) command**.

Non-modal (unmodal) command must be repeatedly written for every block even when commanding the same instructions.
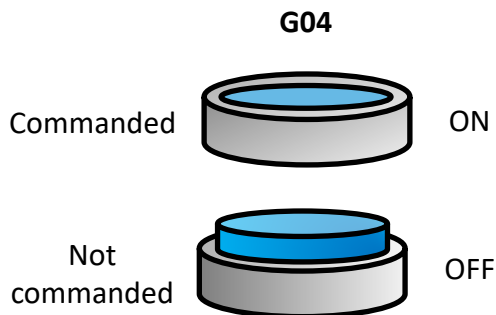
G code	Group	Description
:	:	:
G04	00	Dwell

* G04 (Dwell) is a non-modal (unmodal) G code that provides the dwell period for the axis move.
See 2.8 for the detail of dwell command.

1.7   Spindle function (S code)

"S" is the letter address to be used to tell the control the spindle rotation speed.
The word "S" accompanied by a numerical value is called spindle function, or S function.

The numerical value after the letter address is to specify the angular velocity of the spindle (rev/min) (number of revolutions of spindle per minute).

When the speed of spindle is 500 revolutions per minute (500min^{-1}), write as shown below.

S500 ;

The spindle cannot start rotation only by writing the S command.
To make the spindle start the rotation, add M03(clockwise rotation) or M04(counterclockwise rotation). (See 1.9 for detail of the command using address letter M.)
To instruct the spindle speed and the start of clockwise rotation within one block, write as shown below.

S500 M03 ;

To stop the spindle rotation, command M05.

M05 ;

"T" address letter is used to select the tool stored in the tool magazine of ATC.
The word "T" accompanied by a numerical value is called the tool function or T code.

The numerical value after the address letter "T" corresponds to the tool number.
Once this address is commanded, the tool having the corresponding tool number is selected.
The tool currently installed on the spindle will be replaced with the designated tool in the tool magazine specified by the automatic tool change command (M06).
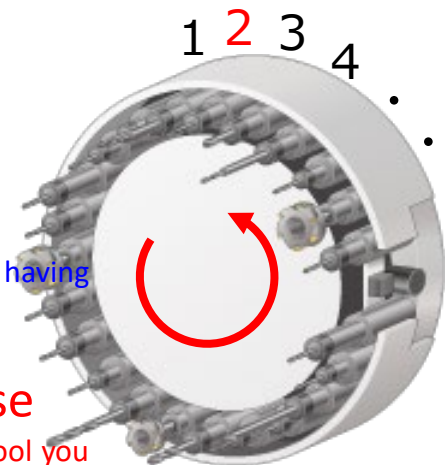
T02 M06 ;

<Tool exchange command>
... Replaces the current tool with the tool having specified tool number.

Tool number you want to use
... Corresponds to the tool number of the tool you want to use.

1 2 3 4

"M" is the address used to instruct the CNC control to perform miscellaneous functions of the machine tool such like the ON and OFF of spindle (CW/CCW), turning on/off the coolant.
The word "M" accompanied by a numerical value is called the miscellaneous function or M code.

There are the types of M codes as shown in table below.

Example of M code

M code	Function	Description
M00	Program stop	Halts the execution of the program.
M01	Optional stop	Halts the execution of the program. You can choose not to halt the program execution with the corresponding switch on the machine operation panel.
M02	End of program	Tells the control the end of execution of the machining program.
M03	Spindle on (clockwise)	Turns on the spindle rotation **in clockwise**.
M04	Spindle on (counterclockwise)	Turns on the spindle rotation **in counterclockwise**.
M05	Spindle stop	Turns off the spindle **rotation**.
M06	Automatic tool change	Replaces the currently installed tool on the spindle with the tool stored in the tool magazine.
M30	End of program	Tells the control the end of execution of the machining program.

* The machine behaviors and function assigned to each M code may vary from one machine tool to the next.
You must check in the machine tool builder's manual to determine a machine's behaviors and function prior to using M code.

Though this is not categorized to "word" discussed to this point, you can use "**comment**" function when you want to add the description to describe the meaning or notes in your machining program.

The comment can be used to inform the other people of what the program is intended to do.
So it can be a help when you review your program later, or even when you read the program prepared by the other programmer.

The description enclosed in the parentheses "(" and ")" is regarded as the comment. You can use alphabet, numerical value, and symbols to write the comment.

You can use whole of one line to write a comment. You can also add comment after writing the commands.

```
N1 G91 G28 Z0 ;
N2 G91 G28 X0 Y0 ;
(COMMENT-1) ;
N3 T01 M06 (COMMENT-2) ;
N4 S1000 M03 ;
    :
```

Chapter 2 Positioning and interpolation

In this chapter, we will discuss the basic instructions needed to move axes.

For the machining of workpiece, it is necessary to move the tool to the intended position.
In CNC program, the axes move on the machine tool is instructed by the designation of both **the target axes to move** and **the destination** of the axes.
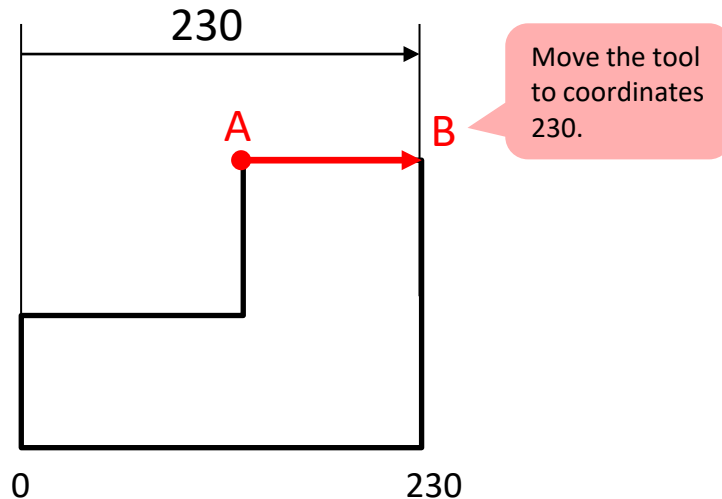
There are two kinds of positioning mode;
- "**Absolute mode**" to specify the destination with the coordinates.
- "**Incremental mode**" to specify the amount of motion relative to the previous position.

In general, we choose the best mode depending on how the dimensions on a drawing are specified.
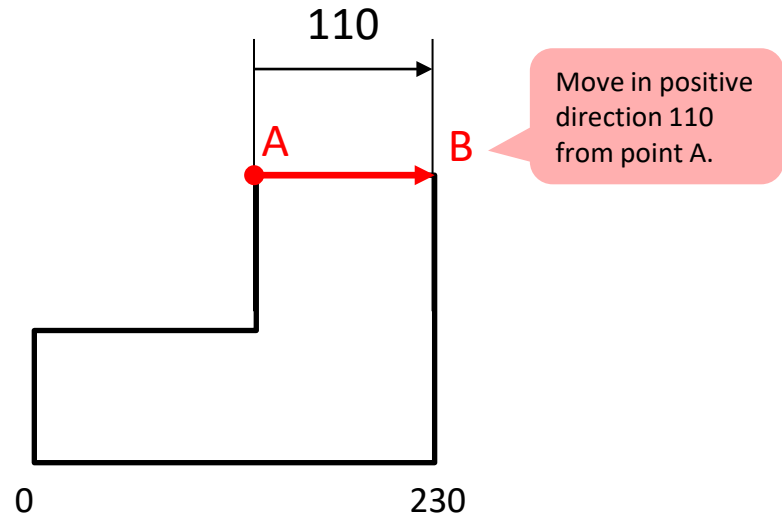- Absolute mode: When the destination is being specified with its **coordinates**.
- Incremental mode: When the destination is being specified with the **amount of motion relative to the current position**.

Following shows how to determine which mode to choose when you move the tool from point A to point B.

Absolute mode

230

Move the tool to coordinates 230.

A → B

0   230

Incremental mode

110

Move in positive direction 110 from point A.

A → B

0   230

To specify the destination of positioning, use address X, Y, and Z discussed in Chapter 1.
Next, let's look at the detail of how to command position.

When using **absolute mode**, write the address letter(s) (X, Y, and Z) after **G90** to specify the coordinates of the destination.

The machining program shown below is the example when moving the tool from point A to point B, whose coordinates is (X, Y)=(20.0, 5.0).
In absolute mode, the command to be used is identical in both cases either when moving the tool from point A to point B or when moving from point A' to point B.

G90 X20. Y5. ;
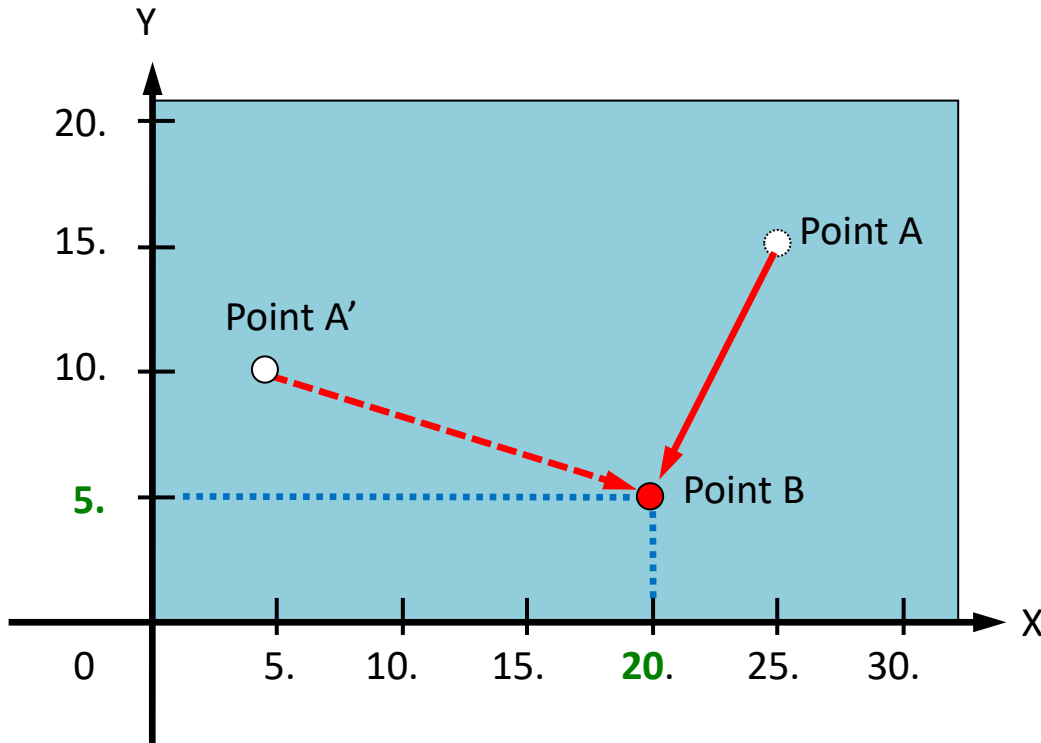
When using **incremental mode**, write the address letter(s) (X, Y, and Z) after **G91** to specify the amount of motion.

The machining program shown below is the example when moving the tool from point A (X, Y)=(5.0, 10.0) to point B (X, Y)=(25.0, 5.0).
You must use the address letters corresponding to the X-axis and Y-axis, and also the amount of motion (Coordinates of point B - coordinates of point A)=(25.0--5.0, 5.0-10.0)=(20.0, -5.0).
When the direction of motion is negative, add minus (-) symbol to the amount of motion.

In incremental mode, the destination is specified by the amount of motion relative to the position at commanding, thus the command (G91 X20. Y-5) moves the tool to point B when it is told at point A, but moves the tool to point B' when it is told at point A'.
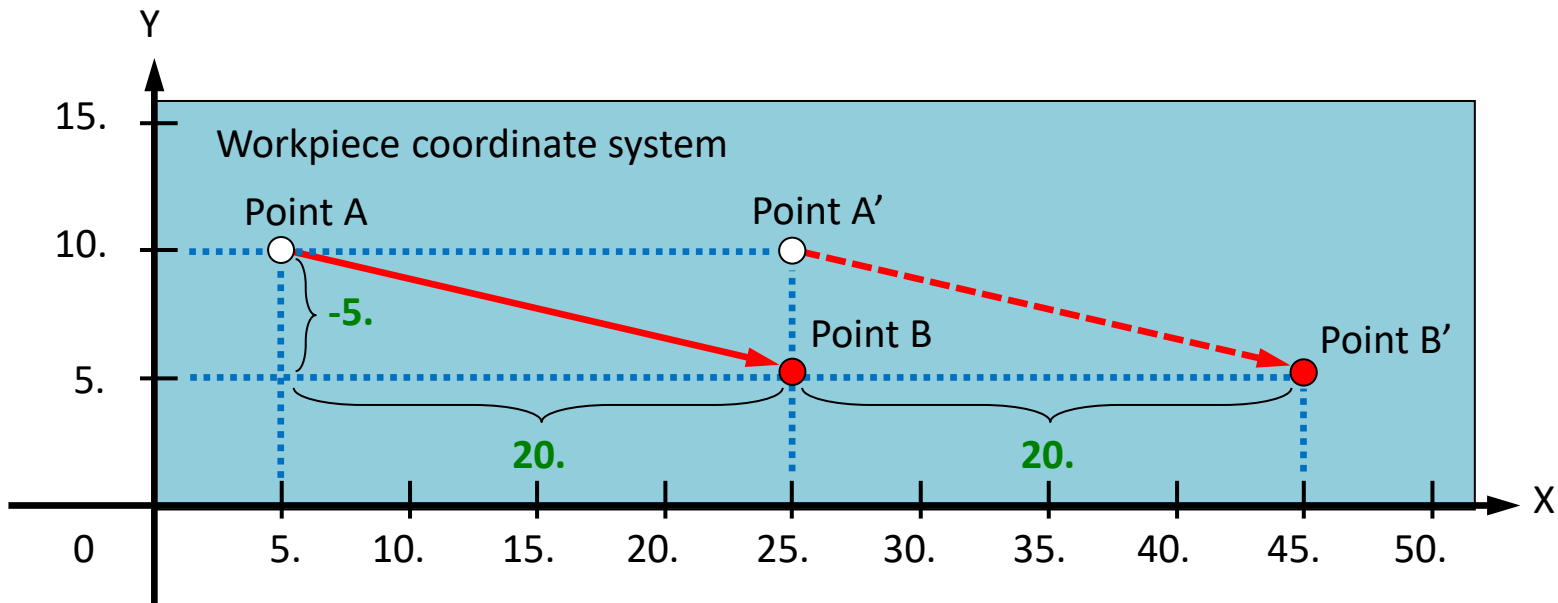
CNC machine tool distinguishes in which unit the command are told, depending on the **use of decimal point** in the numerical value after the address (such like X, Y, X, and F) being used to specify the amount of motion, coordinates, angle, period of time, or speed.

- **With** decimal point: in **mm**
- **Without** decimal point: in **µm**

The drawings are commonly written in mm, so **add decimal point** when preparing your machining program.

- With decimal point

  X123. is identical to X123000. (Corresponds to the position 123.000mm in X axis.)

- Without decimal point

  X123 is identical to X0.123 in X axis. (Corresponds to the position 0.123mm in X axis.)

One of the most common failure in the machining program is to leave out the decimal point.

If the CNC control is told "X10" despite the programmer wanted to write X10.000, the CNC control behaves as the position 0.010mm in X axis.

The tool moves to the specified position by X, Y and Z within the command, and the type of motion is in accordance with the G codes.

When you want to rapidly move the tool without interfering with the workpiece, such as in the case the tool moves to the starting point of machining on the workpiece, use G00 (Positioning) command. G00 is a command to move the tool to the specified position at rapid rate and is also called rapid positioning command.
* The rapid rate is determined in the machine tool builder's manual. You must check in the machine tool builder's manual for detail.
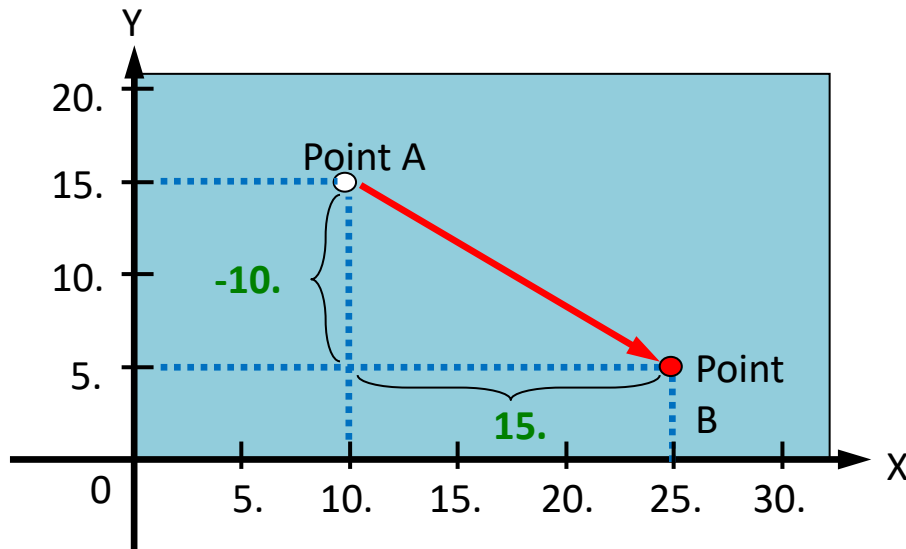
Together with G00, it is necessary to designate the destination point using X, Y, and Z.

When to run rapid positioning from point A to point B, tell the machine as shown below.

Absolute mode: **G90 G00 X25. Y5. ;**
Incremental mode: **G91 G00 X15. Y-10. ;**
                              Positioning   Destination

To perform cutting by the linear axis motion on the workpiece, G01 (Linear interpolation) command will be used.

Together with G01, it is necessary to designate the destination point using X, Y, and Z. Then the tool moves along a straight line toward the destination point specified.
In G01 command, it is also necessary to specify the **feedrate (F)** to be applied during the axis move, in addition to the instruction to move to the axes.
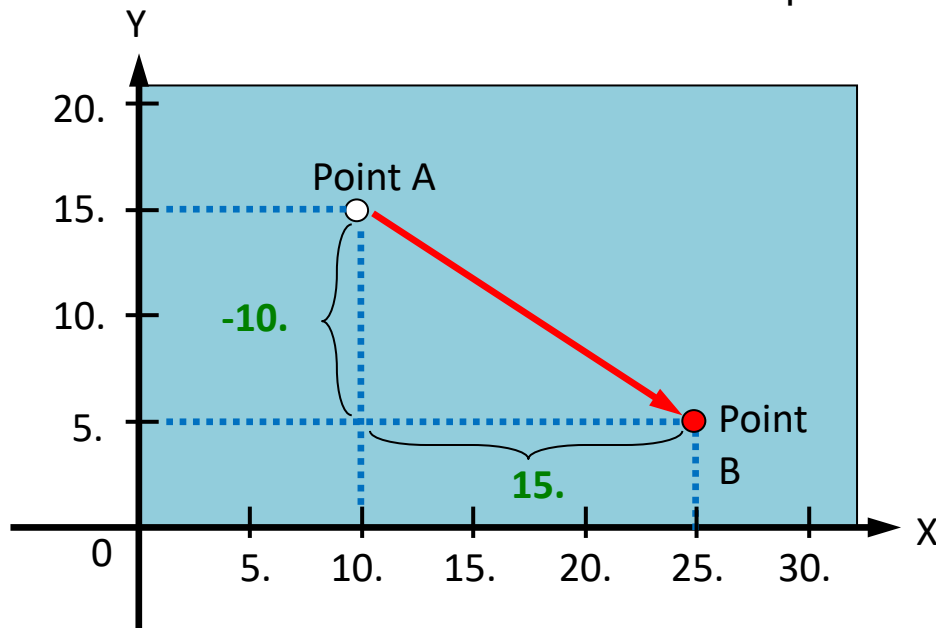* Details of feedrate will be discussed in 2.5.

When to run cutting operation from point A to point B along a straight line, tell the machine as shown below.

Absolute mode: **G90 G01 X25. Y5.   F100. ;**
Incremental mode: **G91 G01 X15. Y-10. F100. ;**

Linear interpolation Destination   Feedrate

During G01 (Linear interpolation) presentation, we discussed that the G01 is used when the axis moves while performing the cutting operation, and mentioned that the federate applied in the cutting operation must be instructed.
To specify the feedrate, use the address letter F and the distance per minute (mm/min) (also called feed per minute).

To specify the feedrate of G01 at 100mm per minute, write as shown below.

$$\text{G01 X-10. F100. ;}$$

The numerical value specified together with the address F is retained once it is specified.
So you can omit the feedrate underlined as shown below when you would like to use the same feedrate during the consecutive commands.

		Feedrate
G01 Y-10. F100. ;		
G01 X10. Y-20. F100. ;		100mm/min
G01 Y-30. F100. ;		
G01 X20. Y-10. F150. ;		150mm/min

For the cutting operation in a straight line motion, we adopted G01 (Linear interpolation).
Now let's discuss the command that is used for the cutting operation along a circular motion.

To perform cutting by the circular axis motion on the workpiece, G02 or G03 (Circular interpolation) command will be used.
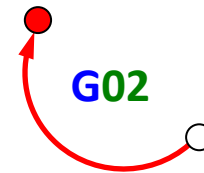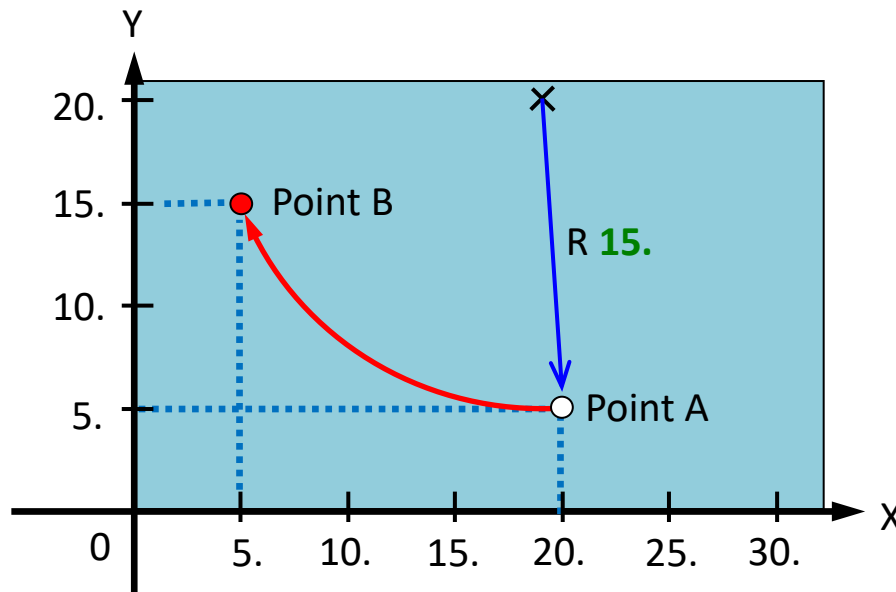G02 can run machining motion in the form of a clockwise (CW) arc path, on the other hand G03 in the form of a counterclockwise (CCW) arc path.

As with G01, both G02 and G03 require the destination point (X, Y) and feedrate (F).
Additionally, circular motion requires the radius of the arc being specified by programmer, using address letter R.
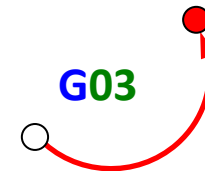
To run CW circular cutting operation from point A to point B at the radius of 15.0[mm], tell the machine as shown below.

G90   **G02**   **X5. Y15.**   **R15.**   **F100.** ;
Absolute mode Clockwise   Destination   Radius   Feedrate

This G code is used to provides a waiting for the axis move with giving a dwell period.

For example, it is necessary to temporary pause the axis motion during performing cutoff operation, to prevent the insufficient cutting at the bottom of the cutoff, or to remove the remaining material.
The command to be used in such case of machining is called **Dwell command (G04)**.

Let's discuss the case that runs the drilling using G01 command.

When the consecutive G01 commands (linear interpolation) exist, the CNC control decelerates as the axis reaches close to the position specified in the first block and prepares for the acceleration for the next block as shown in figure (A) to smoothen the axis motion at the point between the two blocks.

As the angle made by the two blocks gradually decreases as shown in (A), (B) and (C), the axis starts the motion of the second block without reaching the specified position in the first block.   Thus the machine behavior shown in (C) induces an insufficient drilling.

But when the Dwell command (G04) exist between the two blocks, you can avoid this kind of insufficient drilling.

G01

G01

Insufficient drilling

G01

G01

(A)

(B)

(C)

(D)

Boundary between
two blocks

G04
Not commanded

G04
Commanded
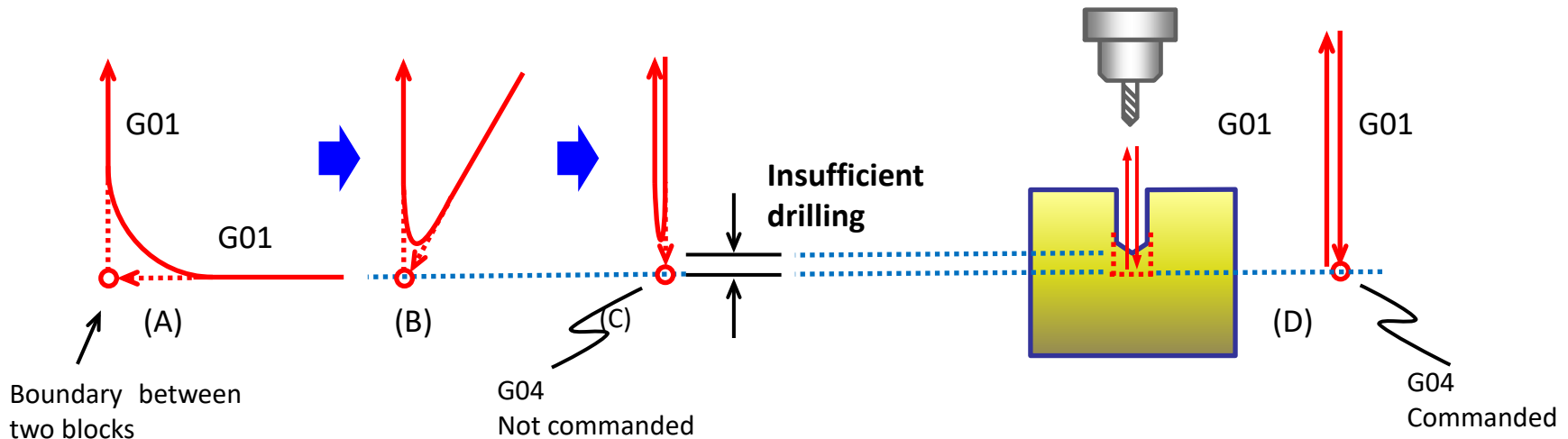
Dwell (G04) causes the axis to pause for a specified period.

The dwell period is specified using the address P or X.
- When using **address P**, the period is specified **in 1/1000 sec**. No decimal point in this value.
- When using **address X**, the period is specified **in sec**. **Use decimal point in this value.**

When specifying the 1.5 sec pause in dwell command, tell the machine as shown below.

With address P: **G04 P1500** ;
With address X: **G04 P1.5** ;

G01   G01

:
G01 Z-10. F100. ;
G04 X1.5 ;
G01 Z50. ;
:

Pause for 1.5 sec
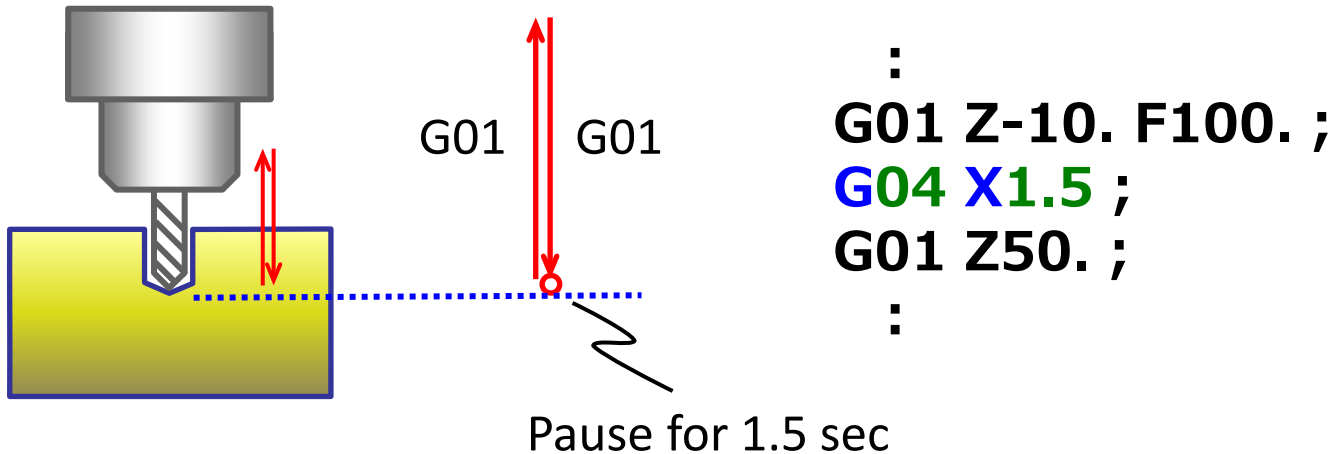
The coordinates specified as the position in the command corresponds to the position at the tool tip.
For the length of the tool (tool length) varies from one tool to another, the position of the tool tip must be compensated with the tool length compensation feature (G43).

The compensation amount to accurately position the tool tip at the specified position along Z axis must be previously determined as the compensation data for tool length.
The compensation data corresponding to the compensation number specified by address H in G43 is added to the position of Z axis being specified in the machining program.

$$\text{G43} \quad \text{Zz} \quad \text{Hh} \; ;$$

Compensation number
… Uses the compensation data corresponding to the number specified.

Z axis destination

Tool length compensation command
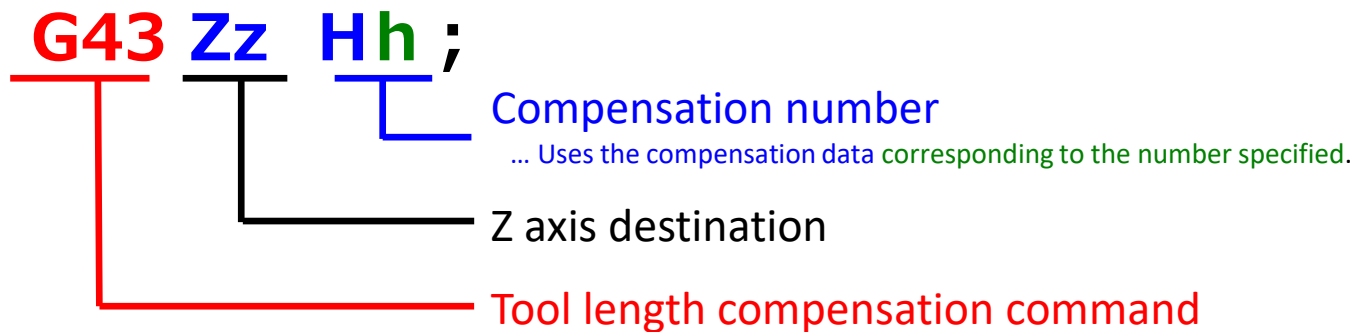
The compensation data for tool length compensation must be previously specified with the tool offset screen on CNC display unit.

Coordinate system

In this chapter, we will discuss the coordinate system on CNC machine tool.

3.1 Coordinate system

3.2 Workpiece coordinate system offset

3.3 Reference position and reference position return: G28

CNC machine tool has coordinate systems, called **Machine coordinate system** and **Workpiece coordinate system**.

The machine coordinate system is used to instate the positions specific to the machine tool.
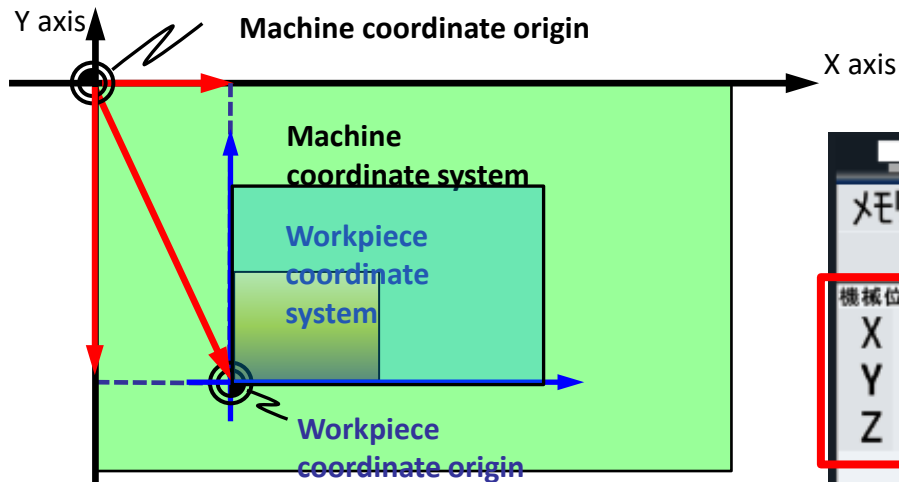The **"Machine position" counter** shown on the monitor screen of CNC unit displays the coordinates in the machine coordinate system.
The position at which the "machine position" counter displays zero is called **machine coordinate origin**.

The workpiece coordinate system is commonly used when writing the machining program.
The position (X, Y, Z)=(0, 0, 0) in the machining program is called **workpiece coordinate origin**.
The **"Workpiece coordinate position" counter** shown on the monitor screen of CNC unit displays the coordinates in the workpiece coordinate system.

Y axis

Machine coordinate origin

X axis

Machine coordinate system

Workpiece coordinate system

Workpiece coordinate origin

メモリ	0	1 N	0 B	0 (0%)
	0	N	B	(%)

機械位置
X　　　0.000
Y　　　0.000
Z　　　0.000

ﾜｰｸ座標 G00
X　　　0.000
Y　　　0.000
Z　　　0.000

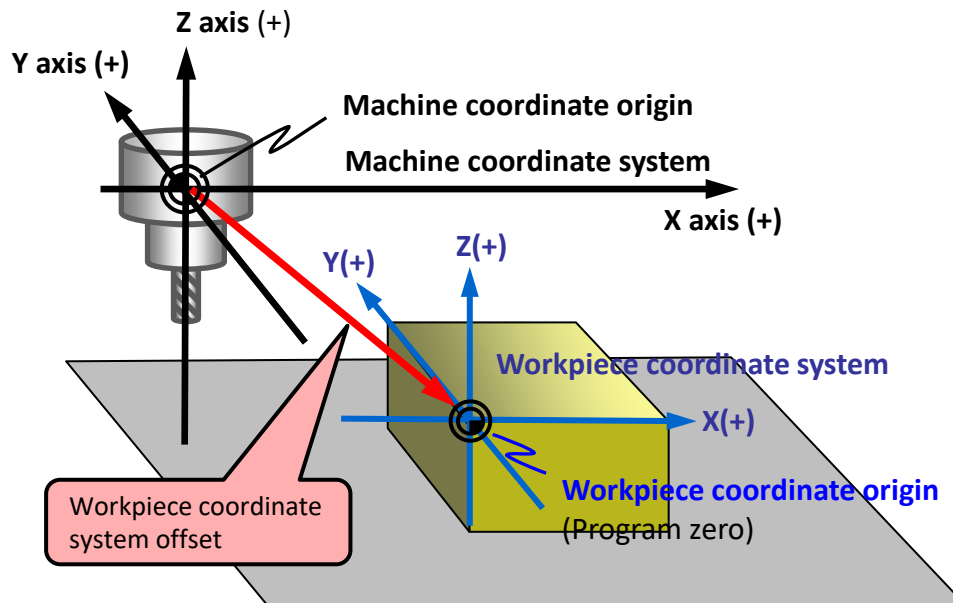Machine position counter　　Workpiece coordinate position counter

The position of workpiece coordinate origin in the machine coordinate system is specified with **Workpiece coordinate system offset**.

The workpiece coordinate system offset can be specified with the CNC unit's setting screen of the coordinate system offset.

**Setting screen of workpiece coordinate system offset**

Reference position is a specific point defined by a coordinates in the machine coordinate system, and is utilized as the point at which the machine is told as the start point of motion.

To move the tool to reference position, we use G28 (Reference position return).
By the axis letter address in absolute mode (X, Y, and Z) included in G28 command, you can specify the intermediate position (**intermediate point**) to which the axis will first move prior to moving to the reference position. This intermediate point is used to avoid unintended interference between the tool and the workpiece.

For example, the command shown below tells the machine to first move the tool from point A to (X, Y)=(200.0, 150.0) then to move the reference position.
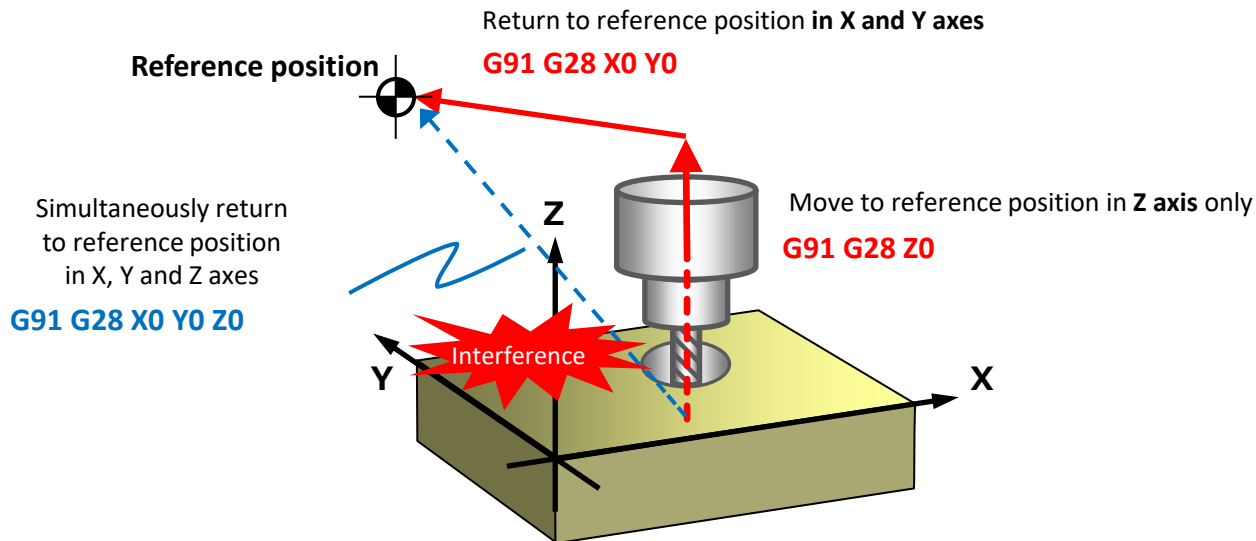
G90 G28 X200. Y150. ;
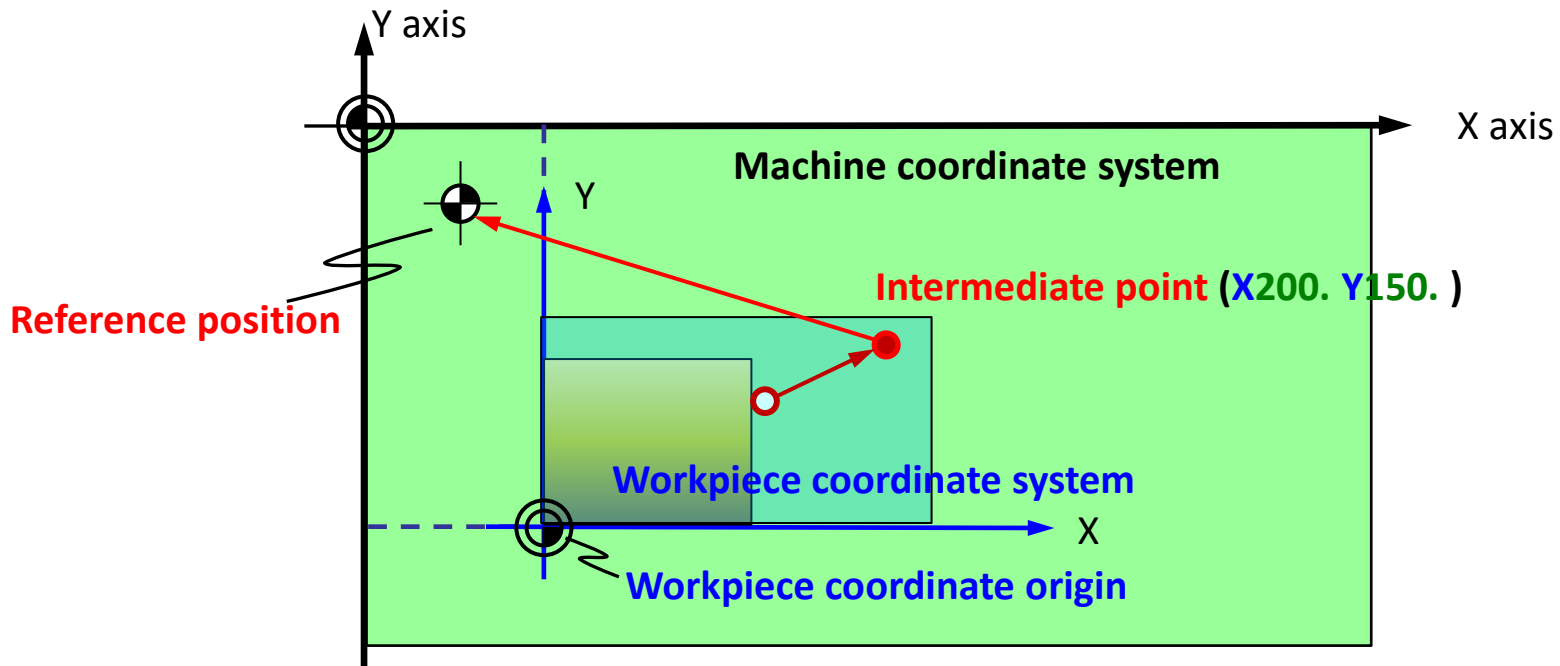
In case you would not like to specify the intermediate point, use zero in incremental positioning (X0, Ｙ0, Z0) in G28 to directly move the tool to the reference position. In this case, it is generally preferred to move Z axis first to avoid unintended interference between the tool and the workpiece.

After commanding only in Z axis, sequentially command in X and Y axes to return to reference position.

G91 G28 Z0 ;
G91 G28 X0 Y0 ;

Return to reference position **in X and Y axes**
G91 G28 X0 Y0

Reference position

Simultaneously return to reference position in X, Y and Z axes
G91 G28 X0 Y0 Z0

Move to reference position in **Z axis** only
G91 G28 Z0

Interference

Z

Y

X

To summarize the whole contents of this course, here is the example machining path and its corresponding machining program.
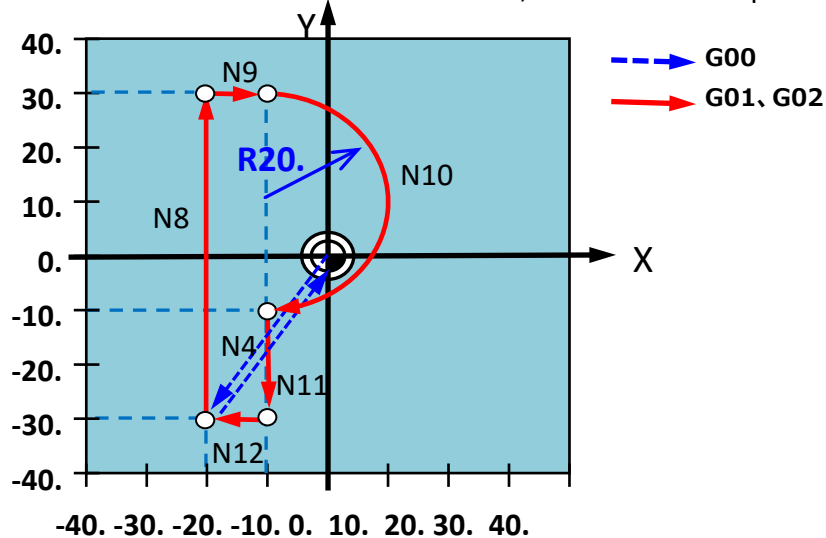
Program	Message
(REVIEW-PROGRAM) ;	This is the comment or notes in program. (1.10)
N1 G91 G28 Z0 ;	Moves the Z axis to reference position. (3.3)
N2 G28 X0 Y0 ;	Moves the X and Y axes to reference position. (3.3)
N3 T01 M06 ;	Selects the tool number at reference position. (1.8)
N4 G90 G54 G00 X-20.Y-30. ;	This is NOT the cutting operation, but the positioning command at rapid rate. (2.3)
N5 G43 G01 Z50. H01 F2000. ;	Runs tool length compensation operation. (2.8)
N6 S6000 M3 ;	Makes the spindle start the rotation prior to beginning cutting operation. (1.9)
N7 G01 Z-0.5 F500. ;	Cutting operation along the straight line. Feedrate required. (2.4)
N8 Y30. F1500. ;	G01 is the modal command, and not necessary to write again except when you need to change the modal.
N9 X-10. ;	F command can be omitted.
N10 G02 Y-10. R20. ;	Runs circular cutting operation. Specifying the radius is required. (2.6)
N11 G01 Y-30. ;	Cutting operation along the straight line.
N12 X-20. ;	
N13 Z50. ;	
N14 M5 ;	Turns the spindle off after the completion of cutting operation. (1.9)
N15 G91 G28 Z0 ;	Moves the Z axis to reference position. (3.3)
N16 G28 X0 Y0 ;	First moves the Z axis to reference position, then X and Y axes. This reduces the risk of interference with the workpiece.
N17 M02 ;	End of program (1.9)
%	End of program (1.3)

Now that you have completed all of the lessons of The Basics of CNC Machining Programming (for Machining center) course, you are ready to take the final test. If you are unclear on any of the topics covered, please take this opportunity to review those topics.
There are a total of 10 questions,(10 topics) in this Final Test.
You can take the final test as many times as you like.

How to score the test
After selecting the answer, make sure to click the **Answer** button. Your answer will be lost if you proceed without clicking the Answer button. (Regarded as unanswered question.)

Score results
The number of correct answers, the number of questions, the percentage of correct answers, and the pass/fail result will appear on the score page.

Correct answers :   **6**

Total questions :   **6**

To pass the test, you have to answer **70%** of the questions correct.

Percentage :   **100%**

Proceed	Review

- Click the **Proceed** button to exit the test.
- Click the **Review** button to review the test. (Correct answer check)
- Click the **Retry** button to retake the test again.

You have completed the Computerized Numerical Controllers (CNC)
The Basics of CNC Machining Programming (for Machining center) Course.

Thank you for taking this course.

We hope you enjoyed the lessons and the information you acquired in
this course will be useful in the future.

You can review the course as many times as you want.