MITSUBISHI ELECTRIC
Changes for the Better

Computerized Numerical Controllers (CNC)
Mitsubishi Electric CNC Fundamentals (Functions)

This course is for those who have completed Mitsubishi Electric CNC Fundamentals (Introduction) and explains the functions of NC and NC machine tools.

L(NA)00225ENG

Purpose of the course

The purpose of this course is to provide first-time learners of NC and NC machine tools with prerequisite knowledge necessary for writing machining programs.

As prerequisites for this course, you should have already completed the following courses or possess the equivalent knowledge.

- Mitsubishi Electric CNC Fundamentals (Introduction)

Course structure

The contents of this course are as follows.

Chapter 1 - How to use NC machine tools

Overview of the procedure for using NC machine tools

Chapter 2 - Specifications of NC machine tools

Functions and specifications of NC machine tools

Chapter 3 - Machining programs for NC machine tools

Axis control and selection commands for tools

Final Test

Pass grade: 60% or higher

Introduction How to use this e-Learning tool

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.

Cautions for use

Safety precautions

When you learn based on using actual products, please carefully read the safety precautions in the corresponding manuals.

Chapter 1 How to use NC machine tools

This chapter contains an outline of the procedure for using NC machine tools and several examples used in the course.

1.1 NC machine tool implementation to machining

The machining process from start to finish is outlined below.

1. **Selecting an NC machine tool**
   Select a model with specifications suited to the machining details.

2. **Using the NC machine tool**

   1. Turn on the NC machine tool.
   2. Attach the tool.
   3. Attach the workpiece.
   4. Enter the instructions for machining details.
   5. Perform machining.
   6. Turn off the NC machine tool.

1.2 Instructions given to NC machine tools

TOC

Proper instructions must be given to the NC machine tool to accurately machine a workpiece to the desired shape. Instruction details are divided into two major categories.

Parameters
These setting items determine the specifications and functions of the NC machine tool.
As shown below, there are two types of parameters.

- **Mechanical parameters** are intended for machine tool manufacturing designers, so they can integrate an NC unit into a machine tool.
- **User parameters** are intended for users, so they can operate the machine tool.

* Chapter 3 of this course covers user parameters.

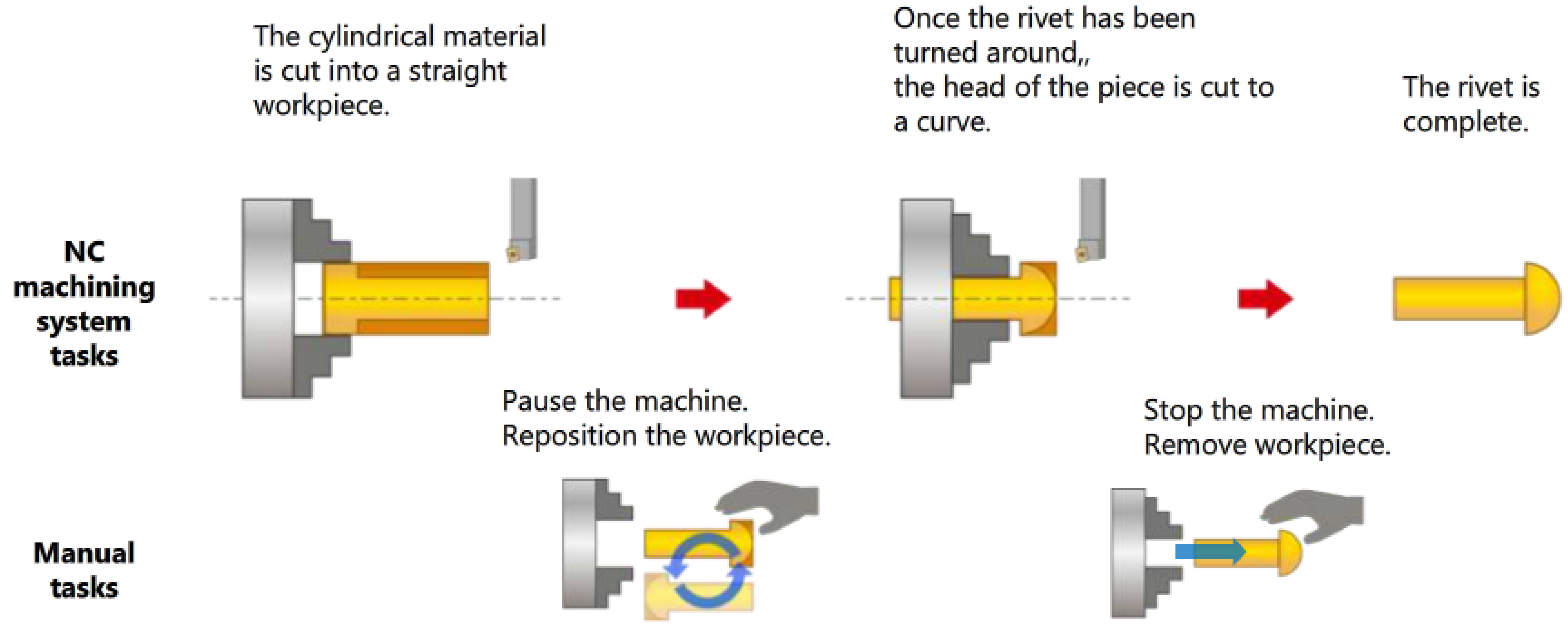
Machining program
This states the machining process and machining details.

The following pages explain the functions and specifications of NC machine tools and provide the information required to write a machining program.

1.3 Machining details

The procedure for using a lathe to machine a part with a screw-like shape (rivet) is shown as an example of simple machining. These machining details are used for explanations throughout this course.

To cut a metal block into a rivet, both straight-line and curved-line machining are performed.

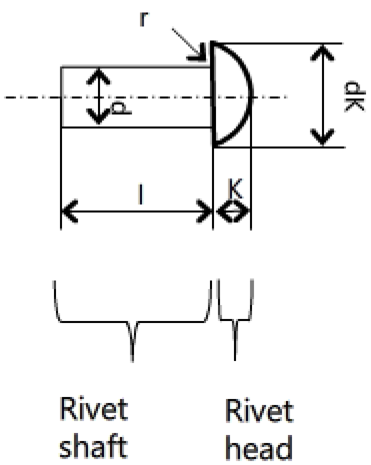


1.4 Machining programs

The machining diagram and machining program for the rivet explained on the previous page are shown below.
The machining program uses a combination of letters and numbers as shown below.
This program for the rivet is broadly divided into an **shaft cutting step** and a **head cutting step**.

Machining diagram

Unit: mm
d: 8.0
dK: 13.3
K: 5.6
l: 20.0
r: 0.2



Machining program

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
G01 Z-19.8 F100
G02 X8.4 Z-20.0 R0.2 F100
G01 X14.0 F100
M05
G30 U0 W0
T0202
M01
N0020 G55 G0  X0.0 Z0.0
M03 S1000
G01 Z-0.4 F100
G03 X13.3 Z-6.0 R6.748 F100
G00 X14.0
M05
G28 U0 W0
M30
```

Program number: 1000
Comment: RIVET

Sequence number: 0010
Cuts the shaft of the rivet.

Sequence number: 0020
Cuts the head of the rivet.

Details regarding the program's notations are explained in the e-learning Machining Program edition.
This course covers the relationship between NC machine tool functions, and commands and values in programs.

1.5 Summary

The contents of this chapter are:

- Overview of the machining process
- Types of instruction methods for machining

Important points to consider:

Instruction methods for machining	Divided into parameters and the machining program
Parameters	• Settings for determining the functions and specifications of NC machine tools • Mechanical parameters are intended for machine tool manufacturing designers, so they can integrate an NC unit into a machine tool, and user parameters are intended for users, so they can operate the machine tool.
Machining program	• Describes the machining procedure and the machining details. • Uses a combination of letters and numbers.

Chapter 2 Specifications of NC machine tools

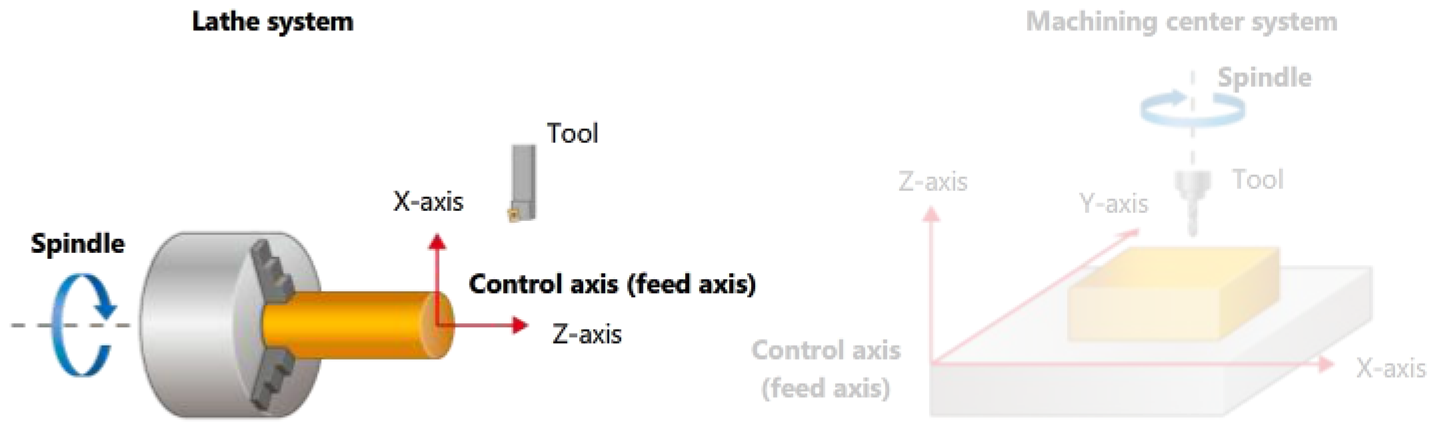
The functions and specifications of NC machine tools are stated below.
This information is necessary for selecting a NC machine tool.

2.1 Number of control axes

The specifications for the number of axes that are controlled by the NC unit are shown below.

Number of basic control axes

The number of basic control axes is the number of axes that are necessary for the machine tool to perform normal machining. As shown below, the number of axes differs depending on the type of machine tool. Explanations in this course focus on the lathe system.



For an NC machine tool in a lathe system, **two axes** (horizontal and vertical to the workpiece) are the basic control axes. In the program, these are expressed as the X- and Z-axes.

Maximum number of control axes

This is the maximum number of axes, including the basic control axes, that can be controlled by an NC unit. When creating complex workpieces, select a high-functionality NC unit with the highest number of control axes possible.

Number of simultaneous control axes

This is the number of axes that can be controlled at the same time. As the number of simultaneous controllable axes increases, more complicated shapes can be machined simultaneously.

2.2 Data units

To denote positions and movement amounts in the machining program, units must be specified.

Minimum command unit
When setting the NC unit's parameters or giving commands in the machining program, specify the movement amount, speed, and so on in mm.
Numbers to the right of the decimal point can be specified, and modern NC units can generally machine at the precision of 0.001 mm (1 µm).

The minimum command unit cannot be changed because it is determined based on the specifications of the NC unit.

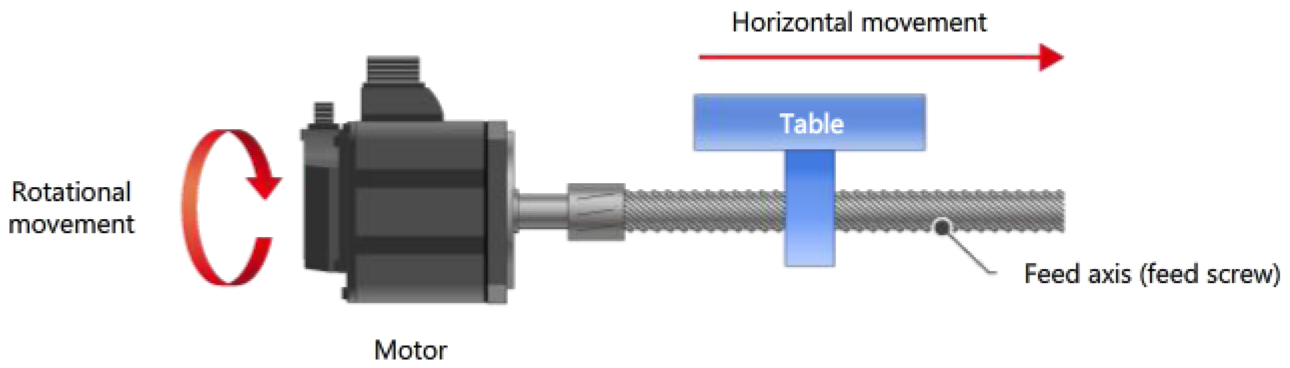
In the following example, the data is written using real numbers.

```
G03 X13.3 Z-6.0 R6.7485 F100
```

The "5" at the end of the number is discarded because the minimum command unit is 0.001 mm.

2.3 Movement amount conversions

To horizontally move (send) the table that holds the machining workpiece, for example, the NC machine tool converts the rotational movement of the motor into horizontal movement.
The movement amount is converted by the transfer mechanism.
The transfer mechanism in the figure below is a feed screw.



2.3.1 Feed rate and motor speed

The horizontal movement speed (**feed rate**) of the table is calculated as follows.

Feed rate = amount of **movement by a single turn of the motor** × **amount of rotations per unit of time (rotation speed)**

For example,

- If the motor moves the table or tool 10.0 mm per turn, this is equal to a feed rate of 10.0 mm per revolution (mm/rev).
- If the motor turns 1,000 times per minute, this is equal to a motor speed of 1,000 revolutions per minute (rpm).

Feed rate = 10.0 (mm/rev) × 1,000 (rpm) = 10,000 (mm/minute) = 10.0 (m/minute)

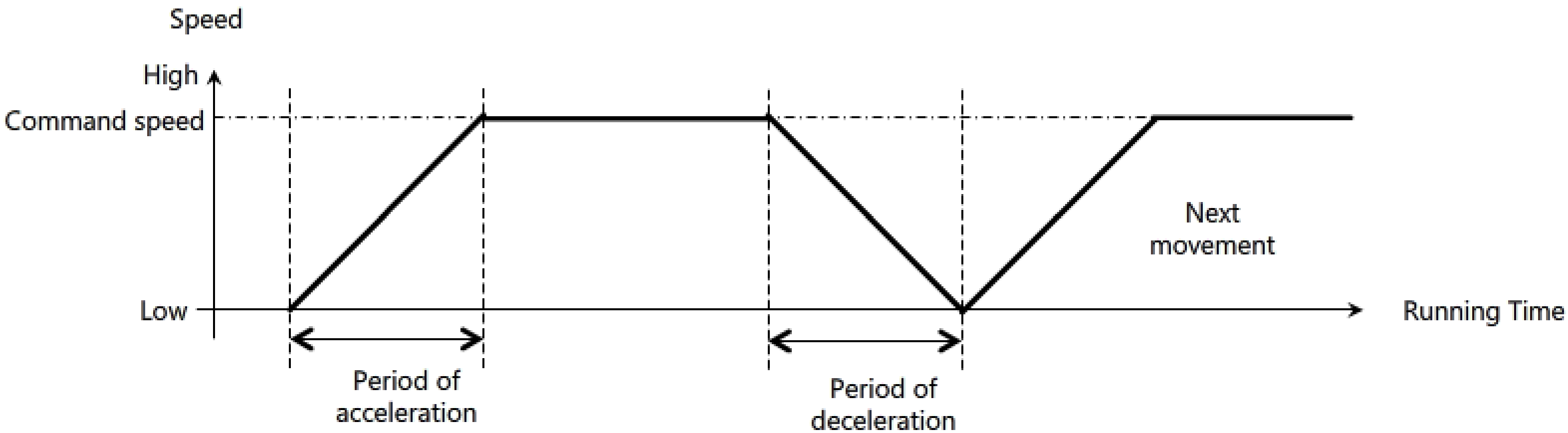
When considering the movement amount and feed rate of the tool or table in this manner, if a transfer mechanism is between the motor and the tool or table, the effect of the movement amount per single turn of the motor and the rotation speed must be considered.

2.4 Acceleration and deceleration time settings

Acceleration and deceleration

When the table and turret suddenly change from a stopped state to moving at the commanded speed, the machine tool cannot keep up.
This may cause malfunctions, shorten the lifespan of the machine tool, or break the tool.
Similar to automobiles, trains etc., the speed must change gradually when accelerating or decelerating.

Changes in speed are represented in the following relationship with running time.



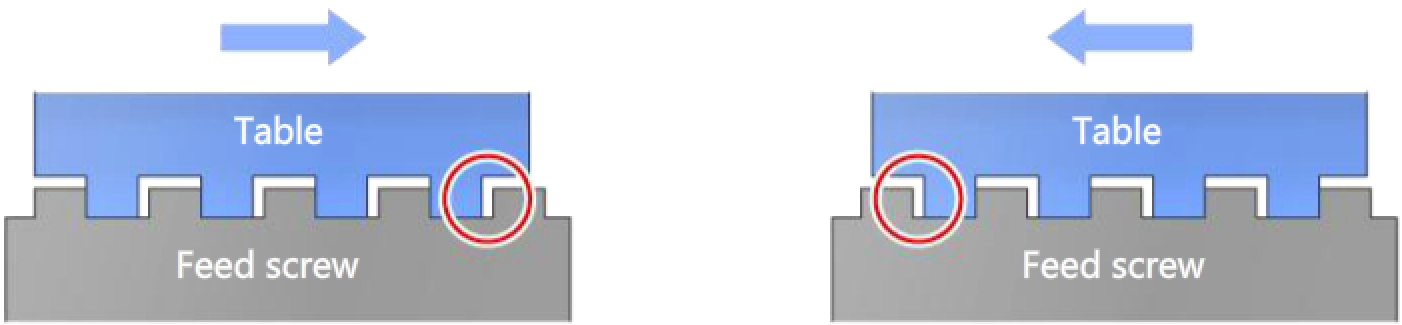
NC machine tools are instructed to accelerate by specifying the command speed and the time it takes to reach it.

2.5 Machine error correction

Errors occur in mechanical structures, such as transfer mechanisms.
Therefore, settings that account for mechanical errors are performed, so that the desired movement is achieved.

The NC machine tool rotates the servo motor and moves the tool via the transfer mechanism (mainly a feed screw).
Tools can be moved precisely to the desired position by correcting the position command to the servo motor in accordance with the characteristics of the transfer mechanism.

Backlash correction
There is a small gap between the feed screw and the table for smooth operation, which is called backlash.
Depending on the direction of movement (→), the direction in which backlash can occur changes, so when movement is reversed, the movement amount is corrected for only the amount of backlash.



Pitch error correction
With a feed screw, although the crest portions and root portions are precisely machined at a fixed interval (pitch), some errors do occur.
The function for correcting these errors is called pitch error correction.

These corrections are different for each machine, so when a machine is being set up, measurements are performed and stored into the memory of the NC unit.

2.6 Summary

The contents of this chapter are:

- Number of axes
- Command unit
- Conversion ratio for movement amounts
- Speed changes
- Error correction

Important points to consider:

Number of basic control axes	• The number of axes in a machine tool that are necessary for basic machining. • A lathe has two such axes.
Maximum number of control axes	• The maximum number of axes that can be controlled by an NC unit. • As the number increases, more complicated shapes can be machined.
Number of simultaneous control axes	• The number of axes that can be controlled at the same time. • As the number increases, more machining can be performed at the same time.
Command unit	• To denote positions and movement amounts, units of length, such as mm, must be specified.
Feed rate	• The tool or table is moved by converting the rotational movement of the motor into horizontal movement. • The movement amount and speed are affected by the conversion ratio of the conveyance mechanism.
Acceleration and deceleration	• Speed changes must be gradual because sudden changes in speed cause malfunctions and damage.
Correction of machine errors	• By accounting for mechanical errors beforehand, corrections can be performed to ensure proper movement.

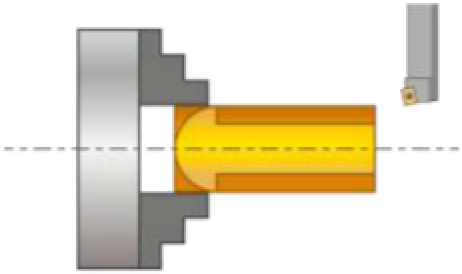
Chapter 3 Machining programs for NC machine tools

How the machining program gives instructions to the NC unit will be explained using the machining program for the rivet introduced in Chapter 1.

3.1 Machining program command details

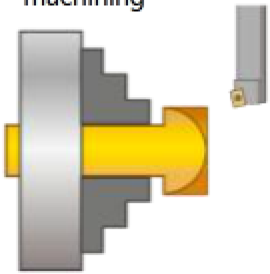
The machining details and machining program for the rivet mentioned previously, are shown again below. An overview of the command details is as follows.

Rivet shaft machining



Reposition the workpiece.

Rivet head machining



O1000(RIVET)

```
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
G01 Z-19.8 F100
G02 X8.4 Z-20.0 R0.2 F100
G01 X14.0 F100
M05
G30 U0 W0
T0202
M01
N0020 G55 G0  X0.0 Z0.0
M03 S1000
G01 Z-0.4 F100
G03 X13.3 Z-6.0 R6.748 F100
G00 X14.0
M05
G28 U0 W0
M30
```

Machining program command details

1. Withdraws the tool to a safe location.
2. Selects the tool used for machining the rivet shaft.
3. Rotates the spindle.
4. Moves the tool to the machining position.
5. Machines the rivet shaft.
6. Stops the spindle.
7. Replaces the tool with the one used for machining the rivet head.
8. Withdraws the tool to a safe location for the next machining job.
9. Rotates the spindle.
10. Moves the tool to the machining position.
11. Machines the rivet head.
12. Stops the spindle.

A detailed explanation is given from the next page.

3.2 Coding scheme used for programs

The first character in each line of the program must be a letter. This is called an address.
An operation command is given by a combination of the address and the numbers that follow.

G code
This is a general-purpose command. This is used mainly for moving the feed axis.

M code
This command is used mainly for controlling the spindle and cutting oil. This is called a miscellaneous function.

Xx.x, Zx.x, (Ux.x, Wx.x), Rx.x
Denotes the movement amount of an axis, along with the coordinates.
When indicating points on the X-axis and Z-axis by absolute values, Xx.x, Zx.x is used.
When indicating by incremental values, Ux.x, Wx.x is used.
Rx.x is a command for the radius of an arc.

T code
This command is used for selecting a tool.

S code
This command is used for the spindle speed.

F code
This command is used for the feed rate of the feed axes.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
G01 Z-19.8 F100
```

3.3 Reference point (origin) return

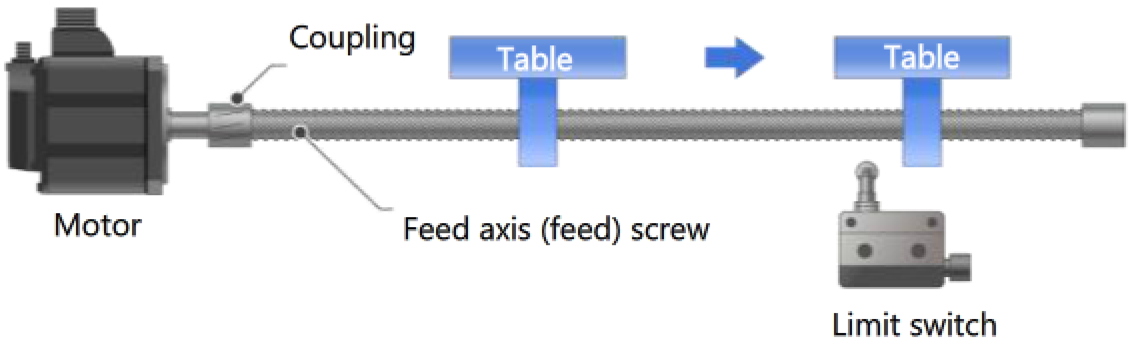
Reference point setting

It was explained earlier that the movement amount of an axis is denoted with coordinates. To specify coordinates, a coordinate base point (0, 0) must be determined. This coordinate point is called the reference point (origin).

At the time of purchase, an NC tool's reference point is not set.
Before machining, the reference point must be set. A basic way of setting the reference point follows.

Each axis of the NC machine tool is equipped with a limit switch that detects the position of the tool or table.
For the mechanism in the following diagram, after the power is turned on, the axis is driven manually, moving the table toward the limit switch.
When the table passes through the limit switch, the axis stops. This position is saved in the NC machine tool as the reference point.

Determining a reference point enables you to accurately denote the position of the tool and the table.



3.3 Reference point (origin) return

Reference point return

In recent NC machine tools, once the reference point is set, the setting is retained even if the machine is turned off. However, some models do not retain the setting. In such cases, the reference point must be set every time the machine is turned on. This is called a reference point return.

Reference point return by the machining program

In a machining program, the command code G28 moves an axis to the reference point.
This code is used to move the tool to a safe location during machining.

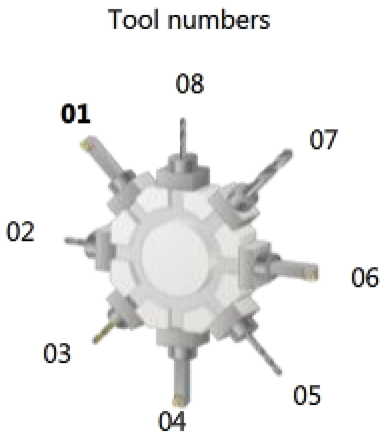
```
O1000(RIVET)
N0010 G28 U0 W0
```

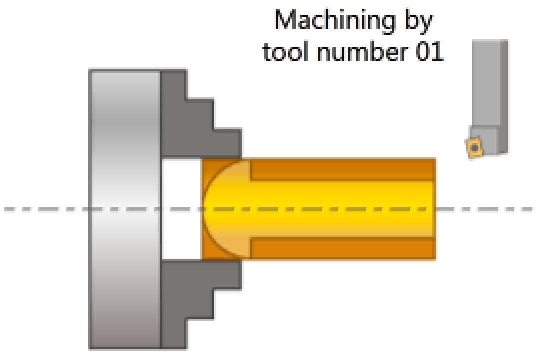
3.4 Selecting a tool

The tool to be used is selected with **T** and its following numbers.

The first two digits of the four numbers that come after T are tool numbers.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
```

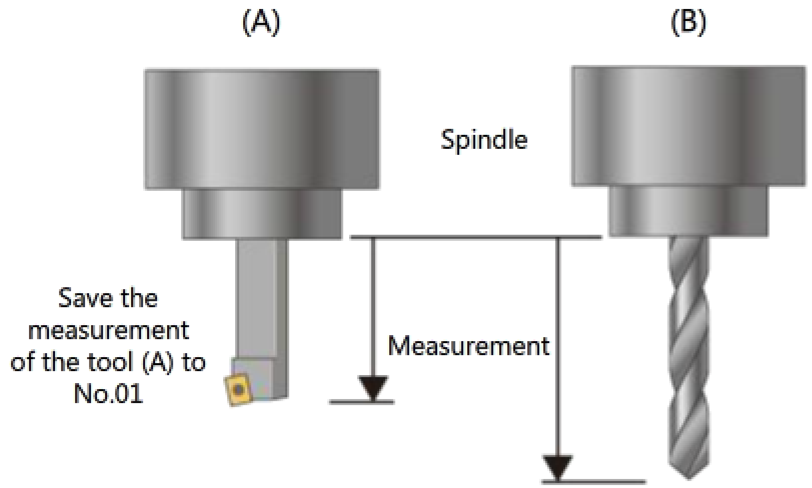




3.4.1 Tool length correction

Position compensation for the selected tool is necessary because the length of the tool differs depending on the tool.

The part of the tool that touches the workpiece during machining is the tip of the tool. This is why the length of the attached tool must be taken into account before machining.
For each tool, measure the length of the attached tool to its tip, and save the measurement to the NC unit's internal memory in advance.
The position including this measurement can be specified for the selected tool.



In the machining program, the data number where tool measurements are saved is specified in the last two digits of the four numbers that come after T.

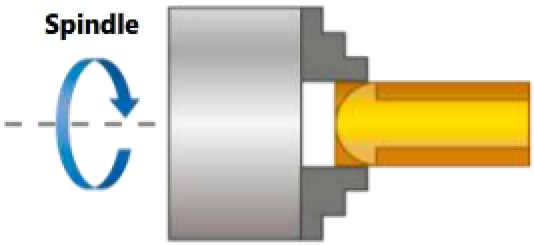
```
O1000(RIVET)
N0010 G28 U0 W0
T0101
```

3.5 Spindle rotation rate command

The rotation rate of the spindle that rotates the machining tool or workpiece is designated using **S** and its following numbers.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
```

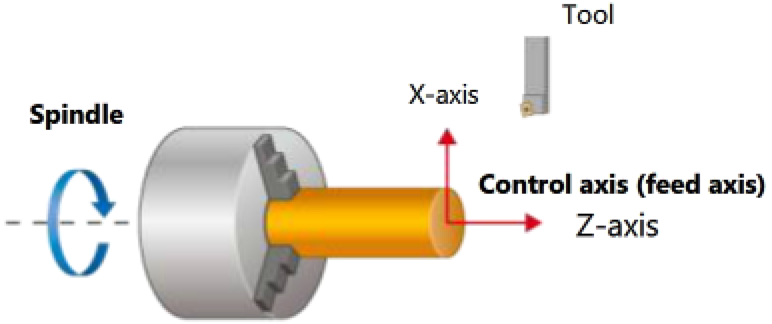
Rotation rate
1,000 rpm



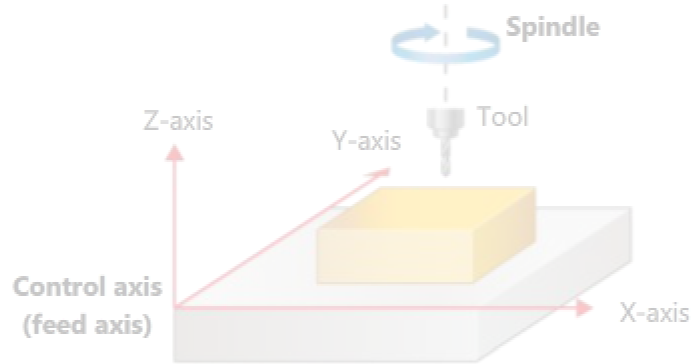
3.6 Coordinate system command

The coordinate system must be defined to give commands regarding the position of the workpiece relative to the movement of the tool.
The basic axes (lathe systems: two axes, machining center systems: three axes) are orthogonal axes defined as the X-, Y-, and Z-axes.

Lathe system



Machining center system

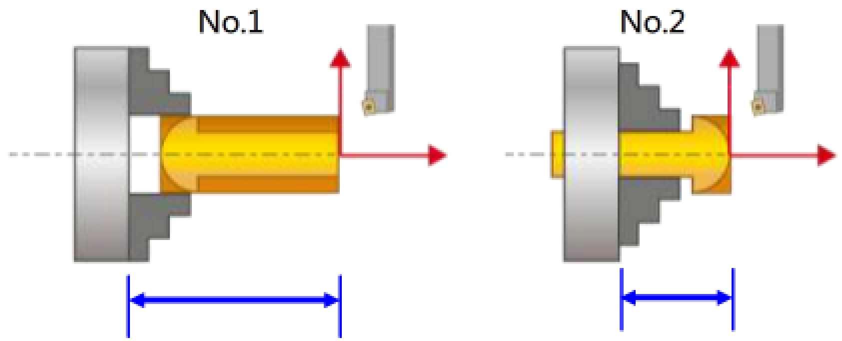


Workpiece coordinate system
This coordinate system designates the positional relationship between the tool and workpiece.
The workpiece coordinate system that applies to the position of the workpiece must be set in advance as a parameter.

G54 (No.1), G55 (No.2) ... G59 (No.6)

Selecting a workpiece coordinate system according to the position of the workpiece enables you to start machining from the appropriate position.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
```



3.7 Axis movement command

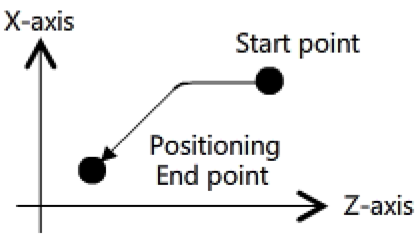
Types of axis movement

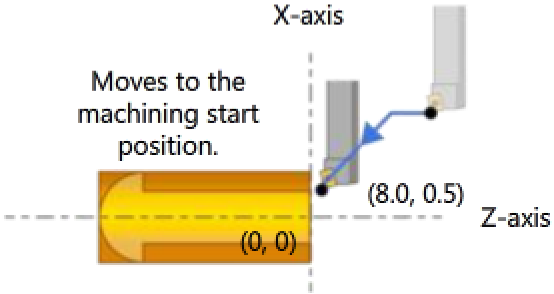
For feed axis movement operations, **cutting** and **rapid traverse** are available depending on whether or not machining is being performed.

Cutting is the operation in which a tool machines the material.
Rapid traverse is the operation that moves a tool without machining the material.

3.7.1 Positioning

Positioning is the command for moving the tool to the machining start position using rapid traverse.

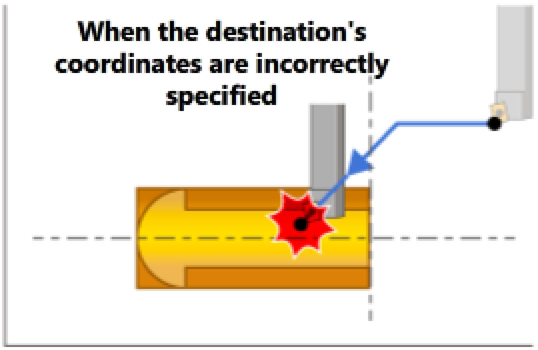




When using rapid traverse, the positioning command must be given. The positioning command code is G00. Specify the coordinates of the destination following the command code.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
```

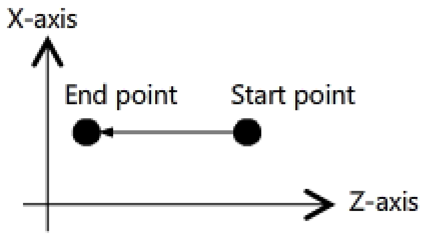
The purpose of positioning is fast movement, but the trajectory is not optimized. Be careful, because inappropriate positioning for machining may lead the tool to hit the workpiece, causing scratches on the material, damage to the tool, or other problems.

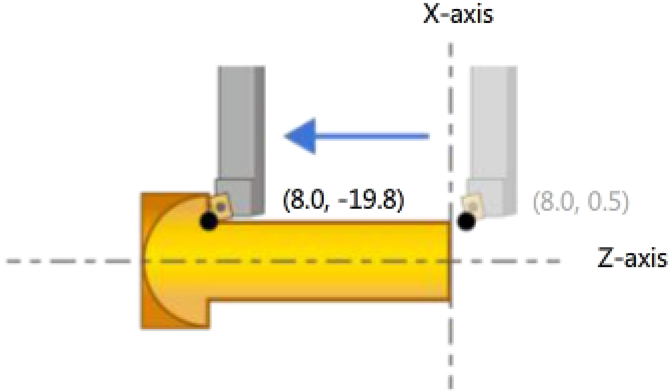


3.7.2 Interpolation (linear)

Interpolation is a movement command for cutting. For example, for movement along both the X and Y axes, the intermediary movement between the points is performed, so it is called interpolation.
The tool moves along the optimal trajectory (calculated route) from the starting point to the end point while machining the material.
There are two broad types, based on the shape of the trajectory while moving.

Linear interpolation
The position and speed of each axis are optimally controlled so that the movement trajectory is a straight line.





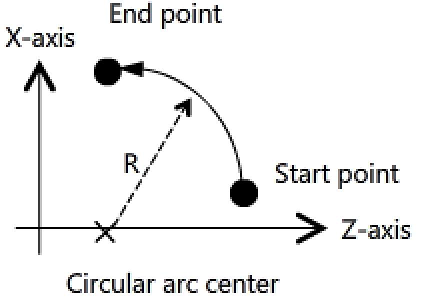
The command code for linear interpolation is G01. Specify the coordinates of the destination following the command code.

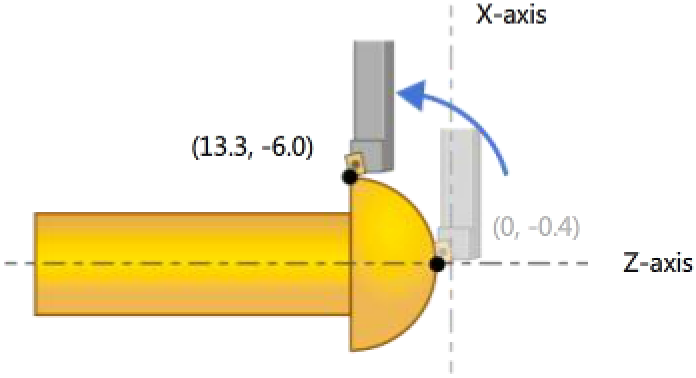
```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
G01 Z-19.8 F100
```

3.7.2 Interpolation (circular)

Circular interpolation

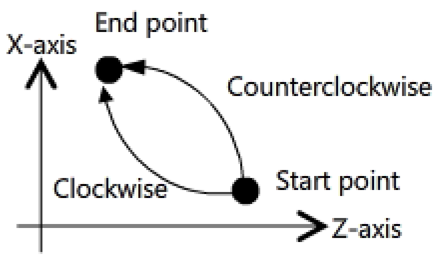
The position and speed of each axis are optimally controlled so that the trajectory forms a circular arc shape around a center point.





The types of circular interpolation command codes are G02 for clockwise and G03 for counterclockwise, based on the direction in which the circle arc is drawn.
Specify the coordinates of the destination and the radius of the circular arc following the command code.

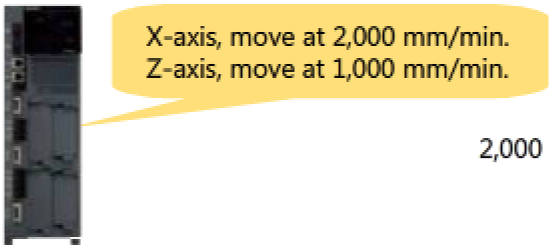
```
...
N0020 G55 G0  X0.0 Z0.0
M03 S1000
G01 Z-0.4 F100
G03 X13.3 Z-6.0 R6.748 F100
```



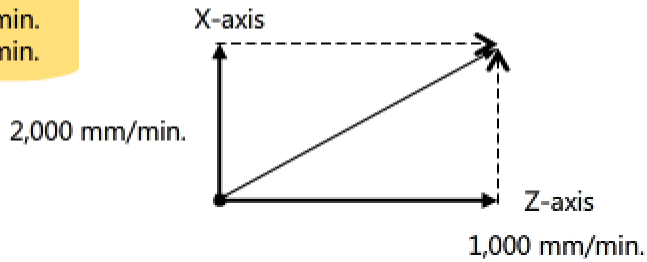
3.7.3 Speed command (rapid traverse)

Feeding can be divided into broad categories, based on the way that the speed command is given to the axes: **rapid traverse** and **cutting feed**.

Rapid traverse
This type of feeding is executed by a positioning command for purposes such as moving the tool to the machining start position.
Normally, speed commands are given separately to each axis.
The combined speed of both axes together is the table movement speed.



Control unit CPU

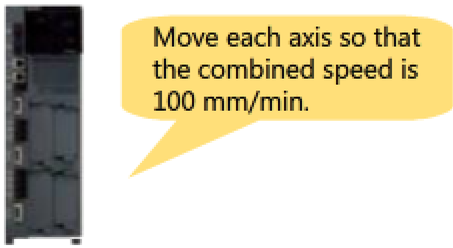


The rapid traverse rate is determined based on the specifications of the NC machine tool.
This means that a speed command cannot be given using the machining program.

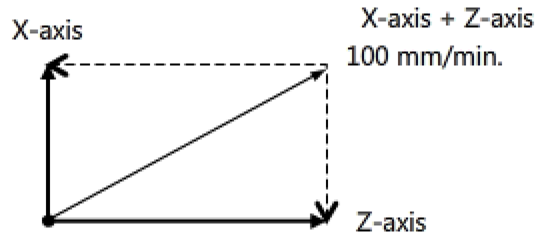
3.7.3 Speed command (cutting feed)

Cutting feed

This type of feeding is executed by an interpolation command for the purpose of machining.
The movement speed across the contact surface of the workpiece and the tool is designated.
The movement speed of each axis is distributed such that the combined speed is equal to the command speed.
The speed of the table is distributed across all axes to become the feed rate of each axis.



Control unit CPU



The cutting speed is designated using the machining program.
The command code is F.

```
O1000(RIVET)
N0010 G28 U0 W0
T0101
M03 S1000
G54 G00 X8.0 Z0.5
G01 Z-19.8 F100
```

3.8 Miscellaneous functions

Aside from the main commands for cutting, miscellaneous functions are available for controlling the spindle, cutting oil, and so on. These commands are designated using M and its following numbers.

O1000(RIVET)	
N0010 G28 U0 W0	Reference point return
T0101	Tool number/tool length correction number commands
M03 S1000	**Turning the spindle clockwise/rotation rate**
G54 G00 X8.0 Z0.5	Positioning
G01 Z-19.8 F100	Linear interpolation
G02 X8.4 Z-20.0 R0.2 F100	Circular interpolation
G01 X14.0 F100	Linear interpolation
M05	**Stopping the spindle**
G30 U0 W0	Reference point return
T0202	Tool replacement
M01	**Pausing the program**
N0020 G55 G0 X0.0 Z0.0	Positioning
M03 S1000	Turning the spindle clockwise/rotation rate
G01 Z-0.4 F100	Linear interpolation
G03 X13.3 Z-6.0 R6.748 F100	Circular interpolation
G00 X14.0	Positioning
M05	**Stopping the spindle**
G28 U0 W0	Reference point return
M30	**End program**

3.9 Summary

The contents of this chapter are:

- Coding scheme
- Basic command codes for machining

Important points to consider:

Reference point return	• The G command code is used to move axes to standard positions.
Tool	• The T code is used for commands regarding tools. • Designates tool selection and correction.
Spindle	• The S code is used for specifying the rotation rate.
Workpiece coordinate system	• The coordinate system designates the positional relationship between the tool and workpiece. • This data must be registered for each workpiece in advance, and the G command code is used when machining.
Axis movement operation	• Cutting is an operation in which a tool machines the material. • Rapid traverse is an operation that simply moves a tool without machining the material.
Positioning	• This is the movement command for rapid traverse, which uses the G code. • The coordinates of the destination are specified following the command code.
Interpolation	• This is the movement command for cutting, which uses the G code. • The coordinates of the destination are specified following the command code. • The types are linear and circular.
Speed	• The rapid traverse rate is determined based on the specifications of the NC machine tool. • The F code is used for the cutting feed rate.
Miscellaneous functions	• The M code is used for miscellaneous commands.

Test Final Test

Now that you have completed all of the lessons of the Mitsubishi Electric CNC Fundamentals (Functions) 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 (12 items) 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
Percentage :	100%

To pass the test, you have to answer **60%** of the questions correct.

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.

Test Final Test 1

Instructions to the NC machine tool

Select the correct description of the instruction methods for an NC machine tool.

- (selected) Instructions are given using parameters that determine the functions and specifications and using a machining program that indicates the machining procedure and machining details.
- All instructions must be given by the machining program.

Answer Back

Test Final Test 2

Minimum command unit

Select the correct description of the minimum command unit.

- ○ The unit can be freely changed when machining.
- ● The unit cannot be changed because it is determined based on the specifications of the NC unit.

Answer | Back

Test Final Test 3

Feed rate

Select the correct description of the feed rate.

- (●) Speed (rate) is affected by the conversion ratio of the mechanism that transfers motor rotation.
- ( ) Speed (rate) is not affected by the transfer mechanism.

Answer Back

Test Final Test 4

Acceleration and deceleration

Select the correct statement regarding acceleration and deceleration when the tool or the table is moved.

- (selected) Changing speed gradually to reach the commanded speed prevents machine malfunctions and damage to the tools.
- It is OK to instantly accelerate/decelerate the tool or table in order to shorten machining time.

Answer Back

Test Final Test 5

Machine error

Select the correct statement regarding machine error for when the tool or the table is moved.

- (selected) Machine error can be corrected by using parameters to account for machine errors in advance.
- The machine must be made precisely because machine error cannot be corrected.

Answer   Back

Test — Final Test 6

Machining program codes

Select the correct statement regarding the notation of a machining program.

- (selected) The command details are described using a combination of letters, such as G, M, and T, and numbers.
- Only letters are used for notation.

Answer | Back

Test Final Test 7

Reference point

Select the correct statement regarding the reference point.

- ○ The reference point must be set in advance for each workpiece.
- ● It is a basic position for establishing the position of the table or the turret.

Answer Back

Test Final Test 8

Tool length correction

Select the correct statement regarding tool length correction.

- ○ The gap between the feed screw and the table is corrected.
- ● The tool length is corrected by specifying the data number of the measurement that was registered for each tool in advance.

Answer Back

Test Final Test 9

Movement commands
Select the statement that correctly describes “axis movement commands”.

[Q1] Rapid traverse

[Q2] Interpolation

Q1 Movement to the machining start position

Q2 Movement for cutting

Answer Back

Test

Final Test 10

Speed command
Select how speed commands are given for each of the following "feeds".

[Q1] Rapid traverse

[Q2] Cutting feed

Q1 Speed commands are given separately to each axis ▼

Q2 A combined speed command is given ▼

Answer Back

Test | Test Score

You have completed the Final Test. You results area as follows.
To end the Final Test, proceed to the next page.

Correct answers:	10
Total questions:	10
Percentage:	100%

Proceed   Review

Congratulations. You passed the test.

You have completed the Computerized Numerical Controllers (CNC) Mitsubishi Electric CNC Fundamentals (Functions) 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.

Review Close