MITSUBISHI ELECTRIC

Changes for the Better

Numerical Control (CNC)

User's Manual

NC Virtual Simulator

IB-1501743(ENG)-F

Introduction

This document provides instructions on how to use NC Virtual Simulator. Read this document before using NC Virtual Simulator so that you know how to properly use it and prevent problems that can result from incorrect use.
NC Virtual Simulator supports the following NC series.
Only machining center type machines are supported.

Supported model	Detail
M800VW Series	M830VW, M850VW
M800VS Series	M830VS, M850VS
M80VW Series	M80VW
M80V Series	M80V TypeA, M80V TypeB
M800W Series	M830W, M850W
M800S Series	M830S, M850S
M80W Series	M80W
M80 Series	M80 TypeA, M80 TypeB

The following short forms are used in this document.

Short form	Supported model
M800V, M800V Series	M800VW Series, M800VS Series
M80V, M80V Series	M80VW Series, M80V Series
M8V, M8V Series	M800VW Series, M800VS Series, M80VW Series, M80V Series
M800, M800 Series	M800W Series, M800S Series
M80, M80 Series	M80W Series, M80 Series
M8, M8 Series	M800W Series, M800S Series, M80W Series, M80 Series

Notes on reading this document

(1) While this document is intended to cover as much special operations as possible, other operations not described in this document must be construed as not executable.
(2) For the specifications of each individual machine tool, consult the documentation issued by the machine tool builder. The restrictions and available features described in the documentation issued by the machine tool builder take precedence over the descriptions in this document.

Notes on using this software

(1) Decimal separator
As the decimal separator, a period (.) is used regardless of the language of the OS on which NC Virtual Simulator is installed.

Related manuals

Manual	IB No.
M800V/M80V Series Instruction Manual (1/2)	IB-1501806
M800V/M80V Series Instruction Manual (2/2)	IB-1501807
M800V/M80V Series Programming Manual (Lathe System) (1/2)	IB-1501619
M800V/M80V Series Programming Manual (Lathe System) (2/2)	IB-1501620
M800V/M80V Series Programming Manual (Machining Center System) (1/2)	IB-1501621
M800V/M80V Series Programming Manual (Machining Center System) (2/2)	IB-1501622
M800V/M80V Series Alarm Manual	IB-1501808
M800V/M80V Series Parameter Manual	IB-1501809
M800/M80/E80 Series Instruction Manual	IB-1501274
M800/M80/E80/C80 Series Programming Manual (Lathe System) (1/2)	IB-1501275
M800/M80/E80/C80 Series Programming Manual (Lathe System) (2/2)	IB-1501276
M800/M80/E80/C80 Series Programming Manual (Machining Center System) (1/2)	IB-1501277
M800/M80/E80/C80 Series Programming Manual (Machining Center System) (2/2)	IB-1501278
M800/M80/E80 Series Alarm/Parameter Manual	IB-1501279

Precautions for safety

(Be sure to read before using this product.)
When using this product, read this manual and the related manuals introduced in this manual thoroughly, and pay full attention to safety to handle this product correctly.
The precautions shown in this manual are for this product only. For the safety precautions of the NC system, refer to the manual of the numerical controller to be used.

This section "Precautions for Safety" ranks the safety precautions into " ⚠ DANGER", " ⚠ WARNING" and " ⚠ CAUTION".

⚠ DANGER **When the user may be subject to imminent fatalities or major injuries if handling is mistaken.**
⚠ WARNING **When the user may be subject to fatalities or major injuries if handling is mistaken.**
⚠ CAUTION **When the user may be subject to injuries or when physical damage may occur if handling is mistaken.**

Note that even items ranked as ⚠ "CAUTION", may lead to major results depending on the situation.
In any case, important information that must always be observed is described.
Keep this manual in a safe place for future reference and be sure to deliver it to the end user.

The following signs indicate prohibition and compulsory.

	This sign indicates prohibited behavior (must not do). **For example, indicates "Keep fire away".**

	This sign indicated a thing that is pompously (must do). **For example, indicates "it must be grounded".**

The meaning of each pictorial sign is as follows.

 CAUTION	 **CAUTION rotated object**	 **CAUTION HOT**	 **Danger Electric shock risk**	 **Danger explosive**
 Prohibited	 **Disassembly is prohibited**	 **KEEP FIRE AWAY**	 **General instruc-tion**	 **Earth ground**

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

■ Matters concerning design

⚠ To perform control (data change, operation status change, etc.) on an device and equipment (numerical controller, PLC, servo, robot, server, etc.) that is in operation from an industrial personal computer equipped with this product, configure an interlock circuit outside the device and equipment so that the entire system always works on the safe side. Read the manual thoroughly and make sure it is safe before proceeding. In particular, the above control for device and equipment from a remote location via a network may not be able to immediately deal with troubles on the device and equipment side due to abnormal data communication.

⚠ Provide a safety circuit outside the industrial personal computer so that the entire system works safely even if the industrial personal computer equipped with the product fails. Accidents may occur due to incorrect output or malfunction.

⚠ CAUTION

■ Matters concerning design

⚠ While various settings are reflected, do not perform the operation that forces the power of the industrial personal computer equipped with this product to be turned OFF. If you perform an operation such that the industrial personal computer equipped with this product is forcibly turned OFF during the reflection, the data becomes unstable and it needs to be reconfigured and re-reflected. It may also cause the product to malfunction.

⚠ To protect availability, integrity and confidentiality of the system against cyber-attacks including unauthorized access, denial-of service (DoS) (*1) attack, and computer virus from external sources via a network, take security measures such as firewall, VPN, and anti-virus software.
(*1) Denial-of service (DoS) refers to a type of cyber-attack that disrupts services by overloading the system or by exploiting a vulnerability of the system.

⚠ Mitsubishi Electric assumes no responsibility for any problems caused to the system by any type of cyber-attacks including DoS attack, unauthorized access and computer virus.

■ Matters concerning operation

⚠ The simulation result does not guarantee the machining accuracy of the actual NC.

Application of this software

Users must agree the following conditions for an unexpected software problem:
- Use the software in a way that the problem will not cause a serious accident.
- Functions for data backup and fail-safe need to be systematically implemented outside the device as preventive measures for the problem.

Trademarks

MELSOFT is either a trademark or a registered trademark of Mitsubishi Electric Corporation in Japan and/or other countries.
Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.
Microsoft® and Windows® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
VMware is either a trademark or a registered trademark of VMware, Inc. in the United States and/or other countries.

Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.
In some cases, trademark symbols such as "™" or "®" are not specified in this manual.

Contents

1. Overview 1

2. Installation and setup 3

2.1 System requirements 3
2.2 Installation 3
2.3 Upgrade 8
2.4 Uninstallation 9

3. License 11

3.1 Types of licenses 11
3.2 License registration 12
3.3 Removing expired license 15

4. Configuration of application screen 16

4.1 Title bar 17
4.1.1 Settings 18
4.1.1.1 Switching display language 19
4.1.1.2 License expiration notification 20
4.1.2 Help 21
4.1.2.1 Accessing Mitsubishi Electric FA Global Website 22
4.1.2.2 Version information 23
4.1.2.3 License information 24
4.2 Function menu 25
4.3 Sub-display area 27
4.3.1 Showing and hiding sub-display area 27
4.3.2 Sub-display switch buttons 28
4.3.2.1 Processing status display 29
4.3.2.2 Machining program display 31
4.3.2.3 NC data list 32
4.3.2.4 Machining simulation settings 39
4.3.2.5 Analysis simulation settings 47
4.3.2.6 Simulation history 53
4.3.2.7 Machining time prediction setting 55
4.4 Display area 56
4.4.1 Resizing display area 56
4.5 Shortcut keys 57

5. Using NC Virtual Simulator 58

5.1 Starting NC Virtual Simulator 58
5.2 Exiting NC Virtual Simulator 59
5.3 Creating a project 60
5.3.1 Creating a new project 60
5.3.1.1 Creating a project using the machine tool's backup data 60
5.3.1.2 Creating a project using the network-connected machine tool's data 64
5.3.1.3 Creating a project using the sample project 68
5.3.2 Opening a project 70
5.3.3 Saving a project 71
5.3.4 Closing a project 71
5.3.5 Renaming a project 72
5.3.6 Deleting a project 74
5.4 Importing machine model to NC Virtual Simulator 76
5.5 NC Trainer2 plus link 77

6. Machining simulation 78

6.1 Machining simulation window 79
6.1.1 Screen configuration and display items 79

6.1.2 Machining simulation commands 80
6.2 Preparation 83
6.2.1 Registering tool data 83
6.2.2 Setting workpiece data 83
6.3 Operation 90
6.3.1 Set NC data (coordinate system, parameters, etc.) 90
6.3.2 Register jig model 91
6.3.3 Check interference 94
6.3.4 Change the simulation priority 96
6.3.5 Select a program 97
6.3.6 Start simulation 97
6.3.7 Stop simulation 98
6.3.8 Change the point of view on workpiece 99
6.3.9 Adjust the workpiece display position, angle and size 101
6.3.10 Partially enlarge workpiece shape 104
6.3.11 Switch between machine simulation and workpiece simulation 107
6.3.12 Superimpose product CAD model on workpiece shape 108
6.3.13 Adjust display position of product CAD model 110
6.3.14 Measure distance on workpiece shape 112
6.3.15 Display cross section of workpiece shape 116
6.3.16 Display machining shape side by side 120
6.3.17 Save/restore the point of view 123
6.3.18 Show or hide scale 124
6.3.19 Read machining time prediction setting from another project 125
6.3.20 Specifying machining programs to be skipped 126
6.3.21 Exporting/importing a workpiece shape 127
6.4 Caution 128

7. Analysis simulation 131

7.1 Analysis simulation window 132
7.1.1 Screen configuration and display items 132
7.1.2 Analysis simulation commands 133
7.2 Preparation 135
7.3 Operation 136
7.3.1 Enable program link 138
7.3.2 Display machining shape side by side 141
7.3.3 Turn to line segment path (position) 144
7.3.4 Turn to color map display 145
7.3.5 Display CAD model error 150
7.3.6 Turn to plot point highlighting (axis reversal) 155
7.3.7 Selecting a tool for drawing 156
7.4 Caution 157

8. Tool settings 158

8.1 Screen configuration and display items 158
8.2 Tool settings commands 163
8.3 Operation 164
8.3.1 Set tool data 164
8.3.2 Search tool data 167
8.3.3 Sort tool numbers 168
8.3.4 Delete, insert or move lines of tool data 169
8.3.5 Copy and paste one line of tool data 170
8.3.6 Read NC tool management data 171
8.3.7 Import tool data of another project 172
8.4 Caution 173

9. Simulation difference analysis 174

9.1 Simulation difference analysis window 175
9.1.1 Screen configuration and display items 175

9.1.2 Simulation difference analysis commands ... 176
9.2 Operation ... 178
9.2.1 Compare the simulation results ... 178
9.2.2 Switch the analysis type ... 181
9.2.3 Switch to simulation result of a different machining program ... 182
9.2.4 Retain simulation history ... 183
9.2.5 Apply machining conditions of simulation history to current project ... 184
9.2.6 Delete simulation history ... 185
9.2.7 Enable program link ... 186
9.3 Caution ... 188

10. Messages ... 190

10.1 Dialog messages ... 190
10.2 Simulation messages ... 196
10.3 NC alarm messages ... 197

11. Caution ... 198

11.1 General ... 198
11.2 Caution related to machining programs ... 199
11.3 Caution related to re-display of machining simulation results ... 200
11.4 Caution related to NC Trainer2 plus link ... 201
11.5 Caution related to tool display ... 202

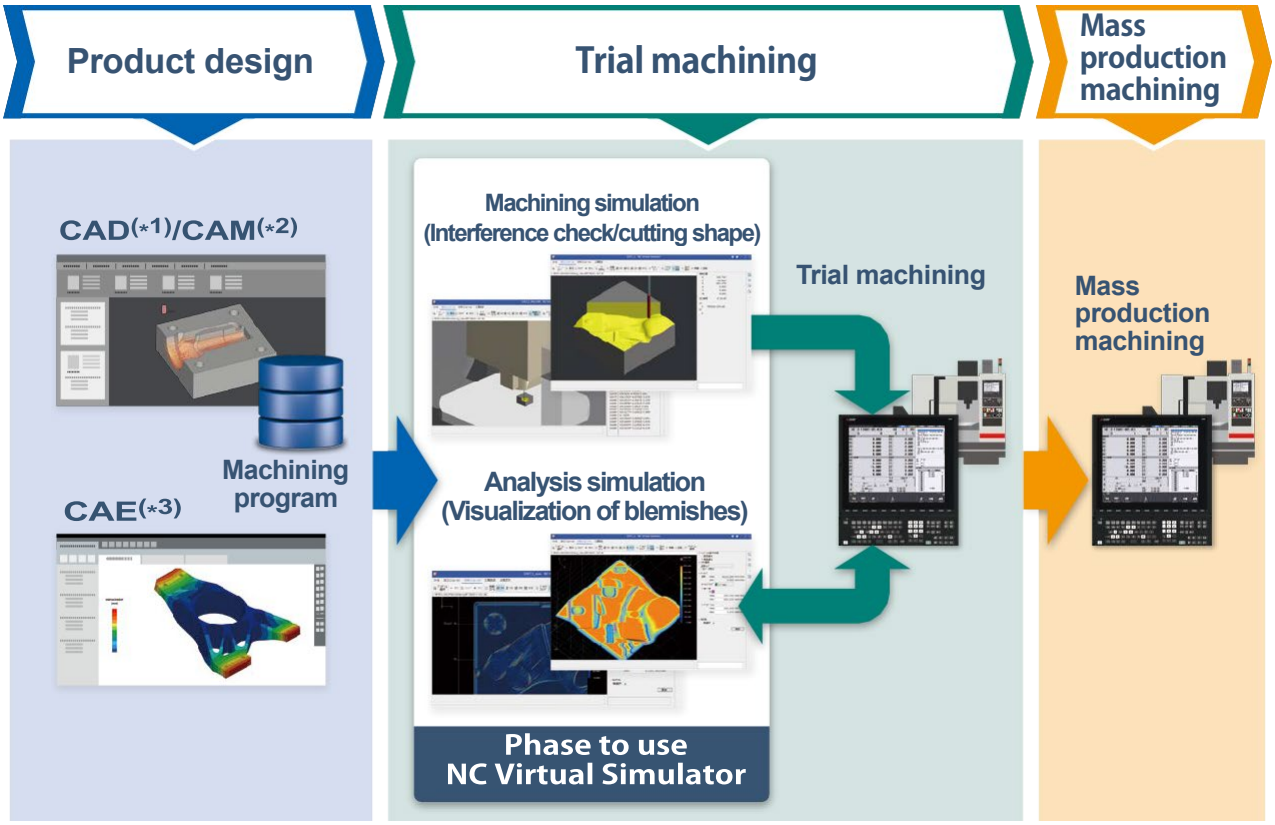
12. Troubleshooting ... 206

12.1 Machine model is not displayed ... 206
12.2 Simulation takes time depending on PC ... 207
12.3 Machining condition selection in [NC Data List] cannot be used ... 209
12.4 Macro alarm message appears during simulation and simulation stops ... 209
12.5 Application does not work correctly when Remote Desktop Connection is used ... 209
12.6 Saving or opening a project takes time depending on the project ... 210
12.7 Application does not work correctly when a product model is selected or displayed by a curved surface in analysis simulation ... 210
12.8 "There is a problem with the registered license file." is displayed when NC Virtual Simulator is started ... 211
12.9 NC Trainer2 plus link may fail when the update program is applied after upgrading Windows 11 to version 23H2 or later ... 215

13. Applicable NC version ... 218

1. Overview

NC Virtual Simulator enables simulation from various angles digitally (on a PC) using the same machining conditions as used by the actual machine tool.
It simulates the motion of the CNC realistically, including smoothing, acceleration/deceleration and delay in servo response. It provides highly-accurate simulation because the same machining conditions as used by an actual machine tool can be used. It helps you to detect the problems before actual machining including machining errors, faulty setup, and machine interference, which you were previously not able to identify until trial machining. This increases your work efficiency and productivity.



(*1) CAD: Computer Aided Design
(*2) CAM: Computer Aided Manufacturing
(*3) CAE: Computer Aided Engineering

NC Virtual Simulator offers the following two main simulation functions.

Machining simulation	Realistic simulation of the motion of the CNC using the same machining conditions as used by the actual machine tool. You can check a highly-accurate graphic representation of the workpiece contour, the calculated machining time, and machine interference. It allows you to detect machining errors and faulty setup before actual machining, which was not possible with the simulation using CAD/CAM tools.
Analysis simulation	Simulation that allows visualization of positions and change in feedrate and acceleration, etc. at a micro segment level using color maps or other means. It makes it easy to recognize and analyze what caused slight surface blemishes.

(*) Only CNC for machining center with a single part system and eight or less NC axes is supported.

NC Virtual Simulator has the following functions.

	Function	Description	Reference
(1)	Workpiece simulation	Digital position data where smoothing, acceleration/deceleration and delay in servo response are reproduced is used to estimate highly-accurate machining times and express highly-accurate surface shapes.	6.3.11 Switch between machine simulation and workpiece simulation
(2)	Superimposition display of product CAD model	A product CAD model is read and superimposed on the cutting simulation. This can be used to check whether a machining process is missed, and so on.	6.3.12 Superimpose product CAD model on workpiece shape
(3)	Pinpoint zoom display	This is the function to cut out the analysis range and partially display it with a higher accuracy. This is used when drawing accuracy is too rough to see the details of the machining surface.	6.3.10 Partially enlarge workpiece shape
(4)	Machine simulation	You can check machine interference in automatic operation using a 3D machine model. When there is interference, the contacting parts are indicated in the interference color, and a warning appears.	6.3.11 Switch between machine simulation and workpiece simulation 6.3.3 Check interference
(5)	Smart measurement	You can measure distance and angle between two points selected on a workpiece shape or a machine model. (Optional function)	6.3.14 Measure distance on workpiece shape
(6)	Jig model display	By reading a jig model and using it with a 3D machine model, you can check interference in a state closer to the actual machining.	6.3.2 Register jig model
(7)	Cross-sectional view	You can display the cross section of the workpiece shape and check the cut state, etc. of the machining surface in details.	6.3.15 Display cross section of workpiece shape
(8)	Color map display	This is the function to display variation of each data, such as speed difference of feedrate/acceleration rate/adjacent line segment, in a color map. This can be used to identify where a machining fault occurs.	7.3.4 Turn to color map display
(9)	Program link	You can identify the machining program block corresponding to the line segment in the simulated shape, and easily check and modify the machining program corresponding to the line segment with a problem found in the color map display.	7.3.1 Enable program link
(10)	Simulation comparison	You can make a comparison with past simulation results since the simulation history is retained. You can see how the machining results differ depending on machining conditions such as NC parameters.	9 Simulation difference analysis
(11)	NC Trainer2 plus link	By linking with NC Trainer2 plus, you can check operations using machine simulation of NC Virtual Simulator as an environment for the machine tool builder to simulate ladder circuit design and machine design.	5.5 NC Trainer2 plus link

2. Installation and setup

2.1 System requirements

NC Virtual Simulator runs in the following environment.

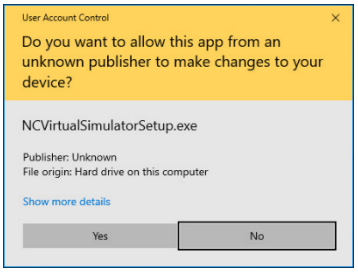
Item	Description
Computer	A PC that has a processor with clock speed of 2.66 GHz or faster and at least 2 cores and 4 threads (4 cores and 8 threads or more is recommended)
Memory	At least 8 GB
Hard disk space	At least 4 GB (excluding the space required for OS operation)
OS	Windows 10 or later (64-bit) (*) WOW64 is used
Interface	10/100/1000M Ethernet (only for network-connected version) USB1.1 or later
Display	A video adapter and monitor with a resolution of HDTV (720p) (1280 x 720) or higher (at least 65,536 simultaneous colors; 16,770,000 simultaneous colors is recommended) PC with OpenGL 4.5 or later display adapter
Supported language	Japanese, English, Simplified Chinese, Traditional Chinese (*1)

(*1) To use a font corresponding to Chinese GB 18030:2022 standard, an update to the following Windows version is required.

Windows10	21H2: Build 19044.3208 22H2: Build 19045.3208 or higher
Windows11	21H2: Build 22000.2176 22H2: Build 22621.1992 or higher

2.2 Installation

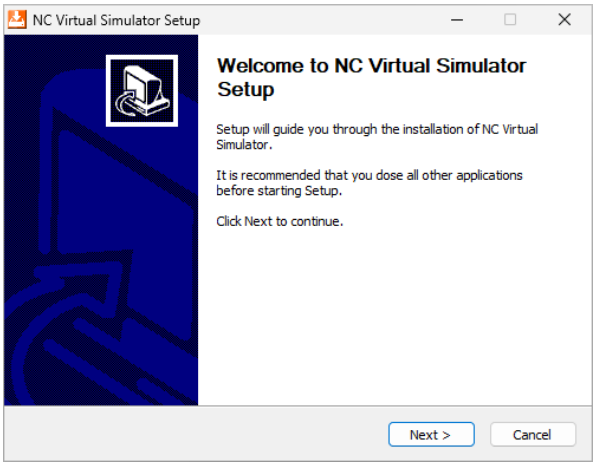
(1) Insert the NC Virtual Simulator installation DVD into the DVD-ROM drive of the PC.
(2) Run NCVirtualSimulatorSetup.exe in the installation DVD.
When you perform installation, you must be logged in as an Administrator.
When User Account Control of Windows is enabled, you see the following confirmation dialog. Select [Yes] to start installation.



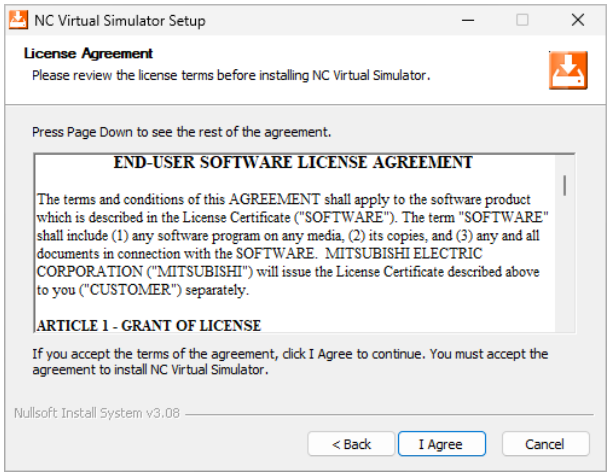
(*) An older version than the installed one cannot be installed.
(*) You cannot perform installation in an OS environment of Windows 8.1 or earlier.

(3) The splash screen appears, and the installer is launched.

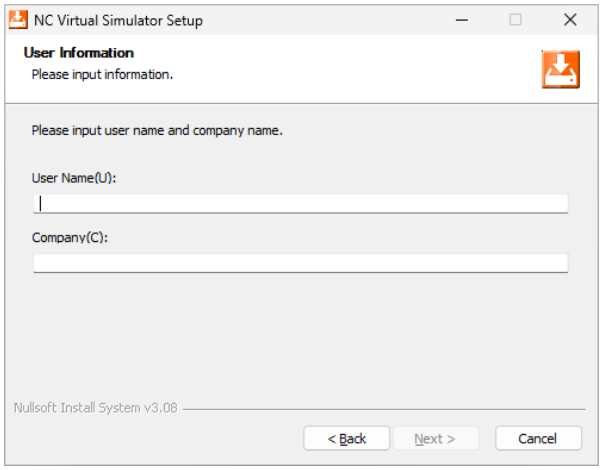
(4) The setup panel appears.
Press [Next].



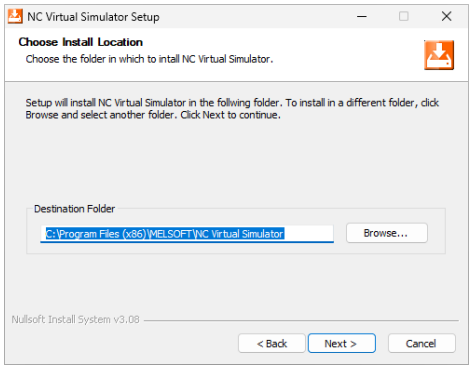
(5) The End-User Software License Agreement appears.
Read the agreement carefully, and press [I Agree].
If you select [Cancel] (do not accept the agreement), the installation of NC Virtual Simulator is canceled.



(6) The Customer Information panel appears. Enter a username and company name, and press [Next].



(7) The panel to choose the destination location appears. If changing the location in which you want to install the application, press [Browse] and choose a location. When you are done, press [Next].



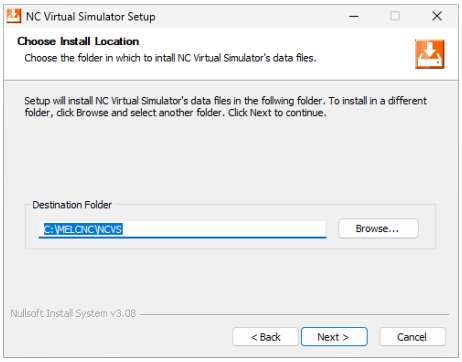
(*) When changing the folder in which to install the application, ensure that the full path does not exceed the maximum length of 80 characters. If it exceeds 80 characters, the folder cannot be set.

(8) The Create a folder to store data file pane appears.

When changing the location, press [Browse] to choose a folder.

Folders that you can specify are restricted. Refer to (*) below.

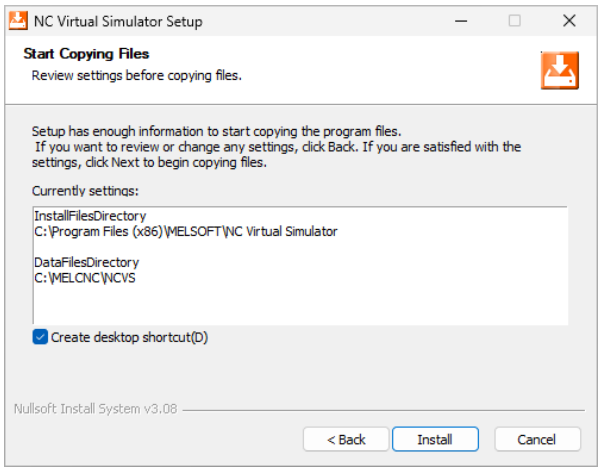
When you are done, press [Next].



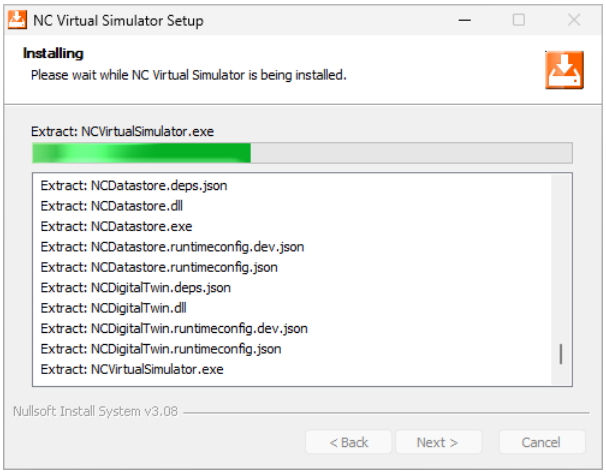
(*) Observe the following restrictions when you change the folder to store data files.

- A folder name cannot contain half-width symbols that are unrecognizable to Windows and two-byte codes such as full-width characters.
- When setting the folder name, ensure that the full path does not exceed the maximum length of 19 characters. If it exceeds 19 characters, the folder name cannot be set.
- Do not specify a folder under "C:\Program Files", "C:\Program Files (x86) ", or "C:\Windows". (These folders are usually write-protected.)
- Do not specify a drive other than C and D. If you do, you may not be able to work with the files in the HD or the memory card device.

(9) The information about the locations is displayed. After verifying it, press [Install]. (If you need to make changes, press [Back].)
Select the "Create desktop shortcut(D)" option if you want to create a shortcut for NC Virtual Simulator on your desktop after installation.



(10) The installation starts. When installation is completed, press [Next].



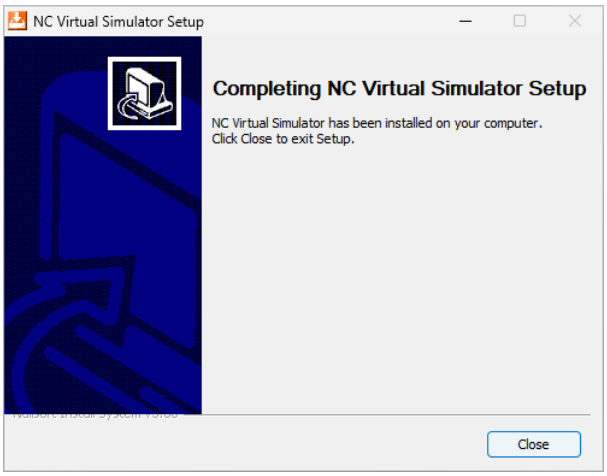
(*) Installation cannot be canceled after being started.

(*) Microsoft .NET Desktop Runtime 5.0.9 (x86) (hereinafter called "Desktop Runtime for Windows") is installed with NC Virtual Simulator. Therefore, it might take time to install them for the first time.

(*) If Desktop Runtime for Windows fails to be installed, NC Virtual Simulator is not installed. Obtain the installer of Desktop Runtime for Windows from the Microsoft official website. After Desktop Runtime for Windows is installed manually, install NC Virtual Simulator from the beginning.

(*) Do not launch NC Virtual Simulator or link NC Virtual Simulator with NC Trainer2 plus until the installation is completed.

(11) When installation completes successfully, the completion panel appears.
Press [Close] to exit the wizard.



(12) Then, register a license.
See "3 License".

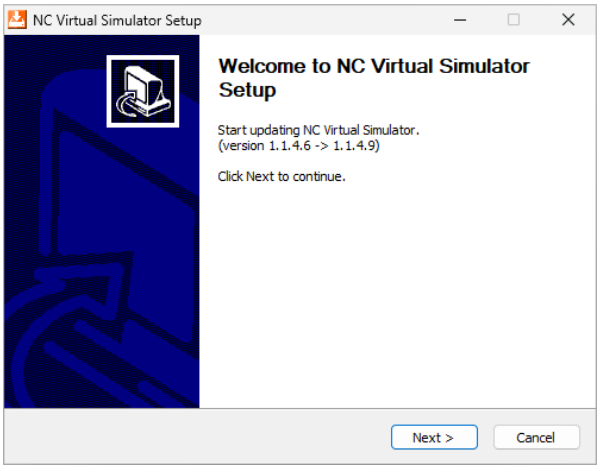
2.3 Upgrade

When upgrading NC Virtual Simulator, take note of the following.

- When NC Virtual Simulator is running, exit it.
- When NC Virtual Simulator is linked with NC Trainer2 plus, close NC Trainer2 plus also.
- Do not launch NC Virtual Simulator or link it with NC Trainer2 plus during the upgrade.

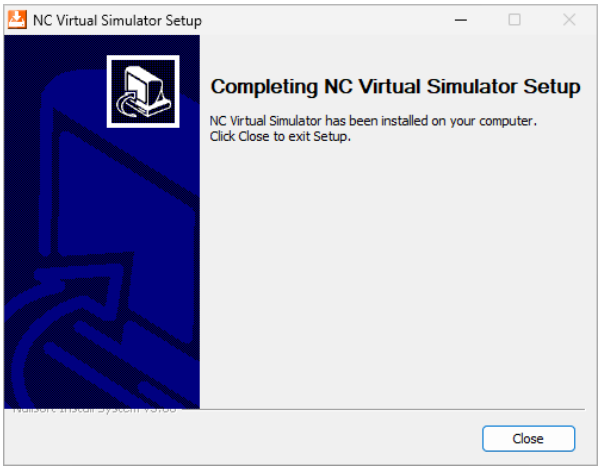
(Note) After the upgrade, take note of the following.
To use a project created with NC Virtual Simulator version A4 or earlier in a version A5 or later, launch the simulation again. If it is used without launching the simulation, operations such as program link may not function correctly.

(1) Run "NCVirtualSimulatorSetup.exe" for a new version of NC Virtual Simulator.
When you perform upgrade installation, you must be logged in as an Administrator.
Press [Next].



(*) Once started, update installation cannot be canceled.

(2) After installation is completed, press [Next] to open the completion panel.
Press [Close] to exit the wizard.

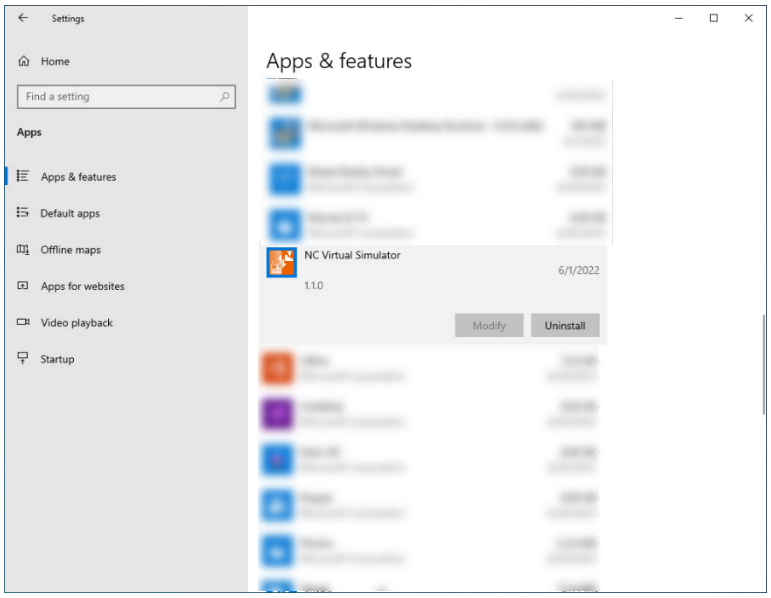


2.4 Uninstallation

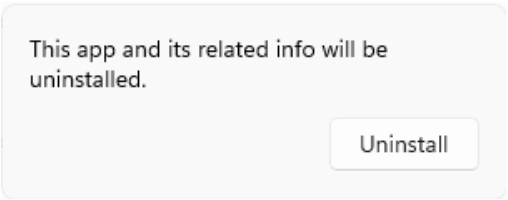
When uninstalling NC Virtual Simulator, take note of the following.

- When NC Virtual Simulator is running, exit it.
- When NC Virtual Simulator is linked with NC Trainer2 plus, close NC Trainer2 plus also.
- Do not launch NC Virtual Simulator or link it with NC Trainer2 plus during uninstallation.

(1) Select [Start] > [Settings] > [Apps] > [Apps & features].
Select "NC Virtual Simulator" from the list, and click [Uninstall].



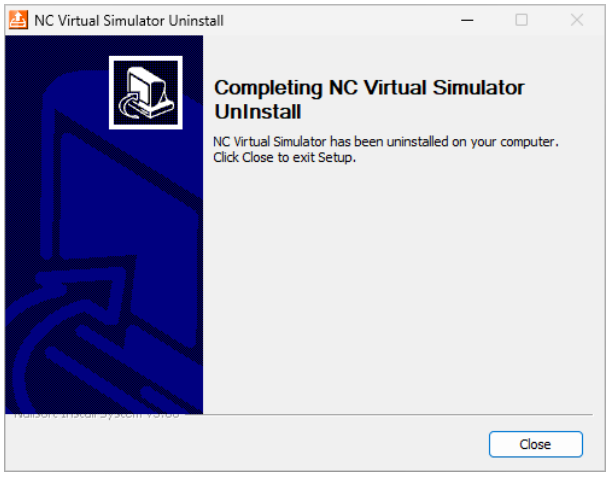
(2) Information about the uninstall destination is displayed.
Press [Uninstall] to start uninstallation.
(Press [Cancel] to go back to [Apps & features].)



(*) Once started, uninstallation cannot be canceled.

(*) "Microsoft.NET Desktop Runtime 5.0.9 (Desktop Runtime for Windows)" is not deleted when NC Virtual Simulator is uninstalled because it may be used by other applications.

(3) After uninstallation is completed, press [Next] to open the completion panel. Press [Close] to exit the wizard.



3. License

To use NC Virtual Simulator, you need to purchase a license for NC Virtual Simulator per PC.
The license is only valid for use on the PC of which a hardware code was obtained when application was submitted.

3.1 Types of licenses

■ License expiration date

NC Virtual Simulator verifies the license on a subscription basis.
The following kinds of licenses are available.

License	Description
Without expiration date	The license can be used indefinitely.
With expiration date (*1)	The license can be used until the date specified at the time of application.

(*1) The license with expiration date expires the day after the expiration date (at 00:00:00).

■ Feature license

The license is issued for each feature. The following kinds of features are available.

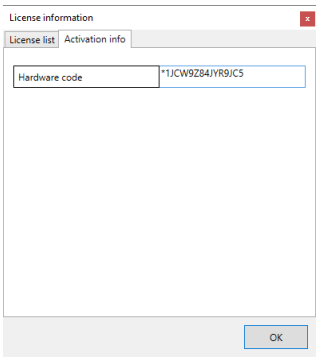
Feature	Description
NC Virtual Simulator	Basic functionality of NC Virtual Simulator. Without this license, you cannot use NC Virtual Simulator.
Smart measure OPTION	The functionality to measure a 3D space distance of the machining shape, product model, etc. in 3D machining simulation.

3.2 License registration

License registration is required to use NC Virtual Simulator. When you start the tool, the "License information" dialog appears. Follow the following steps to apply for and register the license. The same steps are needed to be taken when the license is expired.

(1) Obtain hardware code

Ensure that only the Ethernet port that is used during the operation of NC Virtual Simulator is enabled, and start NC Virtual Simulator. Write down the displayed hardware code, which will be used in step (2).



(*) The hardware code is a code created based on the hardware-specific information.

(*) To verify the hardware code after the license for NC Virtual Simulator is registered, open the license information dialog box by selecting the [Help] button > [License information] in the title bar.

(2) Apply for license

Access the Web form for license issuing, and enter the following information to apply.

- Entitlement ID
- Hardware code
- User information (company name, user name, telephone number, address, email address, etc.)

For the Web form for license issuing, refer to the "NC Virtual Simulator License Application Instructions" included with the installation DVD.

(3) Save license file

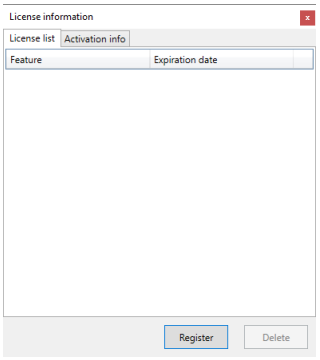
Save the license file sent to the email address in any folder in the PC where NC Virtual Simulator is installed.

(*) Do not change the extension (.elic) of the license file. If you change the extension, the license file will not be displayed in the browse dialog box when registering the license.

(*) The name of the license file can be changed.

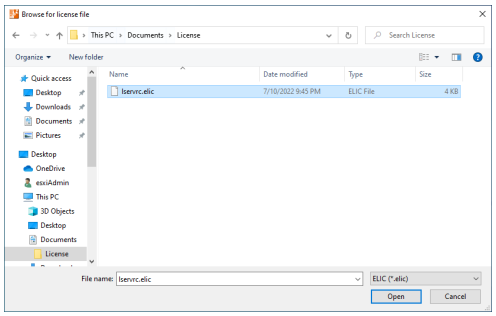
(4) Start application

When NC Virtual Simulator is started, the license information dialog box appears. Click the [Register] button in the [License list] tab.

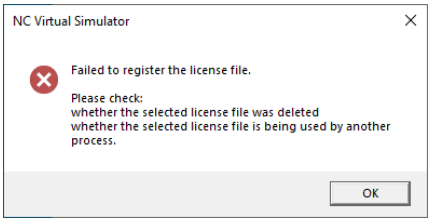


(5) Register license

Select the license file saved in step (3) and click the [Open] button.

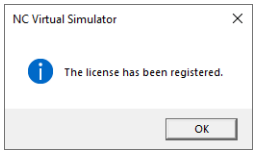


(*) If the registration of the selected license file failed, the following error dialog appears.



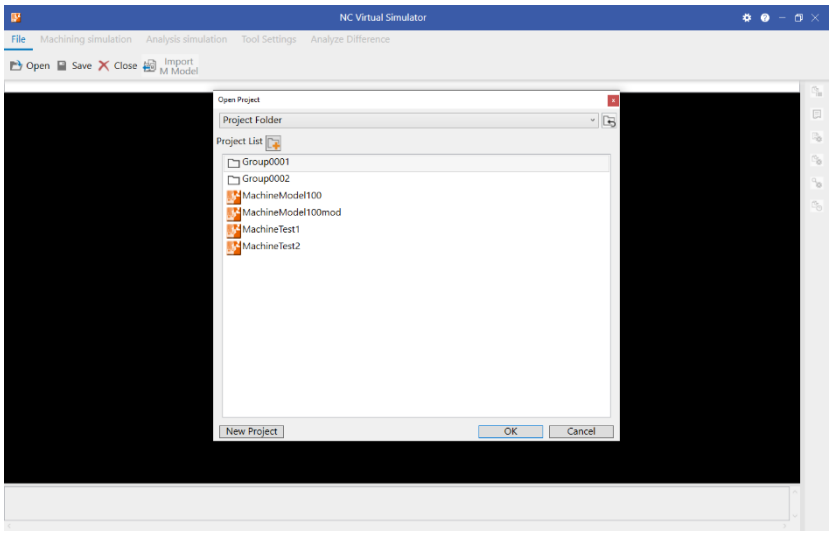
(6) License registration completes

When license registration completes, the following register completion dialog appears. Click the [OK] button.



(7) Restart application

When you restart NC Virtual Simulator after the license is registered, the Open Project dialog box appears.



3.3 Removing expired license

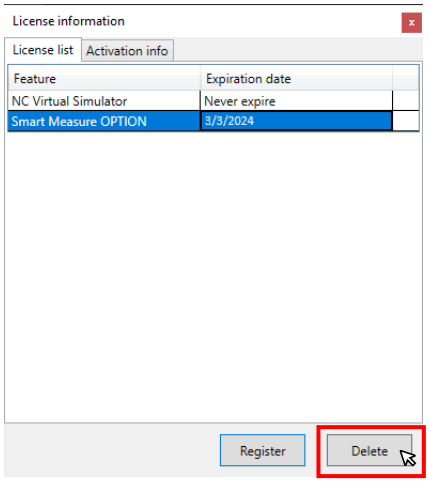
If you start NC Virtual Simulator when an expired OPTION feature is enabled, you see a message box that notifies you of the expired OPTION feature. To disable an expired OPTION feature, remove the OPTION feature by completing the following steps.

(1) Verify license list

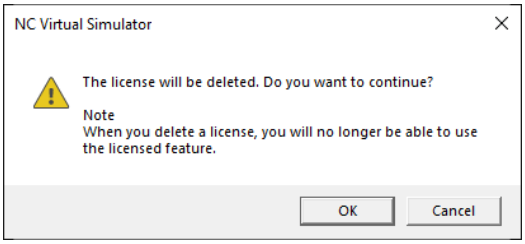
Select the [Help] button > [License information] in the title bar to open the license information dialog box.

(2) Select license to remove

Select the license you want to remove, and click the [Delete] button.



(*) If removing a license that has not expired, the following error dialog appears.

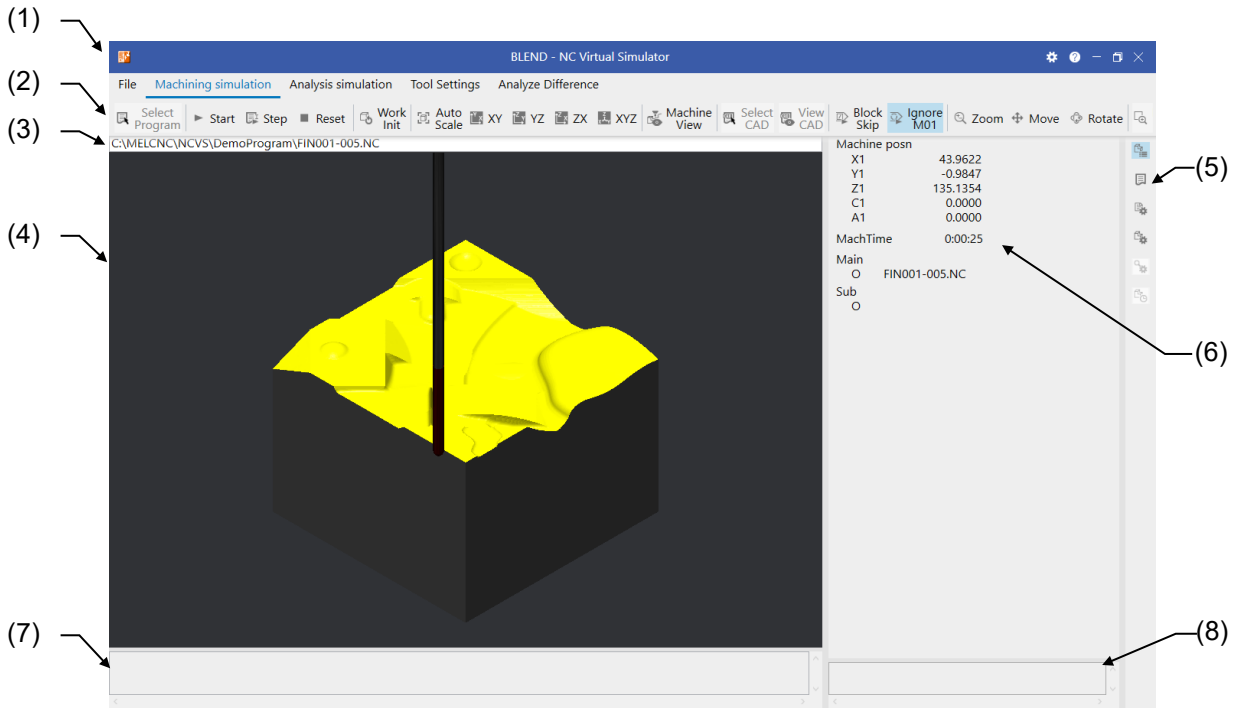


(3) License removal completes

When the license has been removed, the completion dialog appears. Click the [OK] button.

4. Configuration of application screen

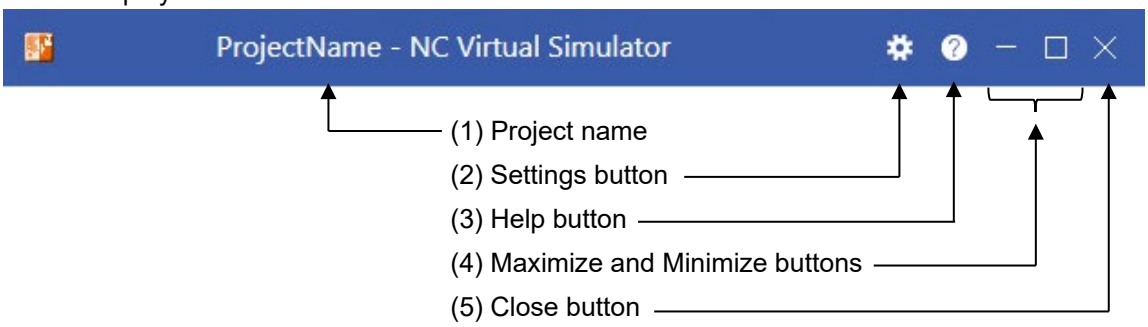
The configuration of the application screen is as follows.



Item	Description
(1) Title bar	Displays the project name, the application name, the Settings button, the Help button, the Maximize and Minimize buttons, and the Close button.
(2) Function menu	The tabs group the icons together based on the function for which the operations of the icons are used. With the function menu, you can work with projects or execute machining simulation and other functions.
(3) Program path display area	Displays the file path of the machining program specified with [Select Program] for machining simulation and analysis simulation.
(4) Simulation display area	Displays the result of machining simulation or analysis simulation.
(5) Sub-display switch buttons	Displays the buttons for switching the sub-display area.
(6) Sub-display area	Allows you to view and set the data used in machining simulation and analysis simulation.
(7) Message display area	Displays the error messages that occurred during machining simulation and analysis simulation. The error messages for the alarms that occurred in simulation and on the NC are displayed. See "10 Messages" for the message contents.
(8) Sub-message display area	Displays the error messages that occurred in the sub-display area.

4.1 Title bar

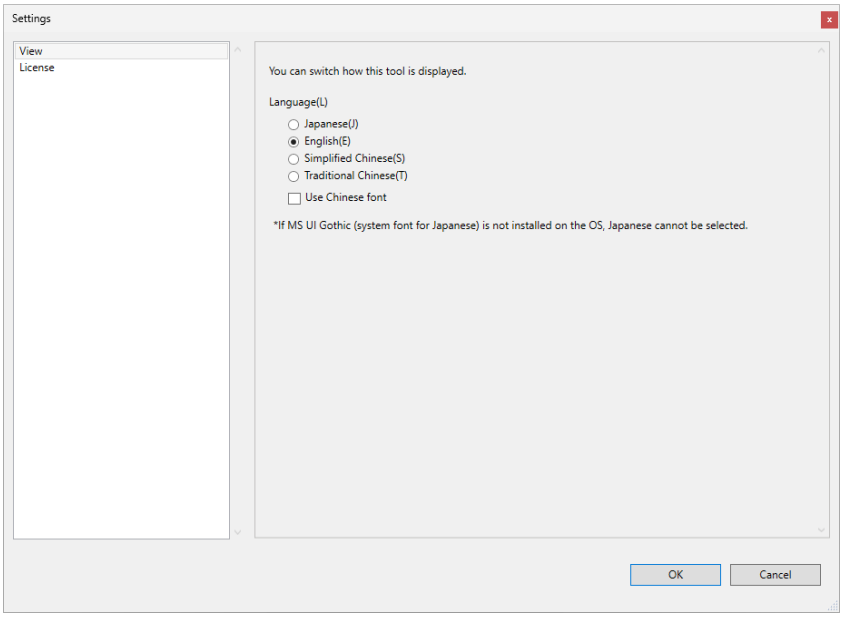
The project name, the Settings button, the Help button, the Maximize and Minimize buttons, and the Close button are displayed.



Item	Description
(1) Project name	Displays the name of the currently open project.
(2) Settings button	Allows you to manage the settings such as the display language.
(3) Help button	Allows you to display the Web site and the version of NC Virtual Simulator, view and register your license.
(4) Maximize and Minimize buttons	Allows you to maximize and minimize NC Virtual Simulator.
(5) Close button	Closes NC Virtual Simulator. For detailed information, see "5.2 Exiting NC Virtual Simulator".

4.1.1 Settings

With the Settings, you can switch the display language and set the notification period for license expiration.

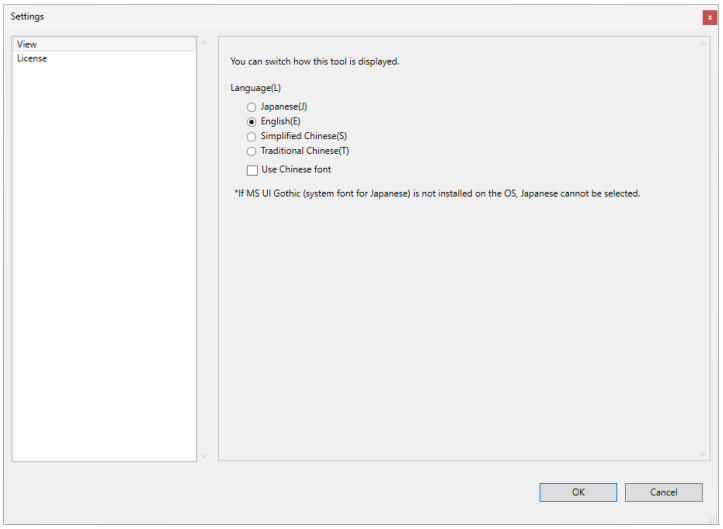


Item		Description
(1)	View	
	Language(_L)	Switch the display language of NC Virtual Simulator. You can choose English, Japanese, Simplified Chinese or Traditional Chinese as the display language. For detailed information, see "4.1.1.1 Switching display language".
	Use Chinese font	Switch the font of NC Virtual Simulator to GB18030:2022 standard (font conforming to GB standard, the China national standard). When "Simplified Chinese" is selected in "Language", this is automatically checked and is kept selected (cannot be clicked). When any other language than "Simplified Chinese" is selected (including "Traditional Chinese"), it can be checked optionally.
(2)	License	
	Notification period for license expiration	Set the notification period for license expiration. You can specify between 7 and 365 days for notification period. For detailed information, see "4.1.1.2 License expiration notification".

4.1.1.1 Switching display language

The following describes how to switch the display language.

(1) Click the [Settings] button in the title bar. In the Settings dialog displayed, select your preferred language in [View] > [Language].



(*) This setting switches the language for the menu bar, message boxes, dialog boxes, etc.

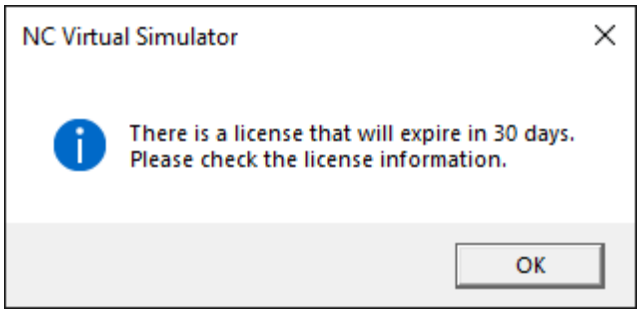
(*) When NC Virtual Simulator is started for the first time after it is installed, the display language is dependent on Windows display language.

Windows display language	NC Virtual Simulator display language
Japanese	Japanese
Chinese (Simplified, China)	Simplified Chinese
Chinese (Traditional, Taiwan)	Traditional Chinese
Other than the above	English

When "Simplified Chinese" is selected, "Use Chinese font" is automatically checked. When "Traditional Chinese" is selected, the check of "Use Chinese font" is not changed.
Windows display language can be set from [Start] > [Settings] (gear icon) > [Time & language] > [Language].

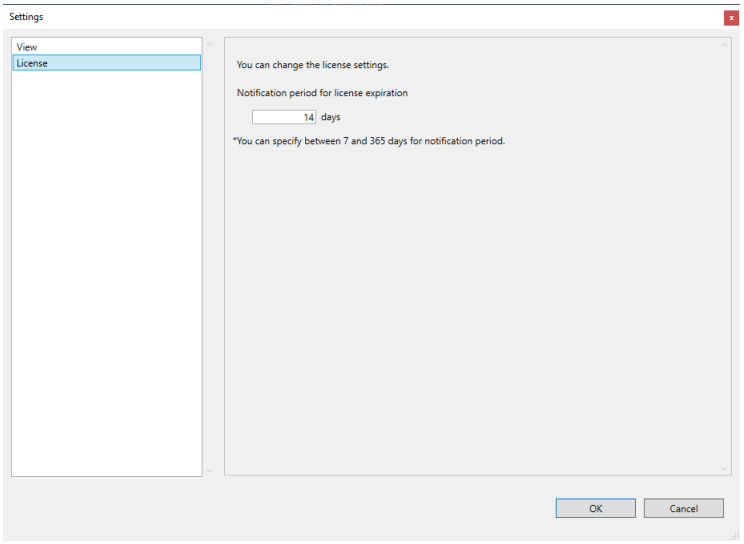
4.1.1.2 License expiration notification

If the number of remaining days before the license expiration date is less than the number of days you set, you are notified with a message box when NC Virtual Simulator is started.
When NC Virtual Simulator is installed, the notification period is set to 14 days.



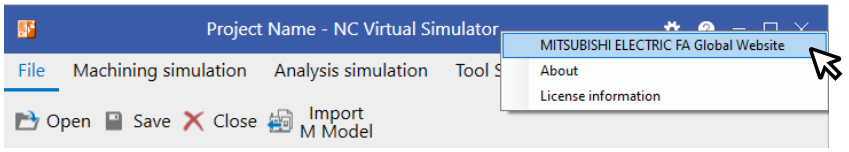
Set the notification period for license expiration with the following steps.

(1) Click the [Settings] button in the title bar. In the Settings dialog displayed, set your preferred period in [License] - [Notification period for license expiration].
You can specify between 7 and 365 days for notification period.



4.1.2 Help

With Help, you can access Mitsubishi Electric FA Global Website and check the version and license information.

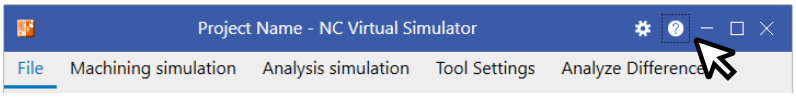


Menu	Description
(1) MITSUBISHI ELECTRIC FA Global Website	Select this menu to display Mitsubishi Electric FA Global Website. For detailed information, see "4.1.2.1 Accessing Mitsubishi Electric FA Global Website".
(2) About	Select this menu to open the version information dialog box. For detailed information, see "4.1.2.2 Version information".
(3) License information	Select this menu to open the license information dialog box. For detailed information, see "4.1.2.3 License information".

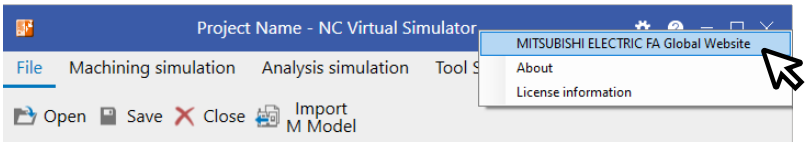
4.1.2.1 Accessing Mitsubishi Electric FA Global Website

You can download the latest version of NC Virtual Simulator and the instruction manuals for CNC via Mitsubishi Electric FA Global Website. To use the service, you need to create an account to register with Mitsubishi Electric FA site in advance (for free).
The following describes how to access Mitsubishi Electric FA Global Website.

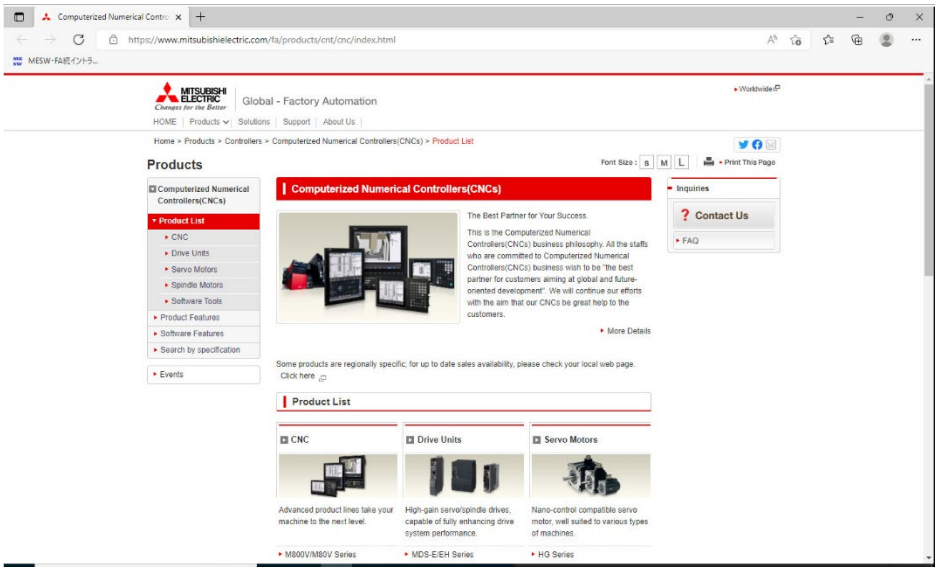
(1) Click [?] (Help).



(2) Click [MITSUBISHI ELECTRIC FA Global Website].



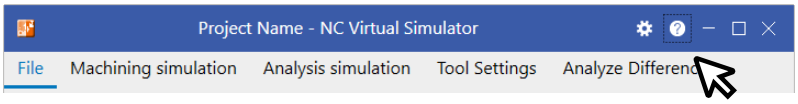
(3) Your web browser is started and Mitsubishi Electric FA Global Website is opened.
You can download documents and the latest version of software.



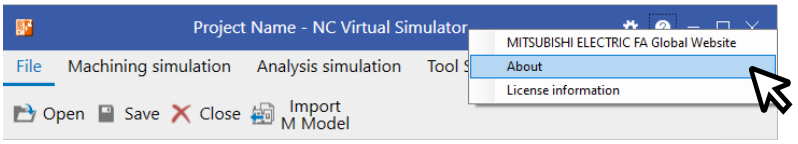
4.1.2.2 Version information

The following describes how to view the version information.

(1) Click [?] (Help).



(2) Click [About].

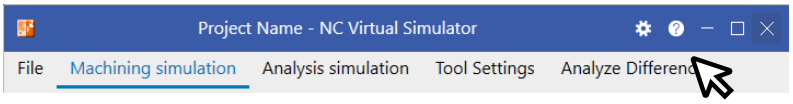


(3) The "About" dialog box appears.

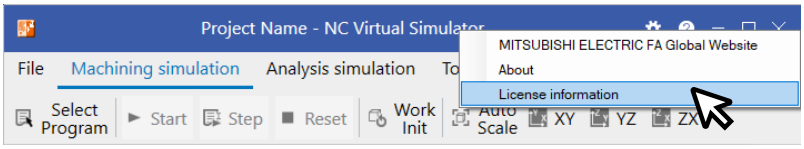
4.1.2.3 License information

The following describes how to check the license information for NC Virtual Simulator.

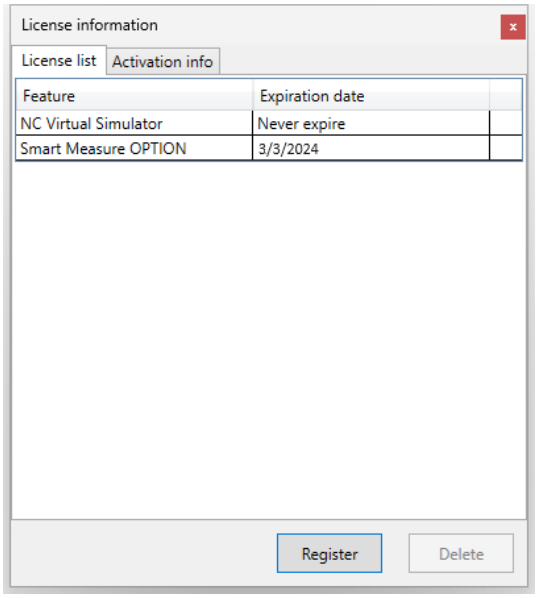
(1) Click [?] (Help).

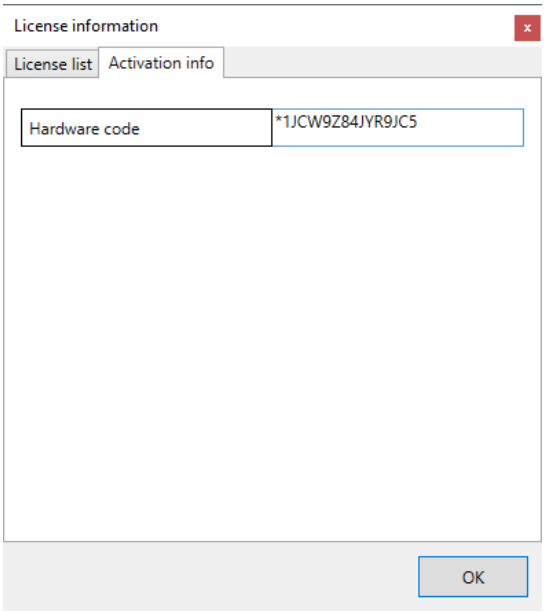


(2) Click [License information].



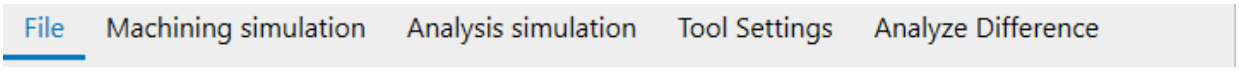
(3) The "License information" dialog box appears.
To check the hardware code, click the [Activation info] tab.





4.2 Function menu

With the function menu, you can perform operations such as to manage projects and work with machining simulation.

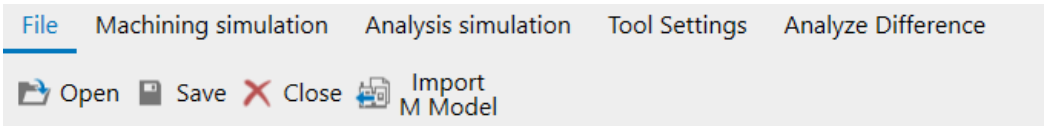


Tab	Description	Command Details
(1) File	You can open, create and save a project, import a machine model, and perform other project-related operations.	-
(2) Machining simulation	You can run machining simulation, initialize the workpiece, display machine simulation, and perform other machining simulation-related operations.	6.1.2 Machining simulation commands
(3) Analysis simulation	You can run analysis simulation, enable auto-scale and program link, and perform other analysis simulation-related operations.	7.1.2 Analysis simulation commands
(4) Tool Settings	You can set tool data.	8.2 Tool settings commands
(5) Analyze Difference	You can compare the past analysis simulation results.	9.1.2 Simulation difference analysis commands

(*) If you select a tab other than the one currently selected while simulation is running, simulation is paused.

(1) [File] tab

NC Virtual Simulator manages the settings required for running the virtual NC system digitally (on the PC) as projects (*1).
With the [File] tab, you can work with projects, including open, create, save and close a project, and import a machine model.

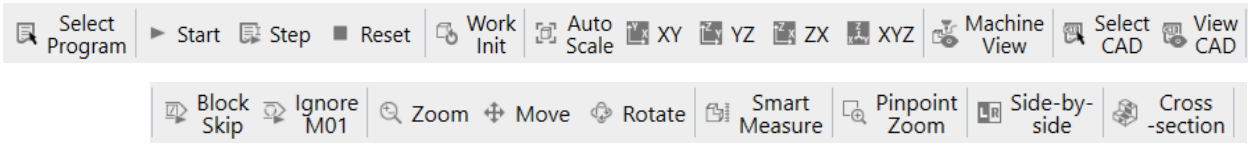


Command	Description
(1) Open	Create or open a project. For detailed information, see the following sections. "5.3.1 Creating a new project" "5.3.2 Opening a project"
(2) Save	Save the currently open project.
(3) Close	Close the currently open project.
(4) Import M Model	Import a machine model to be used in machining simulation. For detailed information, see the following section. "5.4 Importing machine model to NC Virtual Simulator"

(*1) Projects are stored in the project storage folder, which is the "\Project" folder under the data file folder (Default: "C:\MELCNC\NCVS").

(2) [Machining simulation] tab

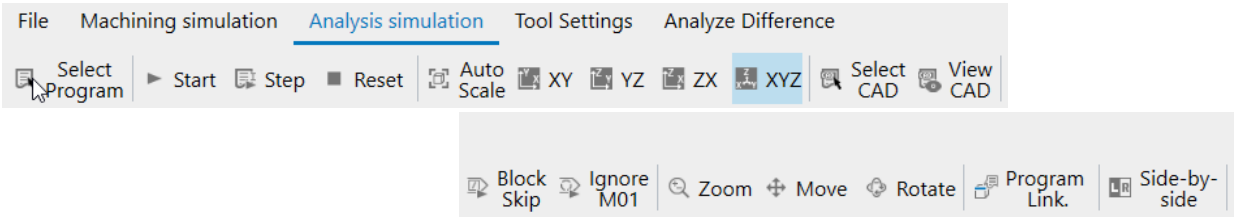
With the [Machining simulation] tab, you can run machining simulation, initialize the workpiece, display machining simulation, and perform other operations.



For detailed information about machining simulation, see "6 Machining simulation".

(3) [Analysis simulation] tab

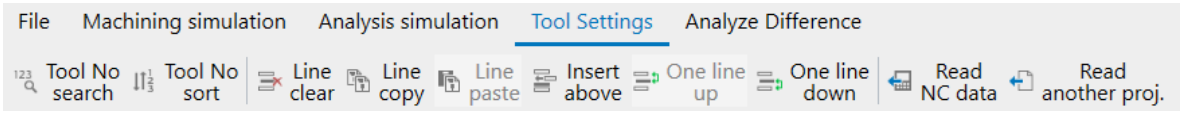
With the [Analysis simulation] tab, you can run analysis simulation and perform other operations such as auto-scale and program link.



For detailed information about analysis simulation, see "7 Analysis simulation".

(4) [Tool Settings] tab

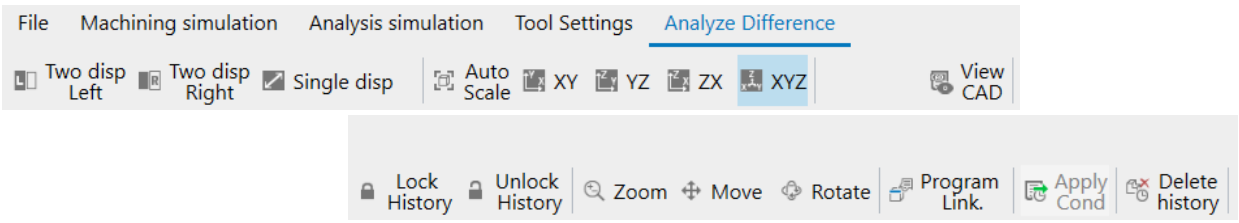
With the [Tool Settings] tab, you can set the tool to be used in machining simulation and analysis simulation.



For detailed information about tool settings, see "8 Tool settings".

(5) [Analyze Difference] tab

With the [Analyze Difference] tab, you can compare the past analysis simulation results on two screens.



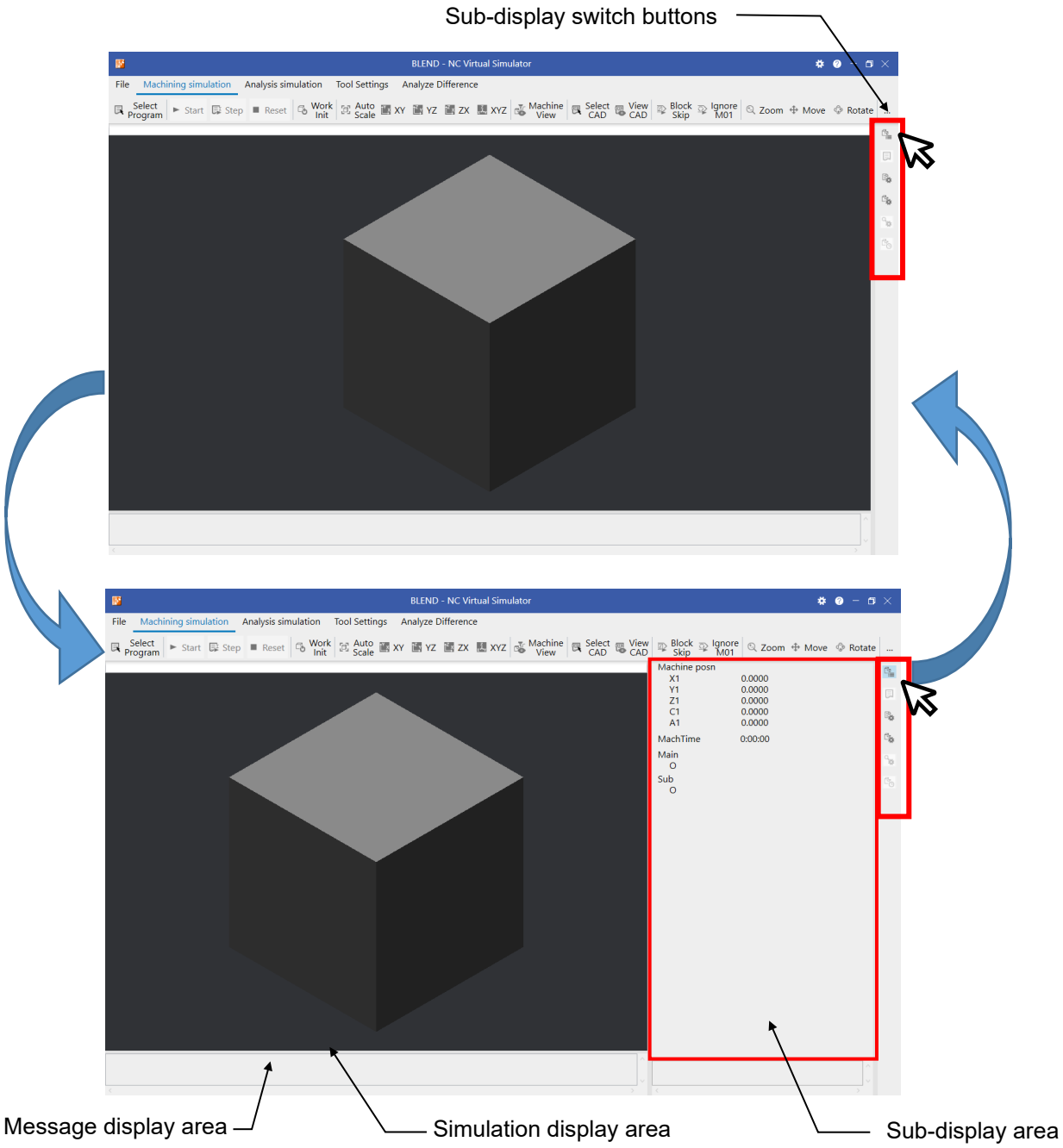
For detailed information about difference analysis, see "9 Simulation difference analysis".

4.3 Sub-display area

The sub-display area displays the processing status, machining program, NC data list, simulation settings, and other information. To switch the display in the sub-display area, click a sub-display switch button.

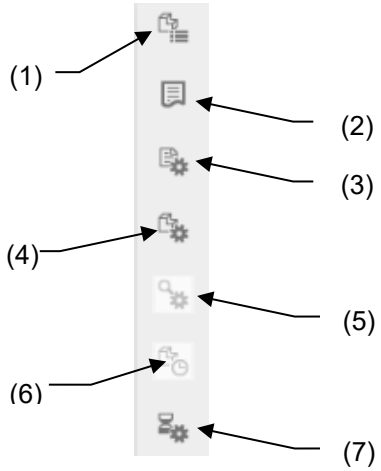
4.3.1 Showing and hiding sub-display area

The sub-display area is not displayed right after you start NC Virtual Simulator.
Click a sub-display switch button to show the sub-display area.
Click the selected sub-display switch button to hide the sub-display area.



4.3.2 Sub-display switch buttons

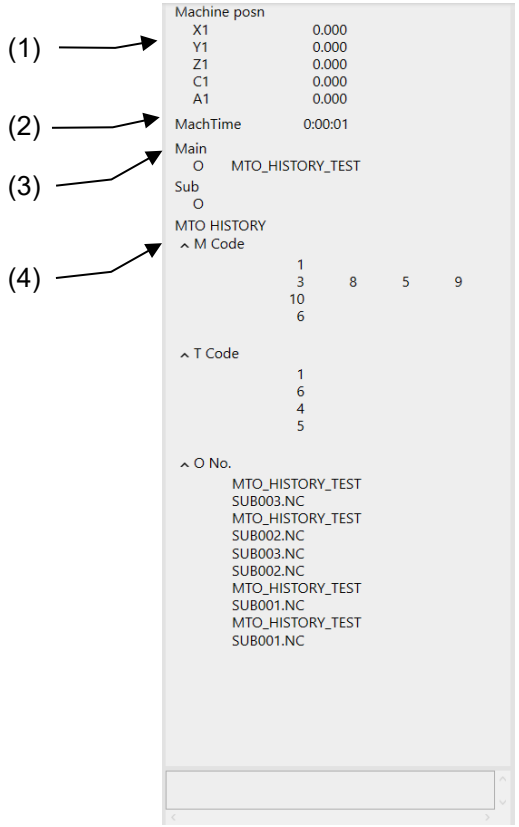
You can view the processing status of machining programs by clicking the sub-display switch button to expand the sub-display area. You can also perform the settings required for simulation in the sub-display area.



	Icon	Description	Reference
(1)	Display processing status	Click this icon to display the processing status in the sub-display. You can view the counters, machining time, machining program name and other information about simulation in the processing status display.	4.3.2.1 Processing status display
(2)	Display program	Click this icon to display the machining program in the sub-display. You can view the program to be simulated in the machining program display.	4.3.2.2 Machining program display
(3)	NC Data List	Click this icon to display the NC data list in the sub-display. You can edit the parameters, coordinate system, common variables, and tool offsets in the NC data list.	4.3.2.3 NC data list
(4)	Machining simulation settings	Click this icon to display the Machining simulation settings in the sub-display. You can configure settings such as the interference check, simulation priority, and workpiece. This icon is not selectable during analysis simulation and simulation comparison display.	4.3.2.4 Machining simulation settings
(5)	Analysis simulation settings	Click this icon to display the Analysis simulation settings in the sub-display. You can configure settings such as the analysis type, color map, and axis reversal. The settings are applied to both displayed screens during simulation comparison display. This icon is not selectable during machining simulation.	4.3.2.5 Analysis simulation settings
(6)	Simulation History	Click this icon to display the simulation history in the sub-display. You can configure settings of the simulation results to be compared and the screen display. This icon is not selectable during machining simulation and analysis simulation.	4.3.2.6 Simulation history
(7)	Machining time prediction setting	Click this icon to display the Machining time prediction setting in the sub-display. You can set the execution time to add to the predicted machining time.	4.3.2.7 Machining time prediction setting

4.3.2.1 Processing status display

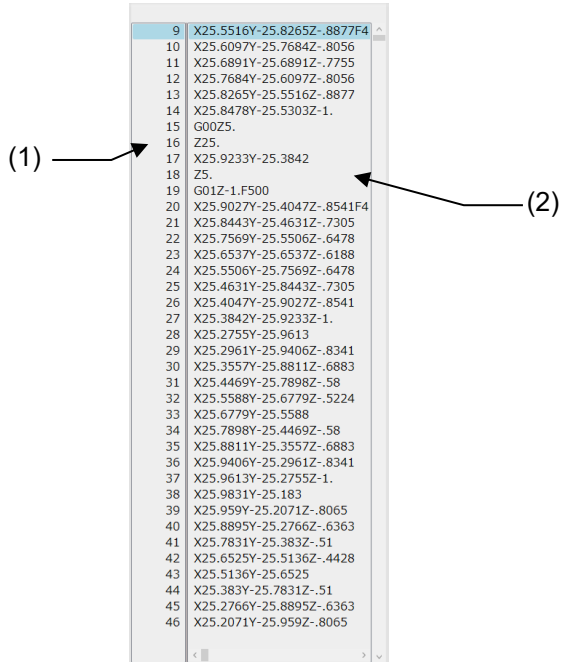
The processing status of the machining program during simulation is displayed.
When displaying side-by-side (a screen each on the left and right), the processing status of the screen currently selected (the screen with a red border) is displayed.

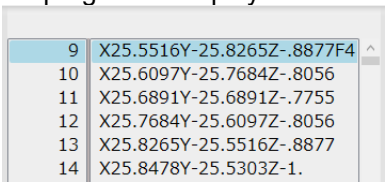


Item		Description
(1)	Counter	Displays the machine coordinate counters for the part system displayed. Displays all axes that belong to the part system displayed.
(2)	Machining time	Displays the time required for the machining.
(3)	Current program	Displays the program number of the currently running program. When the full program number cannot be displayed, "…" appears at the end to indicate omission. When a subprogram is running, the program number of the subprogram is also displayed.
	Main	
	Sub	
(4)	MTO HISTORY	Displays the M command, T command, and program number executed by the currently running machining program. The most recently executed data is displayed at the top. When the number of displayable items is exceeded, the oldest data will be deleted. When a simulation is started from the beginning, the display will be cleared.
	M Code	Displays a history of up to five executed M commands. For commands in the same block, up to four are displayed on one line. When the second or subsequent M command in the same block exceeds five digits, the symbol "…" appears before the first three digits of the M command to indicate omission of digits. Additionally, when the mouse pointer is positioned on an M command that exceeds five digits, the M command is displayed in a tooltip. M commands of fixed cycles, machine tool builder macro and interactive macro stored in the interactive macro area are not added to the history. (Note) M96, M97, M98, M99, and M198 are not added to the history.
	T Code	When there is a tool change command in the machining program, up to five T commands which were executed just before are saved in the history. In the following cases, T commands are not saved in the history. ・ T command is not executed. ・ The same T command as last time was executed. ・ The first T command is "0" after the machining simulation starts. T commands included in the following programs are not saved in the history. ・ Fixed cycles ・ Machine tool builder macro ・ Interactive macro stored in the interactive macro area T0 command is not saved in the history in analysis simulation.
	O No.	Displays a history of up to ten program numbers including the executed main program. The program numbers are added to the history when the executing program is changed (main→subprogram, subprogram level 1→subprogram level 2, subprogram→main). When L (number of repeats) is specified, one instance is added to the history. When the full program number cannot be displayed, "…" appears at the end to indicate omission.

4.3.2.2 Machining program display

The machining program to simulate is displayed.



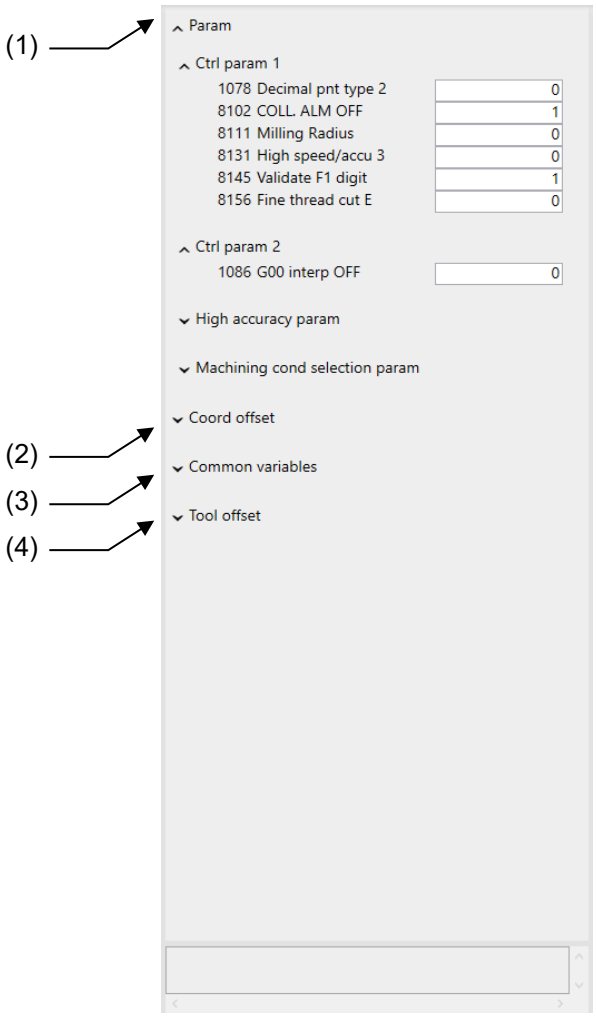
Item	Description
(1) Line number	Displays the line number in the machining program.
(2) Machining program	Displays the content of the machining program. When simulation is running, the block being executed is displayed in the first line and highlighted. When a subprogram is running, the content of the subprogram is displayed.  When simulation is stopped, the scroll bar can be used. Only the mouse can be used to scroll.

(Note 1) Only the machining programs stored in the same folder as the selected main program can be displayed. When a subprogram in a location other than the HD is called by the macro call function, etc., if another file of the same program name exists in the main program folder, the file in the main program folder is opened.

(Note 2) The displayed machining program is write-protected and cannot be edited from an application such as CAM.

4.3.2.3 NC data list

You can set various NC data for machining (e.g., coordinate system offset, common variables).



Item	Description	Range of values
(1) Param	You can view and set simulation-related parameters. These settings cannot be configured while simulation is running.	-
Ctrl param 1	You can view and set control parameter 1.	-
1078 Decimal pnt type 2	Specify the unit for position commands without a decimal point.	0/1
8102 COLL. ALM OFF	Specify how to control the interference with the workpiece from the tool radius in tool radius compensation.	
8111 Milling Radius	Specify whether diameter or radius is specified for the linear axis with which milling (cylindrical/polar coordinate) interpolation is performed.	
8131 High speed/accu 3	Specify whether to treat high-speed high-accuracy control II commands as high-speed high-accuracy control III commands.	

Item	Description	Range of values
8145 Validate F1 digit	Specify how to program F commands, whether by using a 1-digit code or by directly specifying a number.	
8156 Fine thread cut E	Specify what address E specifies for inch screw cutting.	
Ctrl param 2	You can view and set control parameter 2.	-
G00 interp OFF	Specify the type of traversing path for G00	0/1
High accuracy param (*1)	You can view and set high-accuracy parameters.	-
1149 cireft (Arc deceleration speed change)	Select whether to decelerate at the arc entrance or exit.	0/1
1205 G0bdcc (Acceleration and deceleration before G0 interpolation)	Specify the G0 acceleration/deceleration timing.	0 to 2
1209 cirdcc (Arc deceleration speed)	Specify the deceleration speed at the arc entrance or exit.	1 to 999999 (mm/min)
1569 SfiltG0 (G00 soft acceleration/deceleration filter)	Specify the filter time constant for smoothly changing the acceleration rate for the rapid traverse acceleration/deceleration in pre-interpolation acceleration/deceleration.	0 to 200 (ms)
1571 SSSdis (SSS control adjustment coefficient fixed value specification)	Fix the shape recognition range for SSS control.	0/1
7914 ROT_PREFILT (Rotary axis prefilter time constant)	Specify the time constant for rotary axis prefilter.	0 to 200 (ms)
8034 AccClamp ON	Select the method for clamping the cutting speed.	0/1
8036 CordecJudge	Select the condition to decide a corner.	0/1
8038 Path recog. range	Specify the range to recognize the tool paths adjoining to the command position when the smooth fairing function is ON.	0 to 100.000 (mm)
8039 Comp. range limit	Specify the upper limit of the distance between the command position and compensation position when the smooth fairing function is ON.	-1.000 to 100.000 (mm)
8092 ClampCoeff	Specify the clamp speed at the curved section configured of fine segments.	1 to 100
8094 DccWaitAdd	Specify the time to wait for deceleration when the speed FB does not drop to the clamp speed.	0 to 100 (ms)
8096 Deceler. coeff. ON	Select whether to enable the speed coefficients that are used for compensating for a path error and clamp speed under SSS control.	0/1
8097 Corner decel coeff	Specify the compensation coefficient to be used for adjusting a path error and clamp speed at a corner under SSS control.	0 to 2000 (%)

Item	Description	Range of values
Theor R decrease	Displays the theoretical radius decrease error amount, ΔR (mm), from the automatic calculation by NC.	-
R5mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 5 (mm) radius.	-
R1mm arc dec speed	Displays a deceleration speed(mm/min) along an arc of 1 (mm) radius.	-
Theor R decrease	Displays the theoretical radius decrease error amount, ΔR (mm), from the automatic calculation by NC.	-
8098 Arc clamping speed coefficient	Specify the compensation coefficient to be used for adjusting a path error and clamp speed on an arc under SSS control.	0 to 2000 (%)
Theor R decrease	Displays the theoretical radius decrease error amount, ΔR (mm), from the automatic calculation by NC.	-
R5mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 5 (mm) radius.	-
R1mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 1 (mm) radius.	-
12051 Jerk_filtG1	Specify the time constant of filter that is used for smoothing the change of jerk when pre-interpolation acceleration/deceleration is performed in cutting feed.	0 to 50 (ms)
Notch frequency Hz	Displays the notch frequency for the S-pattern filter for #12051.	-
12052 Jerk_filtG0	Specify the time constant of filter that is used for smoothing the change of jerk when pre-interpolation acceleration/deceleration is performed in cutting feed.	0 to 50 (ms)
12067 Tolerance spd coef	Specify the compensation coefficient to adjust a path error or clamp speed in the corner while tolerance control is ON.	0 to 2000 (%)
12068 Smoothing range	Normally set "0" in this parameter. If you run a program with micro segments and its reciprocating paths are uneven, set the parameter to about 3 to 5 times the length of the programmed segments.	0.000 to 10.000 (mm)
12069 Corner angle	Specify the corner recognition angle. Normally set to "0".	0.000 to 180.000 (°)
Machining cond selection param (*1)	You can view and set the machining conditions and parameters for each machining condition.	-
12066 Tolerance ctrl ON	Specify whether or not to enable tolerance control.	0/1
Machining condition	Choose a machining condition. P0: Reference parameter n-m: Machining condition Pn-Qm (n, m = 1, 2, 3)	P0, 1-1, 1-2, 1-3, 2-1, 2-2, 2-3, 3-1, 3-2, 3-3
Standard parameter/Machining cond P-Q	You can view and set the parameters for each machining condition.	-

Item	Description	Range of values
Machining condition name	Displays the value of the [Machining condition] equivalent to G120.1 P_ Q_.	-
1206 G1bF (Maximum speed)	Set the cutting feedrate applied to pre-interpolation acceleration/deceleration.	1 to 999999 (mm/min)
1207 G1btL (Time constant)	Set the cutting feed time constant applied to pre-interpolation acceleration/deceleration.	Without high-accuracy control time constant extension 0 to 5000 (ms) With high-accuracy control time constant extension 0 to 30000 (ms)
Cutting feed Acc	Displays the result of the calculation from #1206 and #1207. (a) Cutting feed acceleration $acc = \frac{G1bF/60}{G1btL/1000} \times \frac{1}{9.80665 \times 1000}$ *G1bF* [mm/min]: Maximum speed (#1206) *G1btL* [ms]: Time constant (#1207) *acc* [G]: Cutting feed acceleration 1(G)=9.80665(m/s²)	-
1568 SfiltG1 (G01 soft acceleration/deceleration filter)	The time constant of the filter that smooths out the change in acceleration applied to the cutting feed acceleration/deceleration when pre-interpolation acceleration/deceleration is active.	0 to 200 (ms)
Notch frequency Hz	Displays the result of the calculation from #1568. (b) Notch frequency (SfiltG1) $f = \frac{1}{SfiltG1/1000}$ *SfiltG1* [ms]: SfiltG1 (#1568) *f* [Hz]: Notch frequency	-
1570 Sfilt2 (*2) (Soft acceleration/deceleration filter 2)	Set the time constant of the filter that further smooths out the change in acceleration when pre-interpolation acceleration/deceleration is active.	0 to 200 (ms)
Notch frequency Hz	Displays the notch frequency for the S-curve filter for #1570.	-
2659 tolerance (Tolerance)	Set the tolerance (allowable error) applied to tolerance control.	0.000 to 100.000 (mm)
8019 R COMP (*2)	Set a compensation coefficient for reducing a control error in the reduction of a corner roundness and arc radius.	0 to 99 (%)
Theor R decrease	Displays the theoretical radius decrease error amount, ΔR (mm), from the automatic calculation by NC.	-
R5mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 5 (mm) radius.	-
R1mm arc dec speed	Displays a deceleration speed(mm/min) along an arc of 1 (mm) radius.	-
8020 DCC ANGLE (*2)	Set the minimum value of an angle (external angle) that should be assumed to be a corner.	0 to 89 (°)
Theor Cor dull amt	Displays the corner dull amount Δc (mm) in respect to the corner's angle (external angle) θ (°).	-

Item	Description	Range of values
Corner dclr speed	Display corner deceleration speed Fc (mm/min) for the corner of the angle (external angle) with θ (°).	-
Theo 90deg dulness	Display corner dull amount when the angle is 90 degree.	-
90deg Cor dec spd	Display corner deceleration speed when the angle is 90 degree.	-
8021 COMP_CHANGE (*2)	Select whether to share or separate the compensation coefficient at the corner/curve during the high-accuracy control mode. Parameters other than reference parameters are read-only.	0/1
8022 CORNER COMP (*2)	Set the compensation coefficient to further reduce or increase the roundness at the corner during the high-accuracy control mode.	-1000 to 99 (%)
Theor Cor dull amt	Displays the corner dull amount Δc (mm) in respect to the corner's angle (external angle) θ (°).	-
Corner dclr speed	Display corner deceleration speed Fc (mm/min) for the corner of the angle (external angle) with θ (°).	-
Theo 90deg dulness	Display corner dull amount when the angle is 90 degree.	-
90deg Cor dec spd	Display corner deceleration speed when the angle is 90 degree.	-
8023 CURVE COMP (*2)	Set the compensation coefficient to further reduce or increase the radius reduction amount at the curve (arc, involute, spline) during the high-accuracy control mode.	-1000 to 99 (%)
Theor R decrease	Displays the theoretical radius decrease error amount, ΔR (mm), from the automatic calculation by NC.	-
R5mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 5 (mm) radius.	-
R1mm arc dec speed	Displays a deceleration speed (mm/min) along an arc of 1 (mm) radius.	-
8025 SPLINE ON (*2)	Specify whether to enable the fine spline function. Parameters other than reference parameters are read-only.	0/1
8026 CANCEL ANG. (*2)	Set the angle where the spline interpolation is temporarily canceled.	0 to 180 (°)
8027 Toler-1 (*2)	Set the maximum chord error (tolerance) in a block that includes an inflection point.	0.000 to 100.000 (mm)
8028 Toler-2 (*2)	Set the maximum chord error (tolerance) in a block that includes no inflection point.	0.000 to 100.000 (mm)
8029 FairingL (*2)	Set the length of the block subject to fairing.	0.000 to 100.000 (mm)
8030 MINUTE LENGS (*2)	Set the fine-segment length where the spline interpolation is temporarily canceled.	-1 to 127 (mm)
8033 Fairing ON (*2)	Select whether or not to use the fairing or smooth fairing function.	0 to 3

Item				Description	Range of values
			8037 CorJudgeL (*2)	Set the length of the block to be excluded when deciding a corner.	0 to 99999.999 (mm)
			8090 SSS ON (*2)	Set whether to enable the SSS control with G05 P10000.	0/1
			8091 StdLength (*2)	Set the maximum value of the range for recognizing the shape.	0.000 to 100.000 (mm)
			8093 StepLeng (*2)	Set the width of the step at which the speed is not to be decelerated. (Approximately the same as the CAM path difference [Tolerance].)	-1.000 to 0.100 (mm)
			12070 Sfilt2_tol (Tolerance control: Soft acceleration/deceleration filter 2)	The time constant of the filter that smooths out the change in acceleration during tolerance control.	0 to 200 (ms)
(2)	Coord offset			You can view and set the simulation-related coordinate systems. The number of sets of extended workpiece coordinate systems that can be displayed varies depending on the option. These settings cannot be configured while simulation is running.	-
	EXT			Displays the set value of the external workpiece coordinate offset.	-99999.999 to 99999.999
	G54-G59			Displays the set value of the workpiece coordinate system.	
	G54.1P			When P number of extended workpiece coordinate offset is set, you can view and set the extended workpiece coordinate offset of that P number. This item is displayed only when extended workpiece coordinate system selection (48 sets, 96 sets, 300 sets) is enabled.	(P number) 1 to 300 *Varies depending on the number of extended workpiece coordinate systems. (Offset) -99999.999 to 99999.999
(3)	Common variables			You can view and set the simulation-related common variables. These settings cannot be configured while simulation is running.	-
	First common var. number			Set the first number of the common variables to be displayed. #400 to #499 are available when the number of sets of common variables is 700 or greater and parameter "#1336 (#400_Valtype) " is "1". Extended common variables cannot be specified.	100 to 199, 400 to 999
	#n to #n + 9			Display ten common variable numbers from the first common variable number (n). The variable names cannot be displayed or set.	-999,999,999 to 999,999,999

Item		Description	Range of values
(4) Tool offset		You can view and set the simulation-related tool offsets. The items displayed vary depending on the compensation type. These settings cannot be configured while simulation is running. *When tool compensation type III (#1037=3/#1046=1) is selected, tool offsets are not displayed.	-
	First number of tool offset	Set the first number of tool offset to be displayed.	1 to 999
	#n to #n + 9	Displays ten offsets from the first number (n) of tool offset. When compensation type I (#1037=1/#1046=0) is selected, "offset" is displayed. When compensation type II (#1037=2/#1046=0) is selected, "length offset/length wear amount" and "radius offset/radius wear amount" are displayed.	-9999.999 to 9999.999

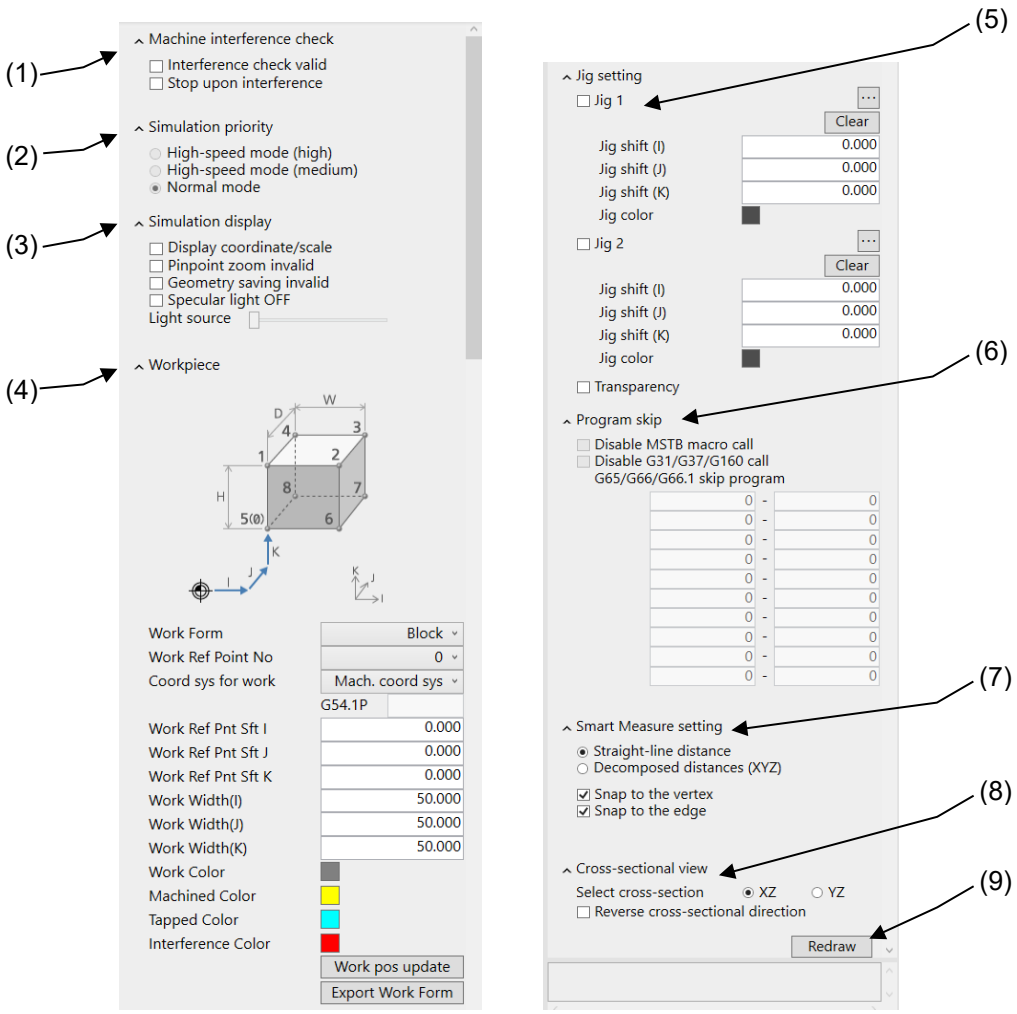
(*1) When it is linked with NC Trainer2 plus, "High accuracy param" and "Machining cond selection param" are not displayed. Check and modify the values on the parameter screen of NC Trainer2 plus.

(*2) When "#12066 Tolerance ctrl ON" is "1 (Enable)", this item is read-only.

4.3.2.4 Machining simulation settings

You can set parameters related to machining simulation operation and display, and workpiece shapes and dimensions.

For details, see "6 Machining simulation".

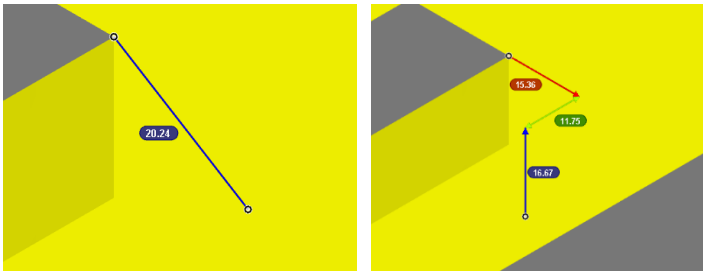


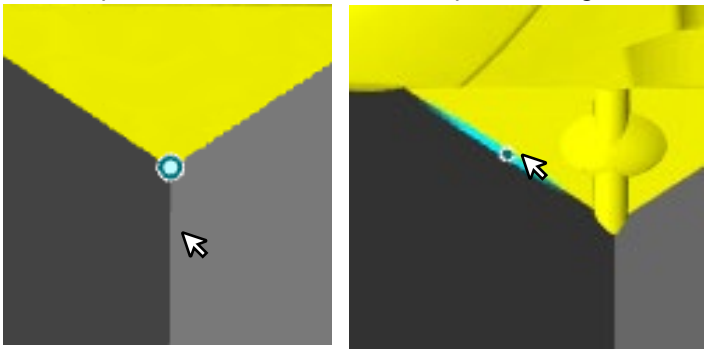
Item		Description	Range of values
(1)	Machine interference check	Interference check-related settings	-
	Interference check valid	When this checkbox is checked, interference between the tool, workpiece, and machine parts is checked when simulation is run. If interference is detected, an alarm is displayed.	ON/OFF
	Stop upon interference	When this checkbox is checked, simulation is stopped when interference is detected in interference check.	ON/OFF
(2)	Simulation priority	Select the simulation priority. (Default: Normal mode) You cannot switch the simulation priority while simulation is running. When you switch the simulation priority after simulation finishes, click [Start] or [Step] again. Simulation is run according to the priority you selected.	High-speed mode (high), High-speed mode (medium), Normal mode
	High-speed mode (high)	Simulation is performed at the highest speed. This operation mode is mainly used to grasp the machining time prediction, check whether machine interference exists, etc. However, the analysis accuracy is lower, and it is not adequate to check the shape of slight blemishes. (Note) Step operation is not available in this mode.	
	High-speed mode (medium)	Simulation is performed at high speed. This operation mode is mainly used to check the analysis results that require accuracy, grasp the machining time prediction, check whether machine interference exists, etc. (Note) Step operation is not available in this mode.	
	Normal mode	Simulation is performed at normal speed. This operation mode can be used for the most purposes to check the analysis results that require accuracy, check the behavior by step operation, identify the interference block, etc.	
(3)	Simulation display	Simulation display-related settings	-
	Display coordinate/scale	When this checkbox is checked, the directions of the coordinate system and the scale for the view range are displayed in the 3D graphics area.	ON/OFF
	Pinpoint zoom invalid	When this checkbox is checked, the [Pinpoint zoom] command is not selectable.	ON/OFF
	Geometry saving invalid	When this checkbox is checked, the workpiece shape of the machining simulation is not saved when the project is saved. (Default: OFF) Check the checkbox when it takes time to save or open a project. By doing so, it takes less time to save and open the project.	ON/OFF
	Specular light OFF	When this checkbox is checked, the reflected light on the workpiece is reduced. (Default: OFF)	ON/OFF
	Light source	Set the light source position. Move the slider right or left to move the light source position. (*1) The light source position moves in the order of upper right (the leftmost of the slider) → right → lower right → bottom → lower left → left → upper left → top (the rightmost of the slider).	8 levels
(4)	Workpiece	Workpiece-related settings	-
	Workpiece guide figure	Displays the guide figure for the set workpiece. (*2)	-

Item		Description	Range of values
	Work Form	Select the work form (the shape of the workpiece).	Block, Pipe, Arbitrary shape
	When [Work Form] is Block		
	Work Ref Point No.	Select the reference point No. of the workpiece.	0 to 8
	Coord sys for work	Select the coordinate system for the workpiece. The coordinate of the workpiece is determined by regarding the selected coordinate system offset as a reference point.	Machine coordinate system, G54 to G59, G54.1P
	G54.1P	Specify the P number of when the extended workpiece coordinate system "G54.1P" is selected for [Coord sys for work]. This is displayed only when extended workpiece coordinate system selection (48 sets, 96 sets, 300 sets) is enabled.	1 to 300 (*) Varies depending on the number of extended workpiece coordinate systems.
	Work Ref Pnt Sft I Work Ref Pnt Sft J Work Ref Pnt Sft K	Specify the shift amount of the reference point of the workpiece from the coordinate system offset position selected with [Coord sys for work].	-99999.999 to 99999.999 (mm)
	Work Width (I) Work Width (J) Work Width (K)	Specify the dimension of the workpiece.	Value range: 0.001 to 99999.999 (mm)
	Work Color Machined Color Tapped Color Interference Color	Specify the colors of the workpiece, machined surface, tapped surface, and interfering surface. When the workpiece is displayed for the first time, the following colors are selected. Work Color: Gray Machined Color: Yellow Tapped Color: Light blue Interference Color: Red	The colors in the "Color" dialog
	Work pos update button	Applies changes of the workpiece reference point coordinate to the machining simulation window. This is enabled when [Work Form], [Work Ref Point No.], [Coord sys for work] or [Work Ref Pnt Sft I/J/K] is changed.	-
	Export Work Form button	Save the workpiece shape currently displayed as a file. The workpiece shape is saved as a DAT format file (*.dat)	-
	When [Work Form] is Pipe		
	Work Ref Point No.	Select the reference point No. of the workpiece.	0 to 1
	Coord sys for work	Select the coordinate system for the workpiece. The coordinate of the workpiece is determined by regarding the selected coordinate system offset as a reference point.	Machine coordinate system, G54 to G59, G54.1P
	G54.1P	Specify the P number of when the extended workpiece coordinate system "G54.1P" is selected for [Coord sys for work]. This is displayed only when extended workpiece coordinate system selection (48 sets, 96 sets, 300 sets) is enabled.	1 to 300 (*) Varies depending on the number of extended workpiece coordinate systems.

Item		Description	Range of values
	Work Ref Pnt Sft I Work Ref Pnt Sft J Work Ref Pnt Sft K	Specify the shift amount of the reference point of the workpiece from the coordinate system offset position selected with [Coord sys for work].	-99999.999 to 99999.999 (mm)
	Extrnl Diameter D	Set the external diameter of the workpiece. (Default: 150 mm)	0.001 to 99999.999 (mm)
	Inner Diameter E	Set the inner diameter of the hole of the workpiece. (Default: 0)	
	Work Length L	Set the length of the workpiece. (Default: 300 mm)	
	Display direction	Select the display orientation of the workpiece.	Horizontal/Vertical
	Work Color Machined Color Tapped Color Interference Color	Specify the colors of the workpiece, machined surface, tapped surface, and interfering surface. When the workpiece is displayed for the first time, the following colors are selected. Work Color: Gray Machined Color: Yellow Tapped Color: Light blue Interference Color: Red	The colors in the "Color" dialog
	Work pos update button	Applies changes of the workpiece reference point coordinate to the machining simulation window. This is enabled when [Work Form], [Work Ref Point No.], [Coord sys for work] or [Work Ref Pnt Sft I/J/K] is changed.	-
	Export Work Form button	Save the workpiece shape currently displayed as a file. The workpiece shape is saved as a DAT format file (*.dat)	-
	When [Work Form] is Arbitrary shape		
	Workpiece shape file	Select a workpiece shape file. Click the [...] button to display the file selection dialog. Select either a file output from CAD, etc. (STL format) or an exported file of a workpiece shape (DAT format). When a file is selected, the file name is displayed on the left side of the [...] button. When the full file name cannot be displayed, "…" appears at the end to indicate omission. Files in STL format (*.stl) and DAT format (*.dat) can be selected.	・ File path: 256 characters or less ・ File name: 32 characters or less
	Work Ref Point No.	Select the reference point No. of the workpiece. Set this to "0" or "1" when an exported workpiece shape file with [Work Form] set to Pipe is read.	0 to 8
	Coord sys for work	Select the coordinate system for the workpiece. The coordinate of the workpiece is determined by regarding the selected coordinate system offset as a reference point.	Machine coordinate system, G54 to G59, G54.1P
	G54.1P	Specify the P number of when the extended workpiece coordinate system "G54.1P" is selected for [Coord sys for work]. This is displayed only when extended workpiece coordinate system selection (48 sets, 96 sets, 300 sets) is enabled.	1 to 300 (*) Varies depending on the number of extended workpiece coordinate systems.

Item			Description	Range of values
	Work Ref Pnt Sft I		Specify the shift amount to the reference point of the workpiece from the coordinate system offset position selected with [Coord sys for work].	-99999.999 to 99999.999 (mm)
	Work Ref Pnt Sft J			
	Work Ref Pnt Sft K			
	Work Width (I)		When a file that was output using the [Export Work Form] button is read, the shape is displayed based on the size of the block that encloses the workpiece shape before the simulation is executed. When a file that was not output using the [Export Work Form] button is read, the shape is displayed based on the size of the block that encloses the workpiece shape. In both cases, this value cannot be changed.	-
	Work Width (J)			
	Work Width (K)			
	Work Color		Specify the colors of the workpiece, machined surface, tapped surface, and interfering surface. When the workpiece is displayed for the first time, the following colors are selected. Work Color: Gray Machined Color: Yellow Tapped Color: Light blue Interference Color: Red	The colors in the "Color" dialog
	Machined Color			
	Tapped Color			
	Interference Color			
	Work pos update button		This is enabled when [Work Form], [Work Ref Point No.], [Coord sys for work] or [Work Ref Pnt Sft I/J/K] is changed, and applies changes of the workpiece reference point coordinate to the machining simulation window.	-
	Export Work Form button		Save the workpiece shape currently displayed as a file. The workpiece shape is saved as a DAT format file (*.dat).	-
(5)	Jig setting		Jig model-related settings	-
	Jig 1		Set whether to show or hide the jig model 1. (Default: OFF)	ON/OFF
		File name	Displays the file name registered as "Jig 1". When the full file name cannot be displayed, "…" appears at the end to indicate omission.	-
		…	Select a model file to be registered as "Jig 1". Files in STEP format (*.stp, *.step) and in STL format (*.stl) can be selected.	-
		Clear	Deletes the registered jig model 1.	-
		Jig shift (I)	Specify the shift amount from the jig model 1 based on the workpiece position determined with the coordinate system offset position selected with [Coord sys for work] and the values of [Work Ref Pnt Sft]. (Default: 0)	-99999.999 to 99999.999 (mm)
		Jig shift (J)		
		Jig shift (K)		
		Jig color	Specify the jig model color. Blue is selected in the initial display.	The colors in the "Color" dialog
	Jig 2		Set whether to show or hide the jig model 2. (Default: OFF)	ON/OFF
		File name	Displays the file name registered as "Jig 2". When the full file name cannot be displayed, "…" appears at the end to indicate omission.	-
		…	Select a model file to be registered as "Jig 2".	-
		Clear	Deletes the registered jig model 2.	-
		Jig shift (I)	Specify the shift amount from the jig model 2 based on the workpiece position determined with the coordinate system offset position selected with [Coord sys for work] and the values of [Work Ref Pnt Sft]. (Default: 0)	-99999.999 to 99999.999 (mm)
		Jig shift (J)		
		Jig shift (K)		

Item			Description	Range of values
		Jig color	Specify the jig model color. Blue is selected in the initial display.	The colors in the "Color" dialog
	Transparency		When this checkbox is checked, the jig model is displayed transparently. (Default: OFF)	ON/OFF
(6)	Program skip		Set the machining programs to be skipped.	-
	Disable MSTB macro call		Disables the user macro program call by macro call of miscellaneous command (macro call of M, S, T, and B codes).	ON/OFF
	Disable G31/G37/G160 call		Disables G31/G37/G160 commands.	ON/OFF
	G65/G66/G66.1 skip program		Disables macro call by G65/G66/G66.1 commands that specify a range. When the following values are set, range specification is disabled. ・ Left column = 0 / Right column = 0 ・ Left column > Right column ・ Left column = 0 to 99999999 / Right column = 100010000 to 199999998	0 to 99999999/ 100010000 to 199999998
(7)	Smart Measure setting		Smart measurement-related settings (Optional function)	-
	Measurement mode		There are the following measurement modes. <Straight-line distance> <Decomposed distances (XYZ)>  	Straight-line distance/Decomposed distances (XYZ)
		Straight-line distance	Displays the distance between two points as a measurement value of the straight line. (Default: 0)	
		Decomposed distances (XYZ)	Displays the distance between two points as a measurement value of the decomposed orthogonal three axes (XYZ).	

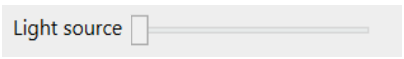
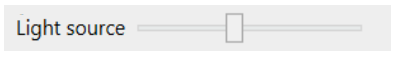
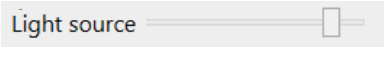
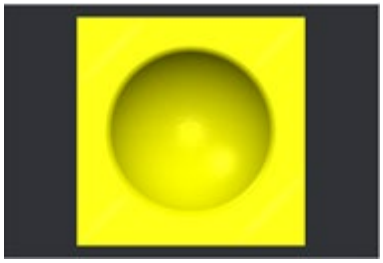
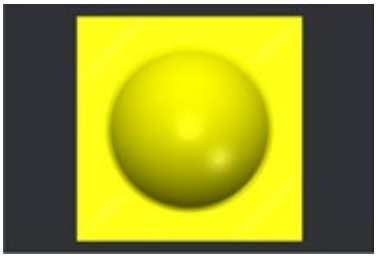
Item			Description	Range of values
	Measurement option		There are the following measurement options. <Snap to the vertex> <Snap to the edge> 	Snap to the vertex/Snap to the edge
		Snap to the vertex	When this checkbox is checked, snap to the vertex is enabled. (Default: ON) When the mouse cursor is moved close to a vertex, the vertex is highlighted so that it is easier to select the measurement point	ON/OFF
		Snap to the edge	When this checkbox is checked, snap to the edge shape of straight line or arc is enabled. (Default: ON) When the mouse cursor is moved close to the edge of straight line or arc, the edge is highlighted so that it is easier to select the measurement point.	ON/OFF
(8)	Cross-sectional view		Cross-sectional view-related settings	-
	Select cross-section		Select the plane for cross-sectional view. (Default: XZ)	XZ/YZ
		XZ	Displays the cross section at the position perpendicular to the coordinate system J axis of the workpiece.	
		YZ	Displays the cross section at the position perpendicular to the coordinate system I axis of the workpiece.	
	Reverse cross-sectional direction		Reverses the direction of the cross-sectional view. (Default: OFF)	ON/OFF
(9)	Redraw button		Redraws the last executed machining simulation result at high speed. The progress window appears during the redraw, and the other operations are not available. When [Cancel] is clicked on the progress window, the redraw can be stopped. While simulation is running, the redraw cannot be performed even by clicking this button. When the simulation history of the sub-display "Simulation history" is selected and [Apply Cond to project] is performed, the redraw can be performed with the selected history simulation result.	-

(Note) If there is a large difference between the values of Work Width (I), Work Width (J) and Work Width (K), or excessively small values are set, the error message "Out of range." appears. Correct the set values.

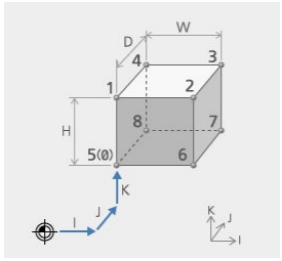
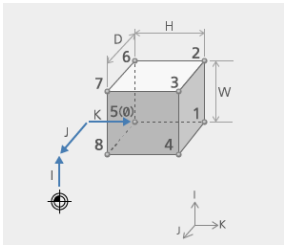
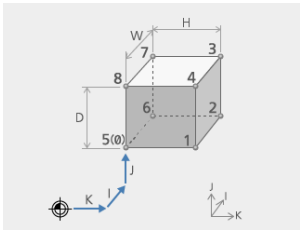
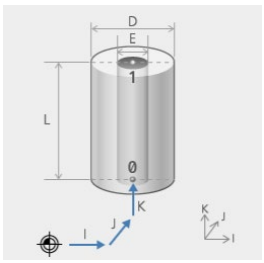
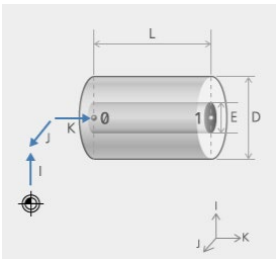
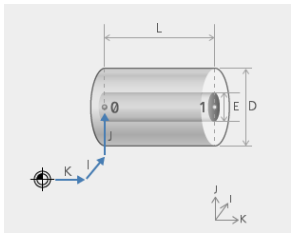
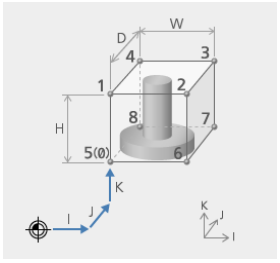
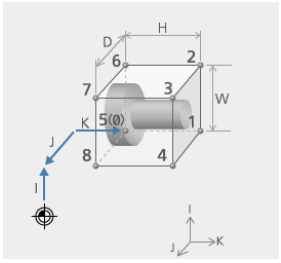
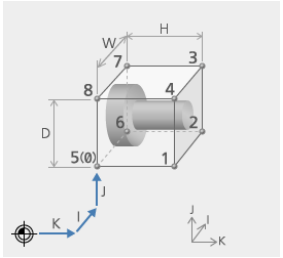
(Example1) Work Width (I) 1000; Work Width (J) 0.099 or smaller

(Example2) Work Width (I) 0.100 or smaller; Work Width (J) 0.100 or smaller; Work Width (K) 0.100 or smaller

(*1) When the slider position (8 levels) is changed, the light source position is moved as follows.

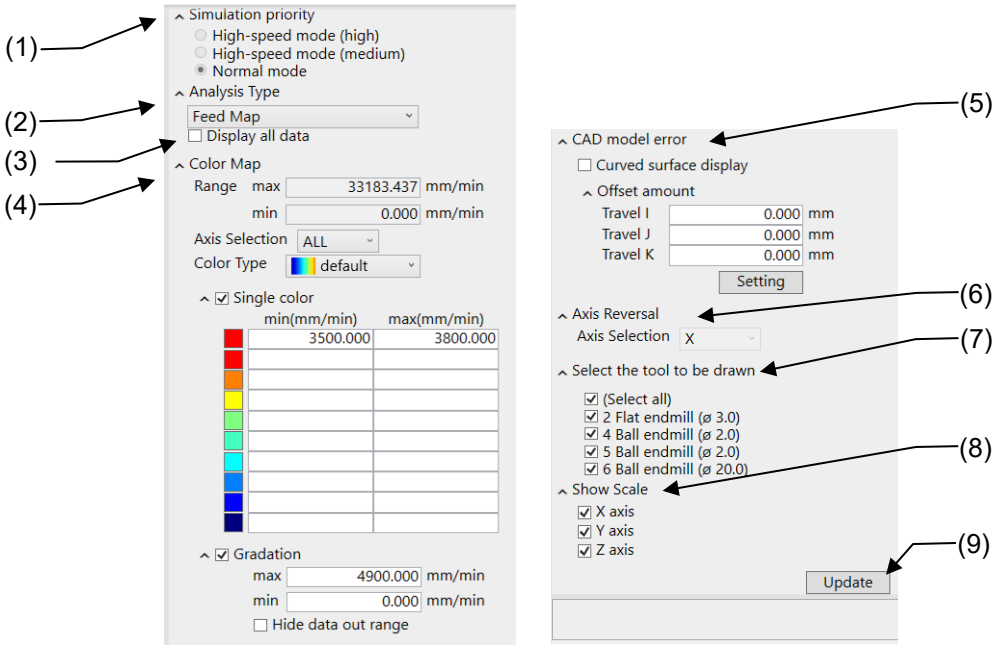
Slider position: 1/8	Slider position: 4/8	Slider position: 7/8
		
Light source position: Upper right	Light source position: Bottom	Light source position: Upper left
		

(*2) The guide figure is switched according to the Work Form specified.

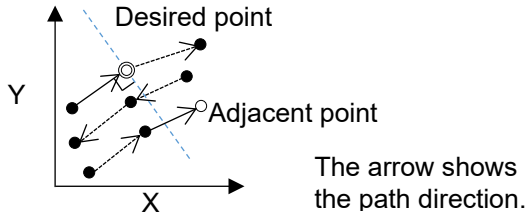
Work Form	Guide figure		
	Vertical machining center	Horizontal machining center	
		X axis is vertical	Y axis is vertical
Block			
Cylinder/Pipe			
Arbitrary shape			

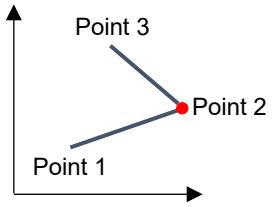
4.3.2.5 Analysis simulation settings

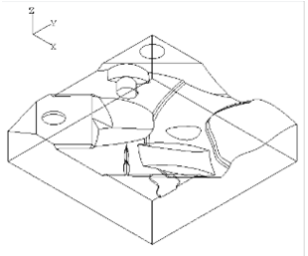
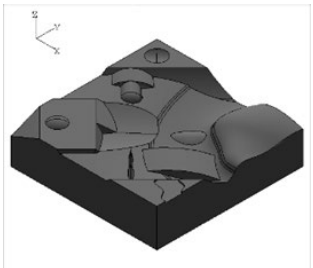
You can set parameters related to analysis simulation operation and display.
For details, see "7 Analysis simulation".



Item		Description	Range of values
(1)	Simulation priority	Select the simulation priority. When you switch the simulation priority after simulation finishes, click [Start] or [Step] again. Simulation is run according to the priority you selected.	High-speed mode (high), High-speed mode (medium), Normal mode
	High-speed mode (high)	Simulation is performed at the highest speed. This operation mode is mainly used to grasp the machining time prediction. However, the analysis accuracy is lower, and it is not adequate to check the shape of slight blemishes. (Note) Step operation is not available in this mode.	
	High-speed mode (medium)	Simulation is performed at high speed. This operation mode is mainly used to check the analysis results that require accuracy, grasp the machining time prediction, etc. (Note) Step operation is not available in this mode.	
	Normal mode	Simulation is performed at normal speed. This operation mode can be used for the most purposes to check the analysis results that require accuracy, check the behavior by step operation, etc.	
(2)	Analysis Type	Select the type of analysis. (Default: Position)	Position, Feed Map, Acceleration Map, Posit. Diff Map, Feed Diff Map, CAD model error, Axis Reversal
	Position	Shows the machining shape that represents the coordinates of the position data. This is used to check the tool path.	
	Feed Map	Visualizes the feedrate obtained from the position data of the axis selected by [Axis Selection] as a color map. This is used to check the change in feedrate.	

Item		Description	Range of values
	Acceleration Map	Visualizes the acceleration obtained from the position data of the axis selected by [Axis Selection] as a color map. This is used to check the change in acceleration.	
	Posit. Diff Map	Visualizes the position change amount of the axis selected by [Axis Selection], as a color map. The change amount is obtained from the adjacent path in the same machining direction. This is used to check the degree of variation of adjacent path position. Position difference is a position difference between the desired point (◎) and the adjacent point (○). The position difference is converted into an absolute position and displayed on the line segment till the desired point in a color map.  (Note1) The adjacent point is searched for among only adjacent points on the XY plane. (Note2) When the screen is changed to another screen while [Posit. Diff Map] is selected and the screen is returned to the analysis simulation screen, the simulation graphic is not updated. To update the view, click the [Update] button.	
	Feed Diff Map	Visualizes the speed change amount of the axis selected by [Axis Selection], as a color map. The change amount is obtained from the adjacent path in the same machining direction. This is used to check the degree of variation of adjacent path speed. Speed difference is a speed difference between the desired point (◎) and the adjacent point (○). The speed difference is converted into an absolute position and displayed on the line segment till the desired point in a color map.	
	CAD model error	Visualizes errors between the machining program path and the product CAD model shape on the path as a color map.	

<table>
<tr><th colspan="2">Item</th><th>Description</th><th>Range of values</th></tr>
<tr><td></td><td>Axis Reversal</td><td>Maps the positions where X, Y, or Z axis is reversed (points of switchover between plus and minus in the position data) with a red plot point. This is used to check the position where the direction of the axis is reversed.
(Example) Where X axis is reversed
If the direction (positive/negative) is reversed between traverse from Point 1 to Point 2 and traverse from Point 2 to Point 3, Point 2 is indicated as an axis reversal position.

</td><td></td></tr>
<tr><td>(3)</td><td>Display all data</td><td>When the [Display all data] checkbox is checked, both cutting feed and non-cutting feed (rapid traverse) are displayed.
When the [Display all data] checkbox is not checked, only the cutting feed is displayed.
(Default: OFF)</td><td>ON/OFF</td></tr>
<tr><td>(4)</td><td>Color Map</td><td>This is a parameter to visualize the feedrate, acceleration, position difference map, speed difference map and error amount between product CAD model and tool contact point from the position data, with color maps.
The color map is displayed using either single color or gradation, or using both settings.
This item is enabled only when [Feed Map], [Acceleration Map], [Posit. Diff Map], [Feed Diff Map] or [CAD model error] is selected for "Analysis Type".
The units of the minimum and maximum values are as follows.
<table>
<tr><th>Analysis type</th><th>Position data</th><th>Unit</th></tr>
<tr><td rowspan="2">Feed map</td><td>mm</td><td>mm/min</td></tr>
<tr><td>inch</td><td>inch/min</td></tr>
<tr><td rowspan="2">Acceleration map</td><td>mm</td><td>m/s^2</td></tr>
<tr><td>inch</td><td>inch/s^2</td></tr>
<tr><td rowspan="2">Posit. Diff Map</td><td>mm</td><td>mm</td></tr>
<tr><td>inch</td><td>inch</td></tr>
<tr><td rowspan="2">Feed Diff Map</td><td>mm</td><td>mm/min</td></tr>
<tr><td>inch</td><td>inch/min</td></tr>
<tr><td rowspan="2">CAD model error</td><td>mm</td><td>mm</td></tr>
<tr><td>inch</td><td>inch</td></tr>
</table></td><td>-</td></tr>
<tr><td></td><td>Range</td><td>Displays the minimum and maximum values of the analysis data (cutting feed part) shown in the color map.</td><td>-</td></tr>
<tr><td></td><td>Axis Selection</td><td>Select the axis for analysis by the analysis type of color map display.
When "ALL" is selected, XYZ axial composite components are analyzed.
<table>
<tr><th>Analysis Type</th><th>Axis</th><th>Default</th></tr>
<tr><td>Feed Map</td><td>X/Y/Z/ALL</td><td>ALL</td></tr>
<tr><td>Acceleration Map</td><td>X/Y/Z/ALL</td><td>ALL</td></tr>
<tr><td>Posit. Diff Map</td><td>X/Y/Z</td><td>Z</td></tr>
<tr><td>Feed Diff Map</td><td>X/Y/X/ALL</td><td>Z</td></tr>
<tr><td>CAD model error</td><td>ALL</td><td>ALL</td></tr>
</table></td><td>X/Y/Z/ALL</td></tr>
</table>

Item			Description	Range of values
	Color Type		Select the color map type to be used for gradation display by color map display. (Default: default)	default/Rainbow/Blue Red/Red White
	Single color (*1)		This is a parameter for single color display. Up to ten colors can be set. When this checkbox is checked and the color and the data range (minimum and maximum values) are specified, the path where the analysis data is in the specified range is displayed in the specified color. When this checkbox is not checked or when the maximum or minimum value is blank, single color display is not performed. Pressing the [DEL] key for the maximum and minimum values will make them blank. (Default: OFF, Default of [max] and [min]: blank)	ON/OFF
		max min	Specify the range of analysis data to be displayed in single color. (Default: blank)	0.000 to 99999.999 (This depends on the minimum and maximum values of [Range].)
	Gradation (*1)		This is a parameter for gradation display. When this checkbox is checked, the path where the analysis data is in the specified range is displayed in gradation. The gradation color can be specified by [Color Type]. When this checkbox is not checked, analysis data other than that in the range specified by [Single color] is not displayed.	ON/OFF
		max min	Specify the range of analysis data to be displayed in gradation. (Default: blank)	0.000 to 99999.999 (This depends on the minimum and maximum values of [Range].)
		Hide data out range	When this checkbox is checked, the path where the analysis data is outside the specified range is not displayed. When this checkbox is not checked, the path where the analysis data is outside the specified range is displayed in the color that corresponds to the maximum value or minimum value of gradation. (Default: OFF)	ON/OFF
(5)	CAD model error		Perform the product CAD model display-related settings.	-
	Curved surface display		Switch whether the product CAD model is displayed by a curved surface or by a wire frame. When this checkbox is checked, the read product CAD model is displayed by a curved surface. When the checkbox is not checked, it is displayed by a wire frame. (Default: OFF)  <Wire frame display>  <Curved surface display>	ON/OFF

Item			Description	Range of values
	Offset amount		Specify the product CAD model position so that the product CAD model is on the machining program path.	-
		Travel I	Specify the offset amount to move and display the product CAD model in each axis direction from the workpiece reference point position. The workpiece reference point position is the coordinate set in [Work Ref Pnt Sft I] to [Work Ref Pnt Sft K] of the [Machining simulation settings] sub-display. Specify the offset amount from this coordinate. When another product CAD model is selected, the value is initialized. (Default: 0.000 mm)	-99999.999 to 99999.999 mm (This depends on the workpiece reference point position.)
		Travel J		
		Travel K		
		Setting button	Applies the set travel amount to the displayed CAD model.	-
(6)	Axis Reversal		Specify an analysis axis when [Axis Reversal] is selected for "Analysis Type". The reversal state of the specified axis direction can be analyzed.	X/Y/Z
	Axis Selection (X/Y/Z)		Select an axis (X, Y, or Z) for which direction reversal is analyzed.	
(7)	Select the tool to be drawn		Select the tools that display machining paths.	-
		Select all	When this checkbox is checked, all tools displayed in [Tool List] are selected. When the checkbox is not checked, all tools displayed in [Tool List] are deselected. When at least one of the tools displayed in [Tool List] is deselected, this item becomes undetermined.	ON/OFF/Undetermined
		Tool List	The names of all the tools defined in the Tool Settings window are displayed as a list in the following format. "Tool No. + Tool Type + Diameter (integer part + one digit after the decimal point)". Check the checkboxes of the tools for which machining paths are to be displayed. (Default: ON) When a checkbox is unchecked, [Select all] becomes undetermined. The machining paths are reflected on the analysis simulation window when one of the following operations is conducted. ・ Executing analysis simulation ・ Clicking the [Update] button ・ Saving a project then reopening it ・ Transitioning from another window Tool selection can be changed while simulation is being executed. The path of a tool not in the Tool List cannot be hidden.	ON/OFF
(8)	Show Scale		Configure the settings related to scale display.	-
		X axis	When this checkbox is checked, the X axis of the scale is displayed. When the checkbox is not checked, the X axis of the scale is hidden. (Default: ON)	ON/OFF
		Y axis	When this checkbox is checked, the Y axis of the scale is displayed. When the checkbox is not checked, the Y axis of the scale is hidden. (Default: ON)	ON/OFF

Item		Description	Range of values
	Z axis	When this checkbox is checked, the Z axis of the scale is displayed. When the checkbox is not checked, the Z axis of the scale is hidden. (Default: ON)	ON/OFF
(9) Update button		Click the [Update] button to start analysis in accordance with the "analysis simulation settings" you set. (*2) During the analysis, the progress window appears. If you press [Cancel] in the progress window, analysis is stopped. When simulation is in progress, this button is not selectable.	-
(10) Message area		If an out-of-range value is set in min or max for [Single color] or [Gradation], the message "This value cannot be set." appears. The message area is two lines high.	-

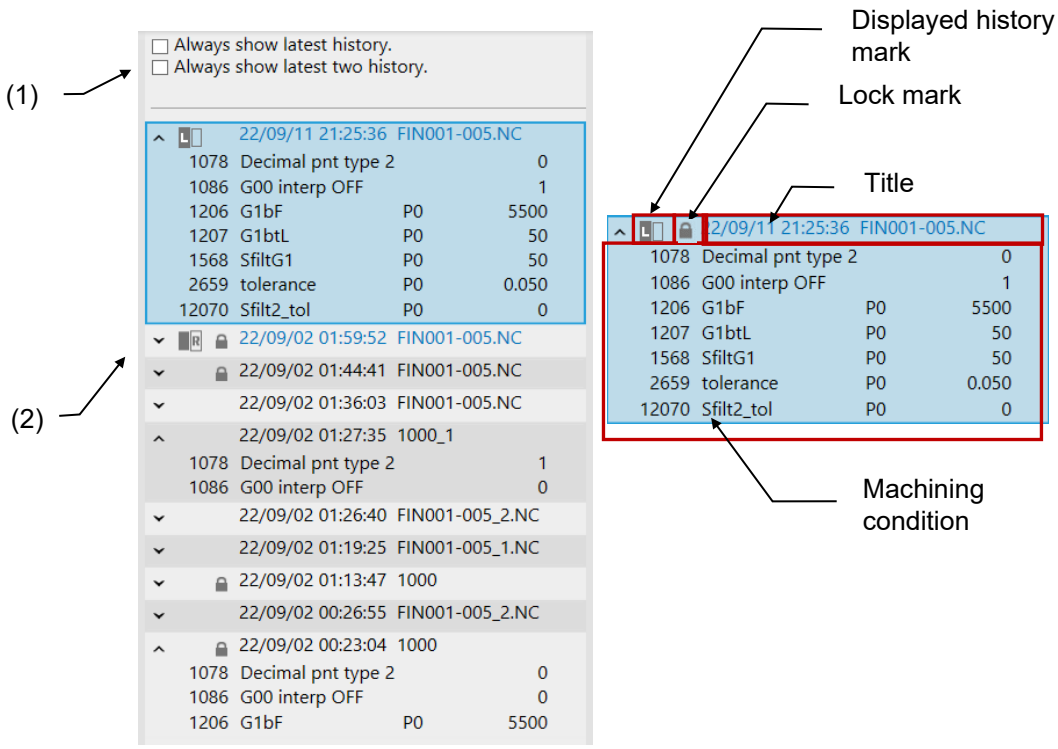
(*1) When the [Gradation] checkbox is unchecked while the [Single color] checkbox is unchecked, the [Single color] checkbox is checked automatically. Similarly, when the [Single color] checkbox is unchecked while the [Gradation] checkbox is unchecked, the [Gradation] checkbox is checked automatically.

(*2) If the Simulation priority setting is different between when simulation is started and when [Update] is clicked, an error message appears in the sub-message display area.

(Note) The settings of each item cannot be changed and applied while simulation is running.

4.3.2.6 Simulation history

"Simulation history" provides a list of the execution history of machining simulation and analysis simulation.
This is used on the simulation difference analysis window.
For details, see "9 Simulation difference analysis".



Item		Description	Range of values
(1)	Difference analysis settings	Specify the behavior of simulation difference analysis window.	—
	Always show latest history.	Displays the latest simulation result in the simulation area (left) and the last displayed simulation result in the simulation area (right) when the [Analyze Difference] tab is selected. (If the last displayed simulation result is deleted, the second latest simulation result is displayed.) If this item is checked while [Always show latest two history.] is checked, [Always show latest two history.] is unchecked. (Default: OFF)	ON/OFF
	Always show latest two history.	Displays the latest simulation result in the simulation area (left) and the second latest result in the simulation area (right) when the [Analyze Difference] tab is selected. If no simulation history exists, the result is displayed on one screen. If this item is checked while [Always show latest history.] is checked, [Always show latest history.] is unchecked. (Default: ON)	ON/OFF

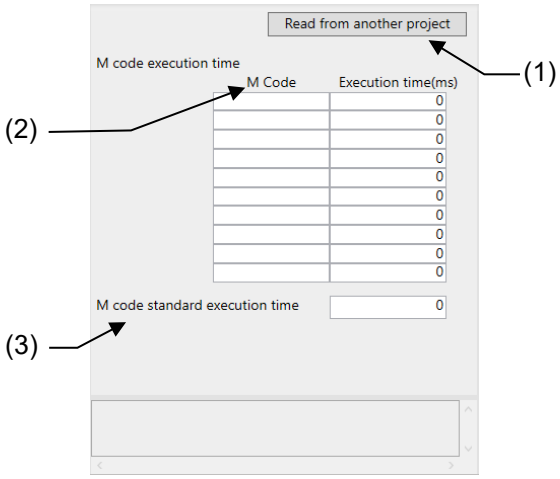
Item		Description	Range of values
(2)	Simulation history list	Displays up to 10 histories of the simulation results in the order of simulation. The latest simulation result is displayed at the top. (Note) When the number of the saved histories exceeds the maximum number, the oldest history is automatically deleted.	—
	Displayed history mark	Displays the mark for the simulation history displayed in the simulation area. The marks to be displayed are as follows. ・ Simulation history in the simulation area (left): ・ Simulation history in the simulation area (right): ・ Simulation history displayed on one screen:	—
	Lock mark	Displays for the locked history. The history with this mark is not automatically deleted.	—
	Title	Displays the time when simulation is started and the program name. "YY/MM/DD HH:MM:SS EIA NAME" The time is the local time of the PC in use. If the complete simulation history title cannot be displayed, "…" to show omission is displayed at the end. When the mouse pointer is positioned, the full path of the machining program name used when simulation is run is displayed in a tooltip.	—
	Machining condition	This is displayed when there is a difference as a result of comparison between histories of the simulated machining conditions. Parameters which have been changed since the new project was created and machining conditions of simulation history executed in the same machining program are compared and displayed. For parameters like the tolerance control parameter "1206 G1bF" which are set for each machining condition, the target machining condition like "1206 G1bF P1-Q1" is also displayed. By clicking and at the left of the displayed history mark or pressing the [Enter] key while the history is selected, the machining condition is switched to be shown or hidden. If two or more histories are selected, the show and hide is switched according to the top among the selected histories.	—

(*) If neither [Always show latest two history.] nor [Always show latest history.] is checked, the last displayed contents are displayed.

4.3.2.7 Machining time prediction setting

You can set execution time of M code command. The execution time of the set M code command is added to the machining time prediction calculated by simulation, and you can obtain more accurate machining time prediction.

For this data, the values of the parameters #42802 to #42820 and #42952 included in the machine tool's backup data are read. When values are set to these parameters by the machine tool builder, the values are set at the time when a new project is created.



Item		Description	Range of values
(1)	Read from another project	Reads the machining time prediction setting saved in another project.	-
(2)	M code execution time	Set M code number and execution time of the M code command to be added as machining time prediction. (Maximum: 10 codes)	-
	M Code	Set the M code number. When the same M code is set to two or more lines, the first setting time is added. A blank line is disabled. To disable a line, make the line blank with the [DEL] key. M00, M01, M02, M30, M96, M97, M98, M99, or M198 cannot be set.	0 to 99999999
	Execution time(ms)	Set the execution time of the M code command. The set execution time is added to the machining time prediction.	0 to 3600000 (ms)
(3)	M code standard execution time	Set the average execution time when an M code command other than the M code set to [M code execution time] is executed. M00, M01, M02, M30, M96, M97, M98, M99, and M198 are miscellaneous functions to be used for a specific purpose; therefore, they are not added to the machining time prediction.	0 to 3600000 (ms)

(Note 1) When a block includes two or more M code commands, only the longest time is added.

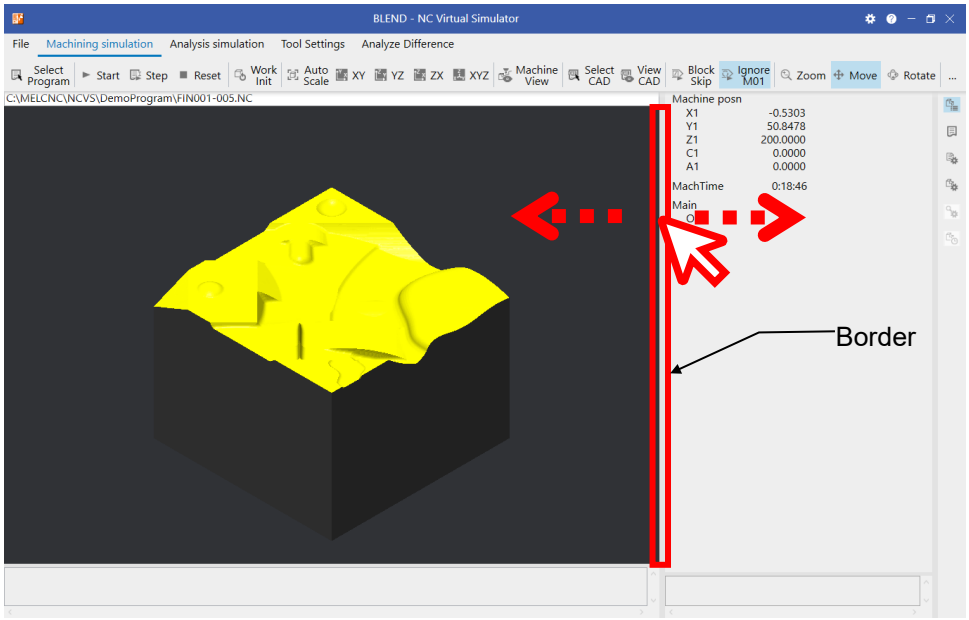
(Note 2) If an out-of-range value is input, the message "Out of range." appears.

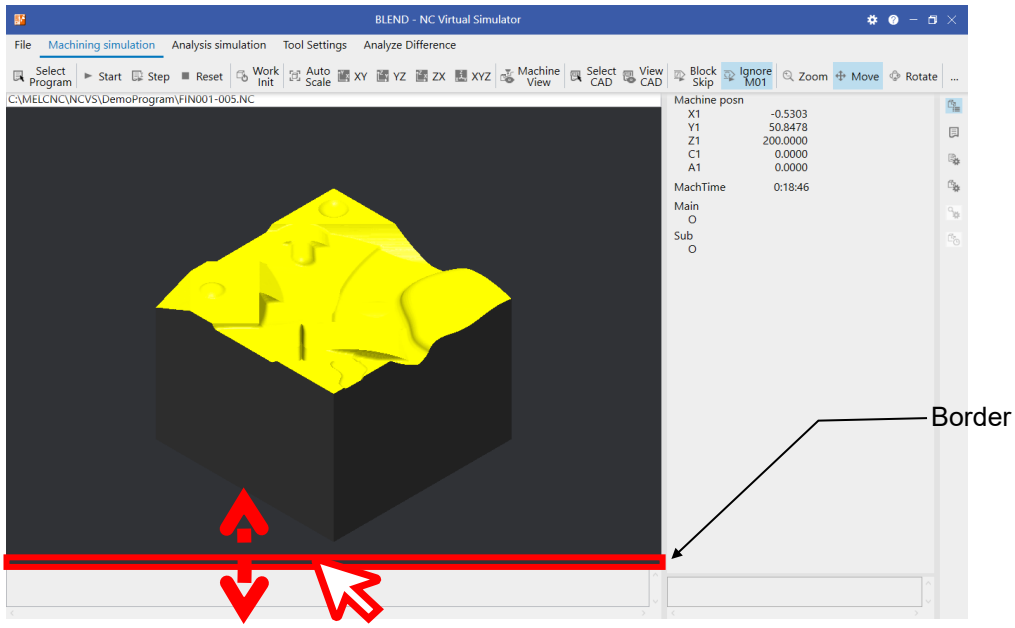
(Note 3) The codes used for the M code macros are not added to the machining time.

4.4 Display area

4.4.1 Resizing display area

You can change the width of the sub-display area and the height of the message display area by dragging and dropping the border between the display areas.





4.5 Shortcut keys

The following lists the shortcut keys that can be used in NC Virtual Simulator.

Shortcut key	Function	Description
Ctrl + C	Copy	Copy one line of tool data. For detailed information, see the following section. "8.3.5 Copy and paste one line of tool data"
Ctrl + F	Tool data search	Search for a tool number. For detailed information, see the following section. "8.3.2 Search tool data"
Ctrl + O	Open	Open a project. For detailed information, see the following sections. "5.3.1 Creating a new project" "5.3.2 Opening a project"
Ctrl + V	Paste	Paste one line of tool data. For detailed information, see the following section. "8.3.5 Copy and paste one line of tool data"
Ctrl + S	Save	Save a project. For detailed information, see the following section. "5.3.3 Saving a project"
Alt + F4	Exit	Exit NC Virtual Simulator. For detailed information, see the following section. "5.2 Exiting NC Virtual Simulator"

(*) The shortcuts "Ctrl + C", "Ctrl + F", and "Ctrl + V" are only available when the [Tool Settings] tab is selected.

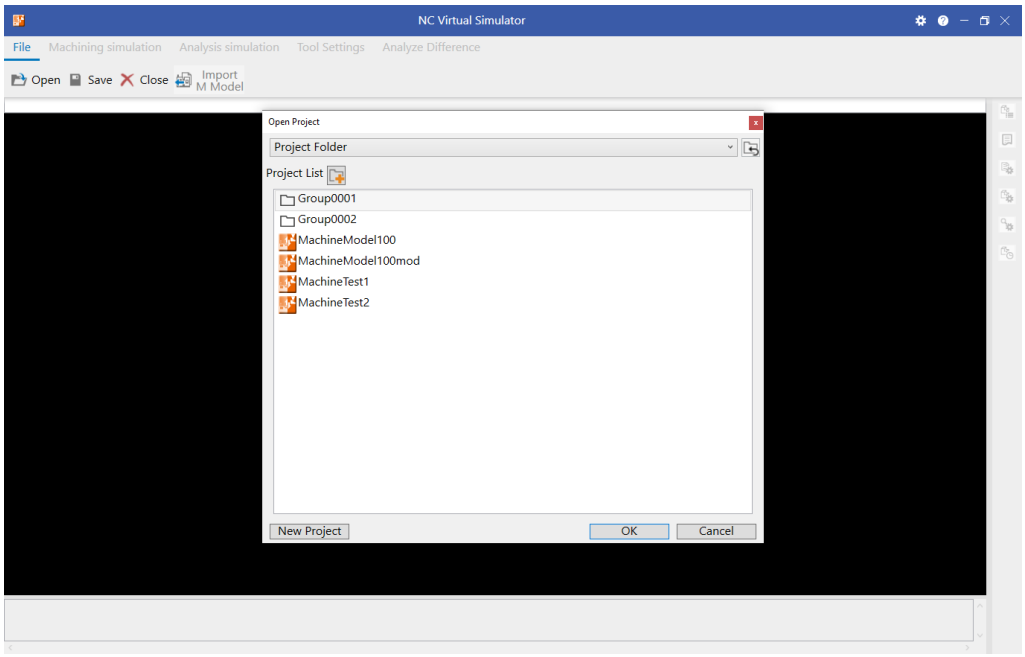
(*) When you enter "Ctrl + O" or "Ctrl + S", the [File] tab is selected. The simulation (if running) is pause.

5. Using NC Virtual Simulator

5.1 Starting NC Virtual Simulator

(1) Select [Start] menu > [NC Virtual Simulator].
(*) This might be [Start] menu > [MELSOFT] > [NC Virtual Simulator].

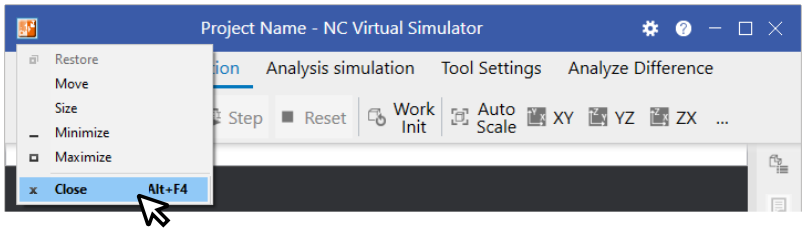
(2) NC Virtual Simulator starts, and the [Open Project] dialog box appears.
To create a new project, go to "5.3.1 Creating a new project". To open an existing project, go to "5.3.2 Opening a project".



(*) The default cursor position in the Open Project dialog box is the project that was last displayed.
(*) If you select [Cancel] in the dialog box, only the main window is displayed.

5.2 Exiting NC Virtual Simulator

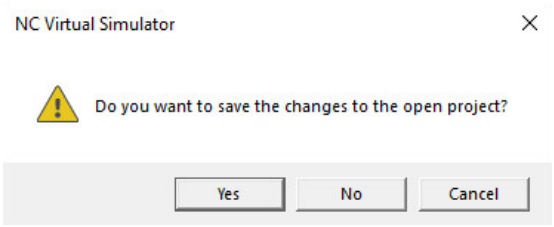
To exit NC Virtual Simulator, click the icon in the title bar, and click [Close] in the menu displayed.



(*) You can also exit NC Virtual Simulator by the following operations.

- Click the [×] button in the title bar.
- Press and hold the [ALT] key and press the [F4] key.

(*) When changes are made to the project, you see a confirmation message asking if you want to save the project. To save the project before exiting NC Virtual Simulator, click [Yes]. To exit NC Virtual Simulator without saving the project, click [No]. To stop exiting NC Virtual Simulator, click [Cancel].



5.3 Creating a project

5.3.1 Creating a new project

There are three ways to create a project.

- Use the machine tool's backup data. (5.3.1.1)
- Use the system data of the network-connected machine tool. (5.3.1.2)
- Use the sample project. (5.3.1.3)

(*) The following rules apply to project names and folder names.

- Up to 80 characters (half-width) can be set. (A full-width character is counted as two half-width characters.)
- Both half- and full-width characters can be used.
- Project names are not case-sensitive.
- A project name cannot contain the following half-width characters.
  \ / : * ? < > | " (the same characters as those not allowed for file names)
- The names of the projects and folders that have already been created cannot be specified.
- A project name cannot contain only spaces.
- A project name cannot begin or end with a space or a half-width period (.).
- CON, PRN, AUX, CLOCK$, NUL, COM0 to COM9, LPT0 to LPT9 (all half-width characters) cannot be used.

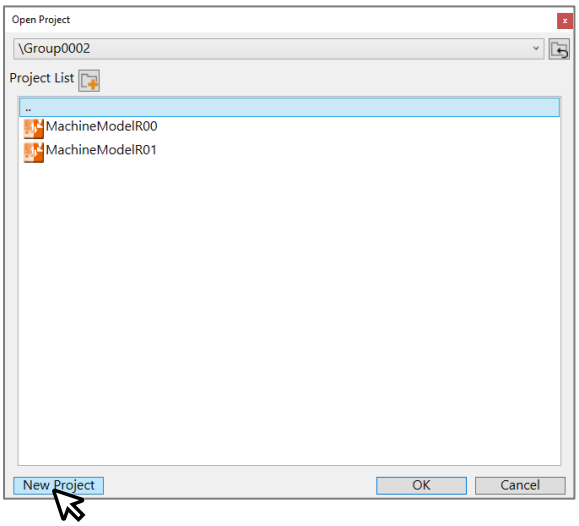
5.3.1.1 Creating a project using the machine tool's backup data

The following describes how to create a project using the machine tool's backup data.

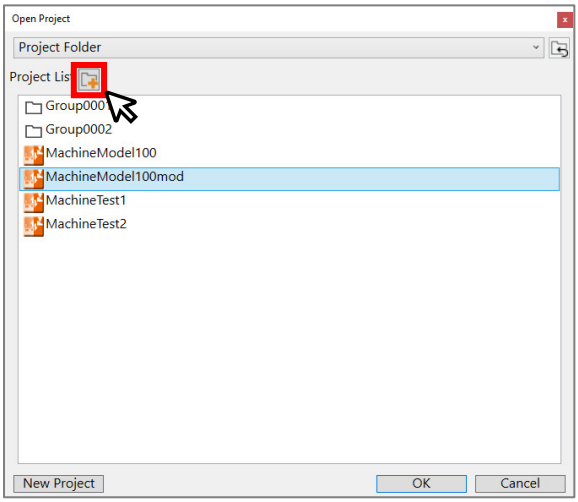
(*) Obtain the backup data on the [Maint] - [All backup] screen of the machine tool in advance.

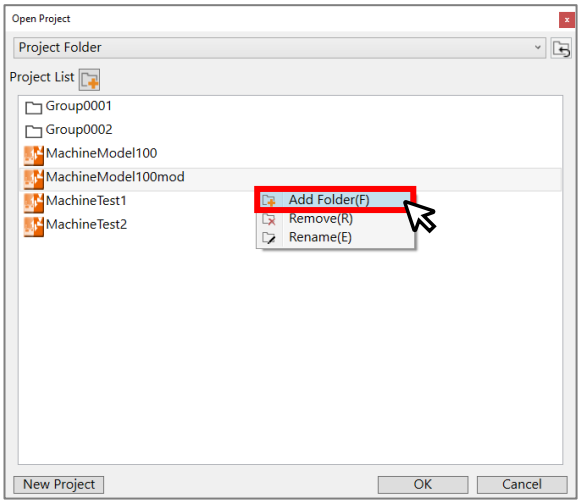
(1) Click the [File] tab > [Open] command.

(2) The Open Project dialog box appears.
Navigate to the folder where the project will be created, and click the [New Project] button.

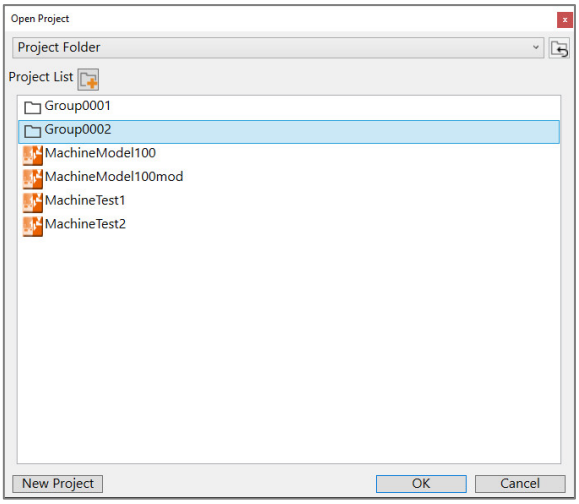


(*) When creating a folder where a new project will be created, click or [Add Folder] on the right-click context menu in Project List.

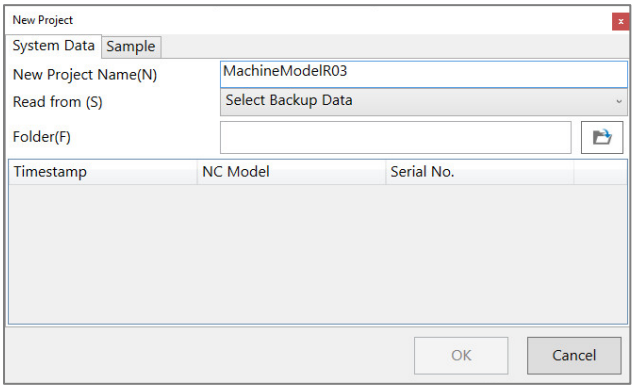




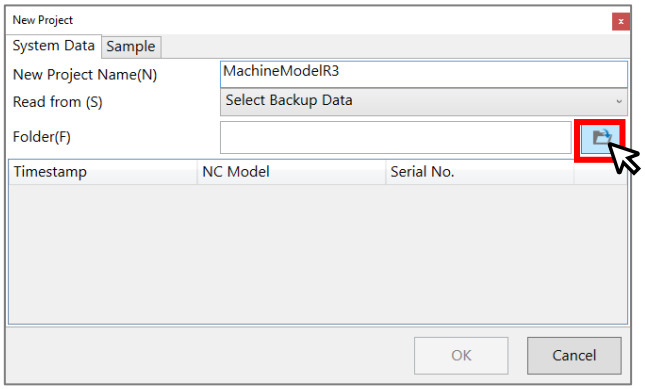
(*) The following describes how to navigate to the folder when a project is created in another folder.
Double-click on in Project List to navigate to the folder.
Click or [..] in Project List to navigate to the folder that the current folder is in.
If the current folder is the project storage folder, [..] is not displayed.



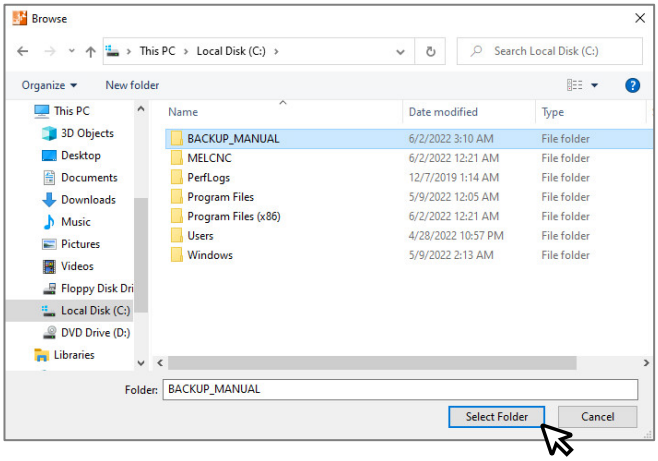
(3) The "New Project" dialog box appears.
Enter a name for a new project in "New Project Name".



(4) Click to specify the folder where the machine tool's backup data is saved.

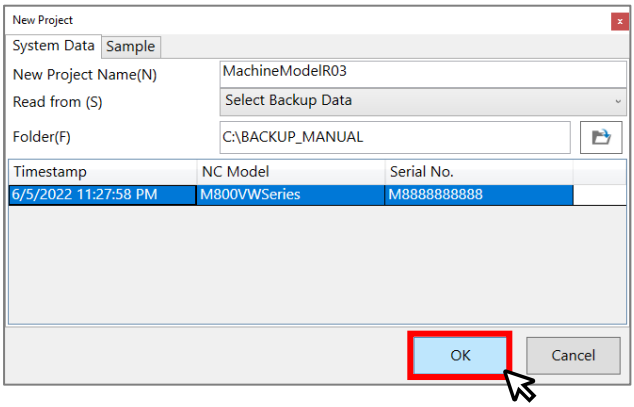


(5) The [Browse] dialog box appears.
Specify the folder where the machine tool's backup data is stored, and click the [Select Folder] button.
(*) The default folder is "Documents".



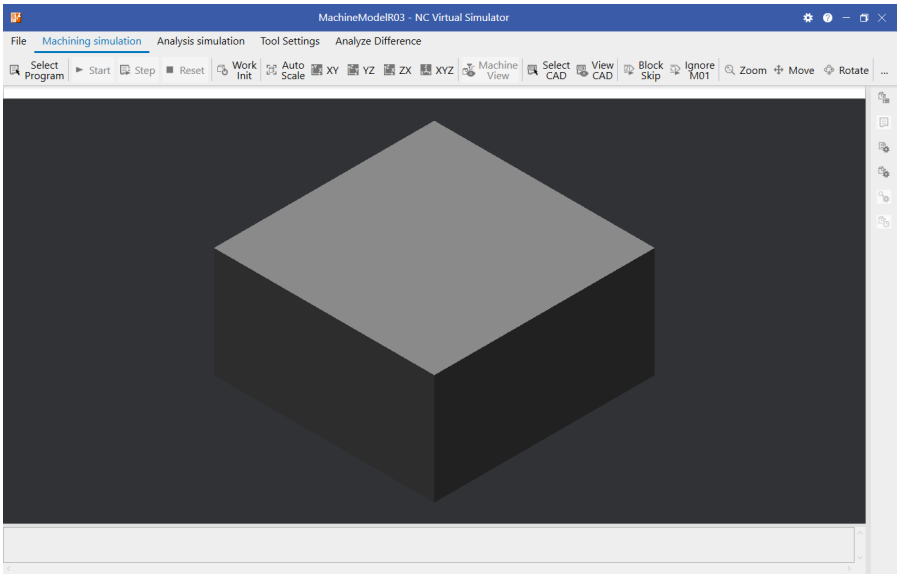
(*) You can specify a folder that contains the backup data of a single machine tool or the backup data (folder) of multiple machine tools.

(6) The content of the machine tool's backup data is displayed.
After confirming the NC model, click the [OK] button.



Item	Description
Timestamp	Displays the date and time that the data was backed up. This date is the timestamp of the system data.
NC Model	Displays the series of the NC model of the backup data.
Serial No.	Displays the serial number of the NC of the backup data.

(7) A new project is created, and the main window opens.

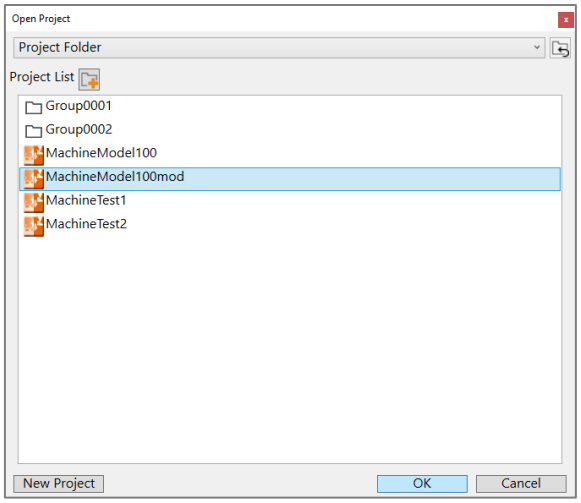


5.3.1.2 Creating a project using the network-connected machine tool's data

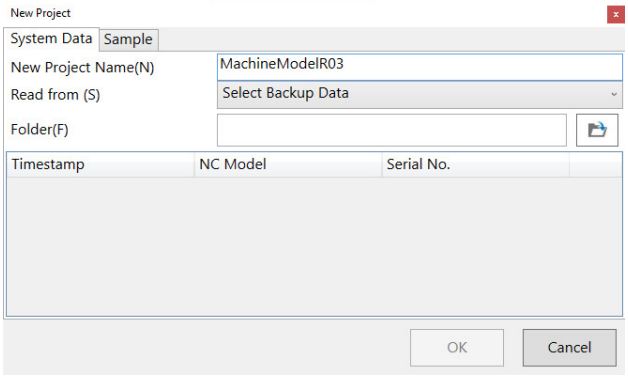
The following describes how to create a project using the system data of the network-connected machine tool.

(1) Click the [File] tab > [Open] command.

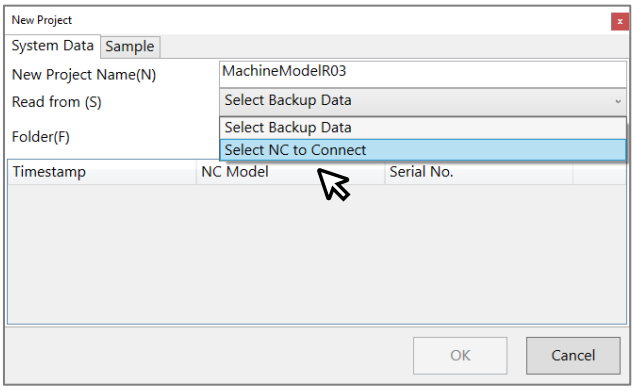
(2) The Open Project dialog box appears. Navigate to the folder where the project will be created, and click the [New Project] button.



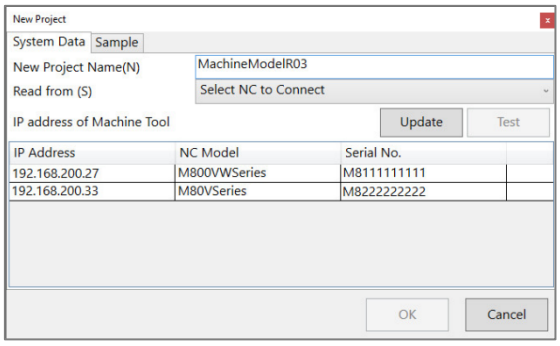
(3) The "New Project" dialog box appears.
For "New Project Name", enter the name of the new project to create.



(4) For "Read from", choose [Select NC to Connect] from the pull-down menu.



(5) Click [OK]. The IP addresses of the network-connected machine tools are displayed.



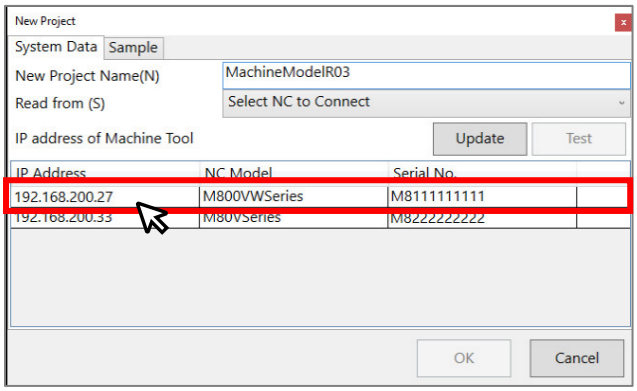
Item	Description
IP Address	Displays the IP addresses of the NCs that are selectable.
NC Model	Displays the series of the NC models.
Serial No.	Displays the serial numbers of the NCs.

(*) If the machine tool to select is not displayed in the list, verify the following and click the [Update] button.

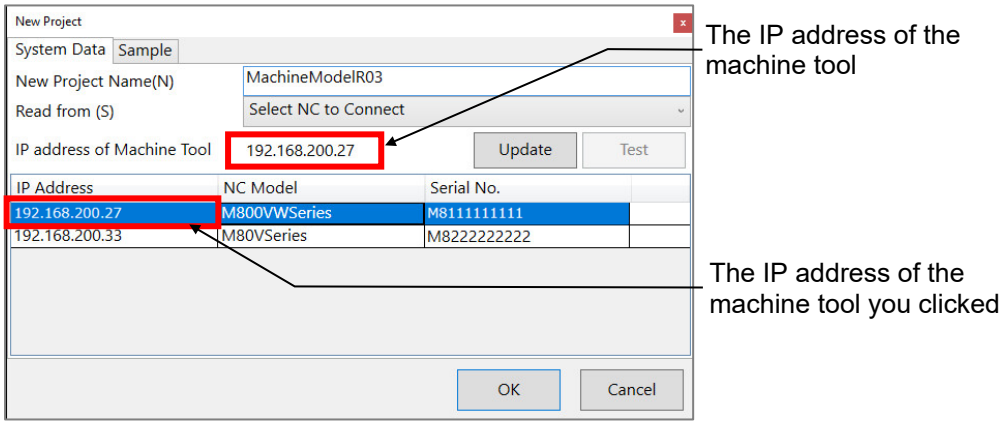
- The machine tool to connect is powered on.
- The network address of the machine tool to connect is the same as that of the PC.

(*) The IP addresses of unsupported models are not displayed.

(6) Verify the NC model and the other information and click the IP address of the machine tool to connect.

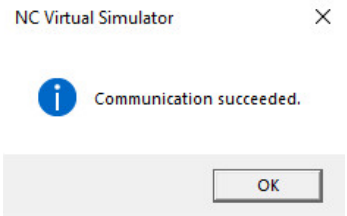


(7) The IP address of the machine tool you clicked appears in the "IP address of Machine Tool" field.

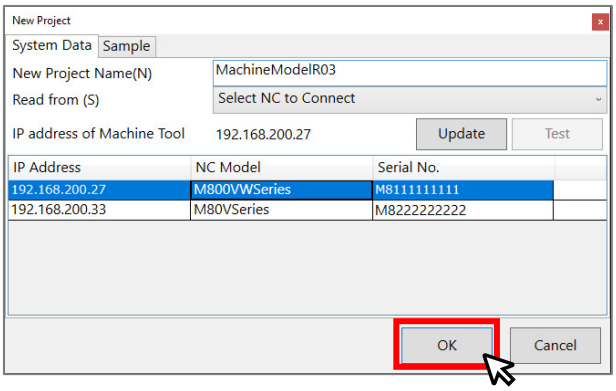


(*) To check the connectivity with the machine tool, click the [Test] button.
NC Virtual Simulator starts communication test with the NC whose IP address is shown in the "IP address of Machine Tool" field. During the communication test, the display of the [Test] button switches to "Testing... ", and the dialog box is inoperable.
To cancel the communication test, click the [Test] button while it displays "Testing...".

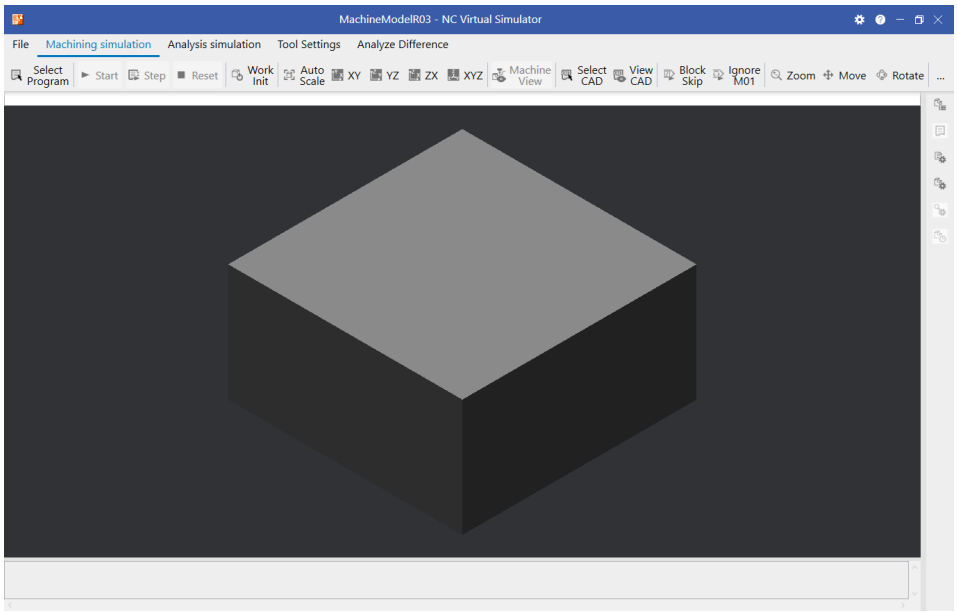
When the communication test is successful, the following message box appears.



(8) Click the [OK] button.



(9) A new project is created, and the main window opens.

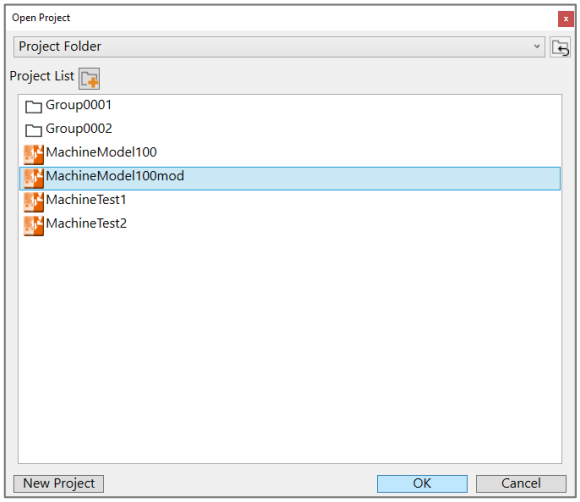


5.3.1.3 Creating a project using the sample project

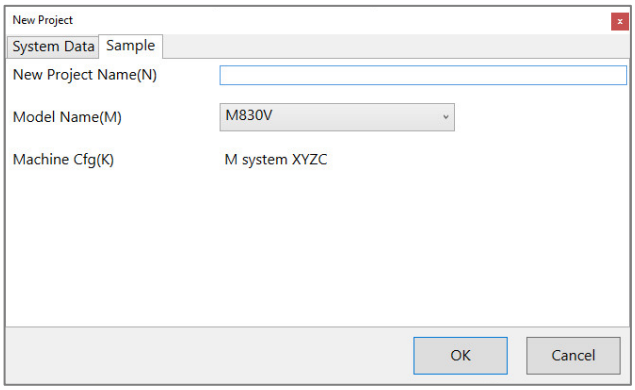
The following describes how to create a project using the sample project.

(1) Click the [File] tab > [Open] command.

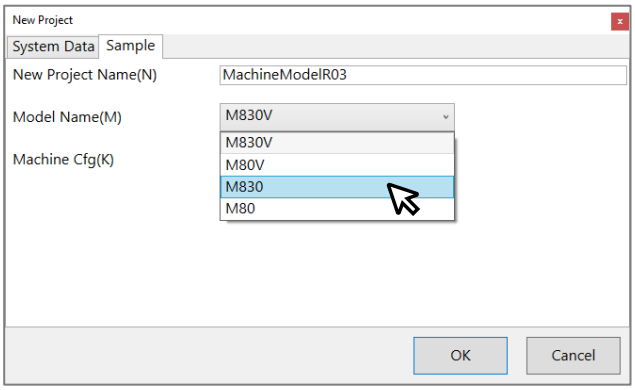
(2) The "Open Project" dialog box appears.
Move to the folder where a project will be created, and click the [New Project] button.



(3) The "New Project" dialog box appears.
Click the [Sample] tab.

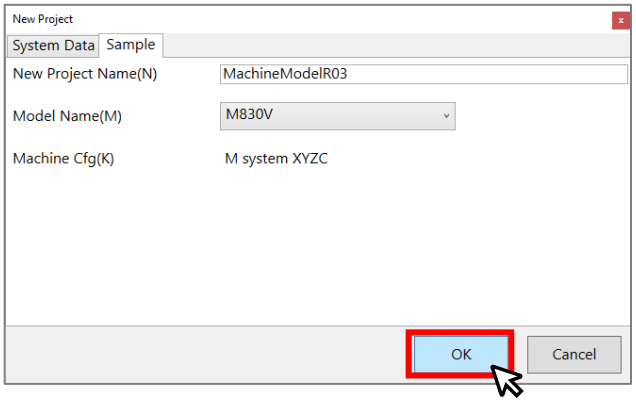


(4) Enter a new project name, and select the model name.

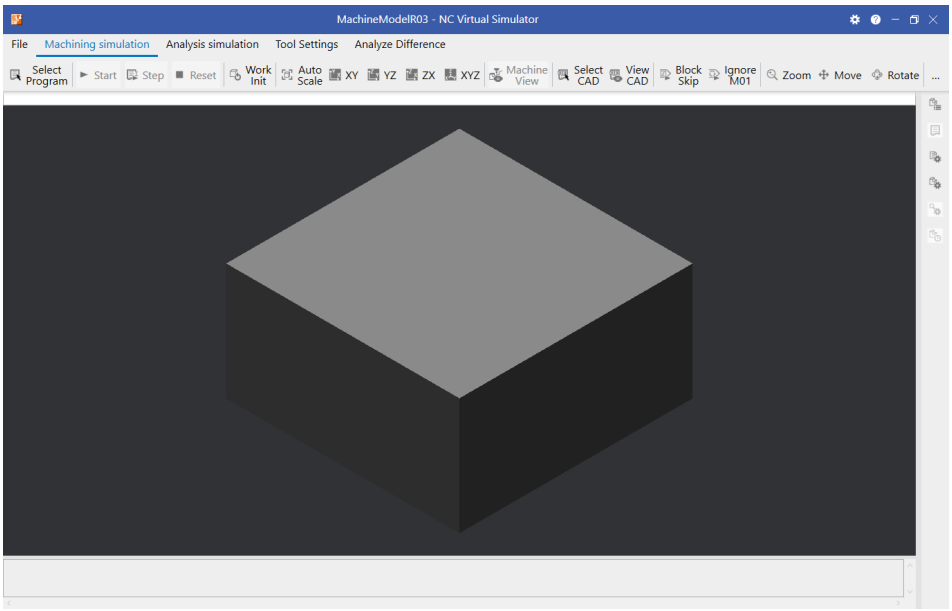


Item	Description
Model Name	Select the model name. (M830V/M80V/M830/M80)
Machine Cfg	Displays the machine configuration of the model selected in Model Name.

(5) Click the [OK] button.



(6) A new project is created, and the main window opens.

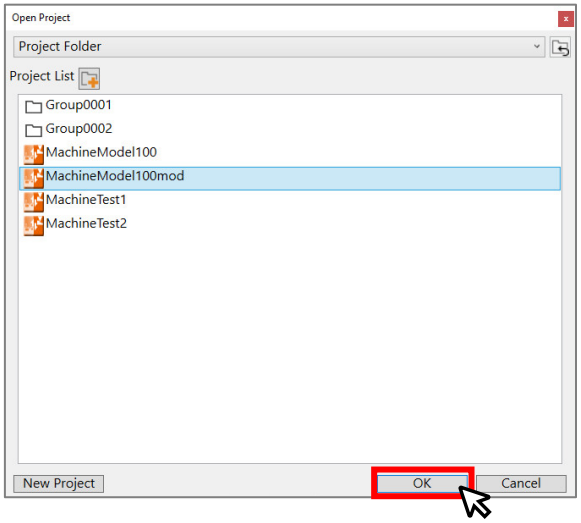


5.3.2 Opening a project

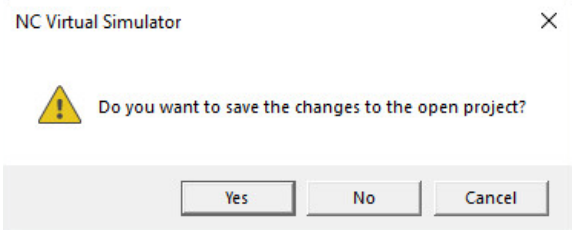
The following describes how to open a created project.

(1) Click the [File] tab > [Open] command.

(2) The "Open Project" dialog box appears.
Select the project to open, and click the [OK] button.

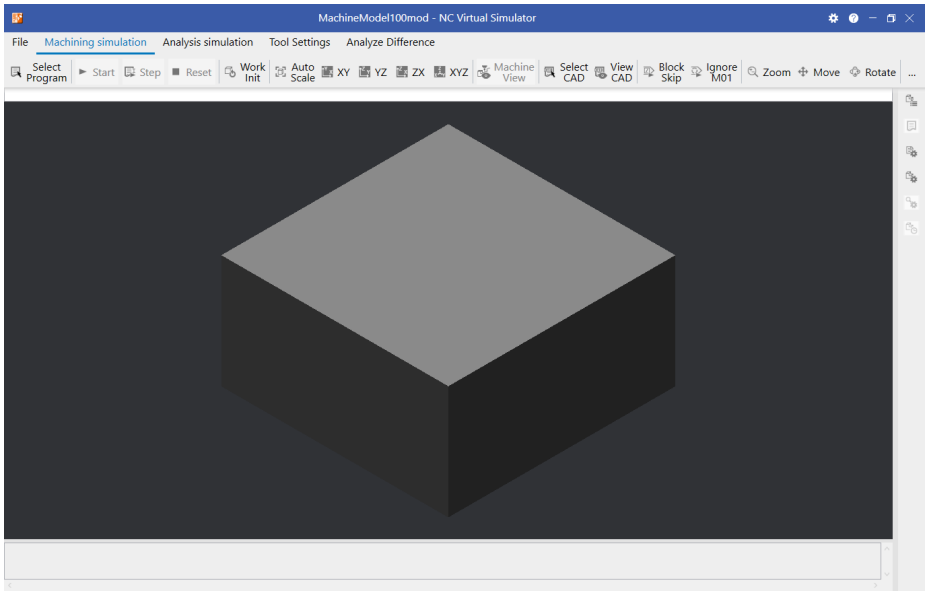


(*) If changes have been made to the currently open project, you see a confirmation message asking if you want to save the project. Select whether to save the project before opening the selected project or to open the selected project without saving the project.



(*) If the currently open project is selected, the dialog box closes without any action being taken.

(3) The selected project is opened, and the main window is displayed.
When an existing project is opened, the last simulated result is displayed.



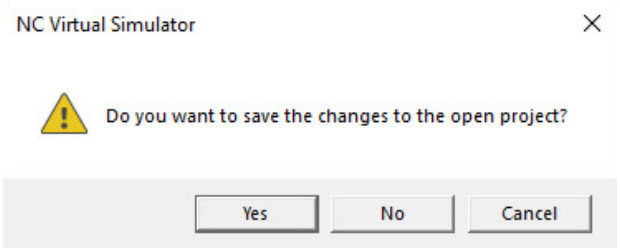
5.3.3 Saving a project

Click the [File] tab > [Save] command to save the currently open project.

5.3.4 Closing a project

Click the [File] tab > [Close] command to close the currently open project.

(*) If changes have been made to the currently open project, you see a confirmation message asking if you want to save the project. Click [Yes] to save the project before closing it. Click [No] to close the project without saving it. Click [Cancel] to cancel closing the project.

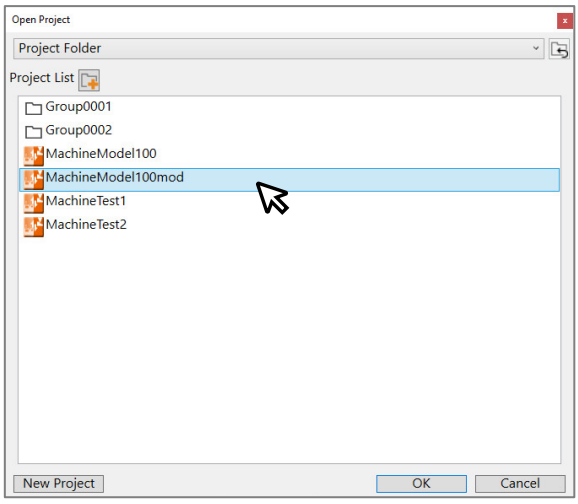


5.3.5 Renaming a project

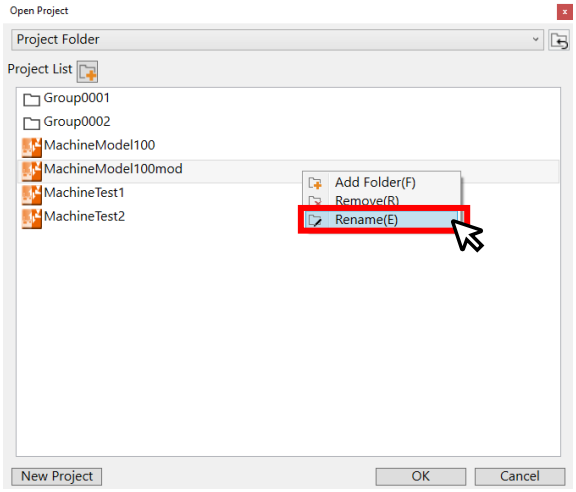
The following describes how to rename a project or folder.

(1) Click the [File] tab > [Open] command.

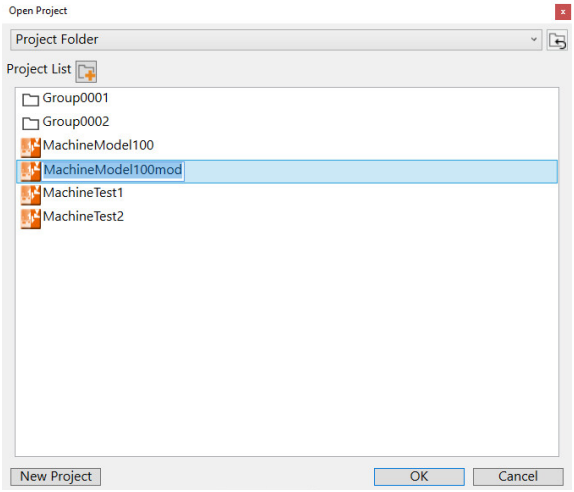
(2) Select the project or folder to rename.



(3) Right-click to open the context menu and select [Rename], or press the [F2] key.



(4) Enter a new name, and press the [Enter] key.



(*) To cancel rename, press the [ESC] key.

(*) There are rules that apply to a new project name and folder name to set.
See "5.3.1 Creating a new project".

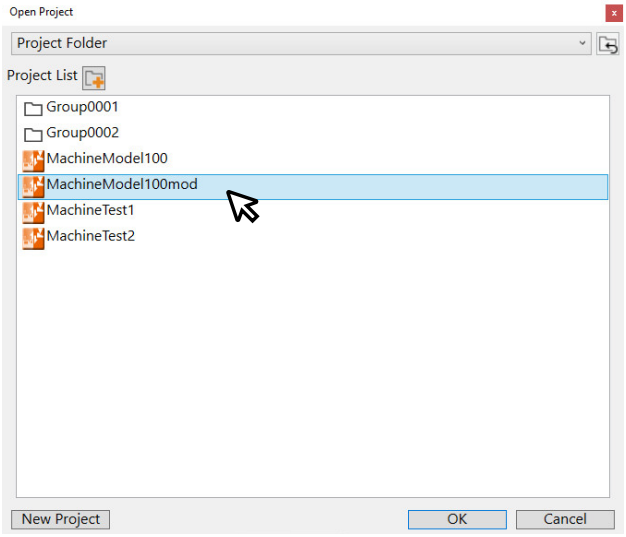
5.3.6 Deleting a project

The following describes how to delete a project.

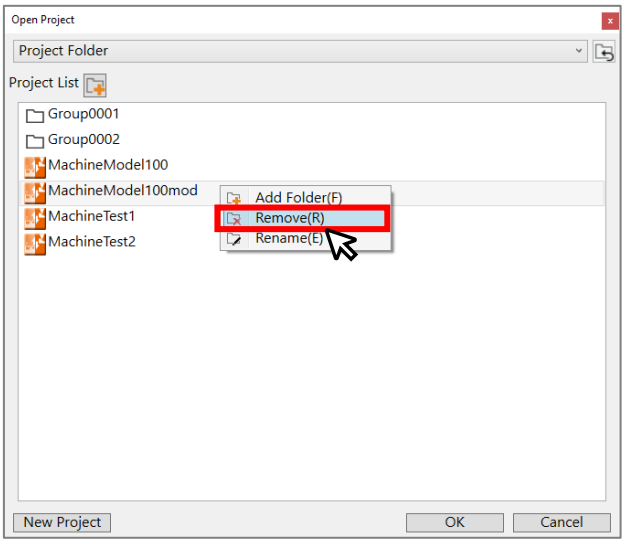
(1) Click the [File] tab > [Open] command.

(2) Select the project to delete.

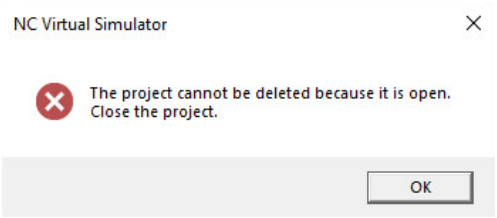
(*) You can also select a folder. In this case, all the files in the folder are deleted.



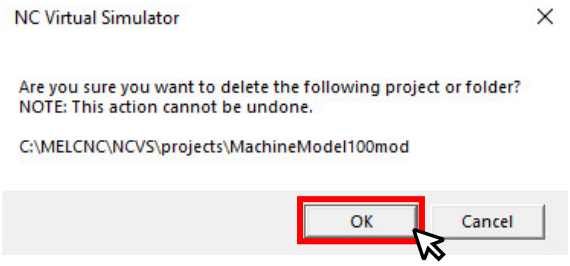
(3) Right-click to open the context menu and select [Remove], or press the [Del] key.



(*) If the currently open project is, or is contained in, the selected project or folder, the following error message appears.

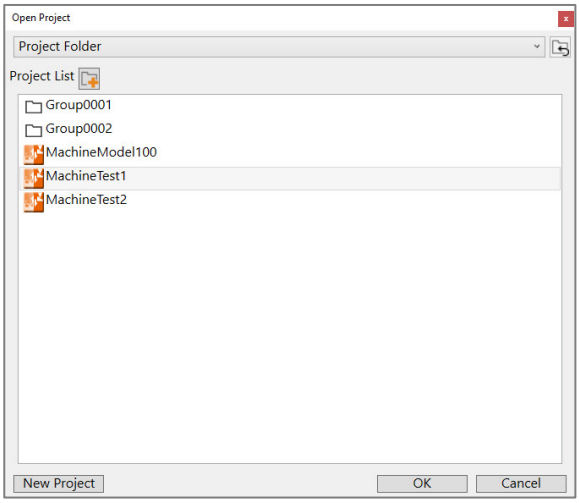


(4) The following confirmation message appears.
To delete the project or folder, click [OK].



(*) Deleting a project or folder cannot be undone.

(5) Project List is updated.



5.4 Importing machine model to NC Virtual Simulator

Interference between machine and workpiece can be checked by importing the machine model. To check the interference, enable [Machine interference check] of the sub-display "Machining simulation settings".

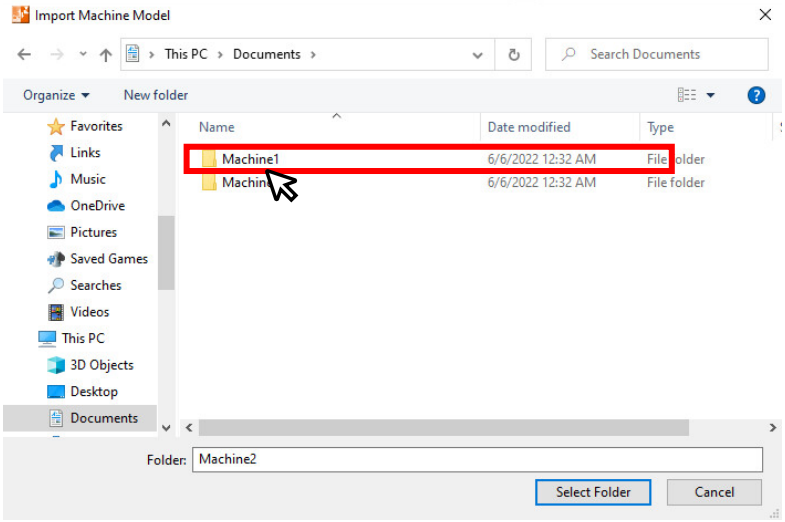
Obtain the machine model data (*1) (*2) from the machine tool builder.

(*1) The machine model data consists of a machine configuration file (*.XML) and a machine parts file (*.STL).
(*2) The machine model file name must not exceed 20 characters (half-width, excluding the three characters for the extension) and the name of the folder where the machine model is stored must not exceed 12 characters (half-width).

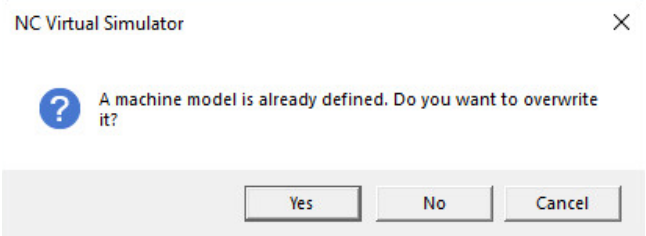
The following describes how to import the machine model.

(1) Click the [File] tab > [Import M Model] command.

(2) The "Import Machine Model" dialog box appears.
Choose the folder where the machine configuration file is stored and click [Select Folder].



(3) The machine model is imported in the project.
(*) If there is a machine model that has already been imported, the overwrite confirmation dialog appears. Click [Yes] to overwrite the machine model. Click [No] to display the Import Machine Model dialog box and choose a machine model again. Click [Cancel] to cancel the import.



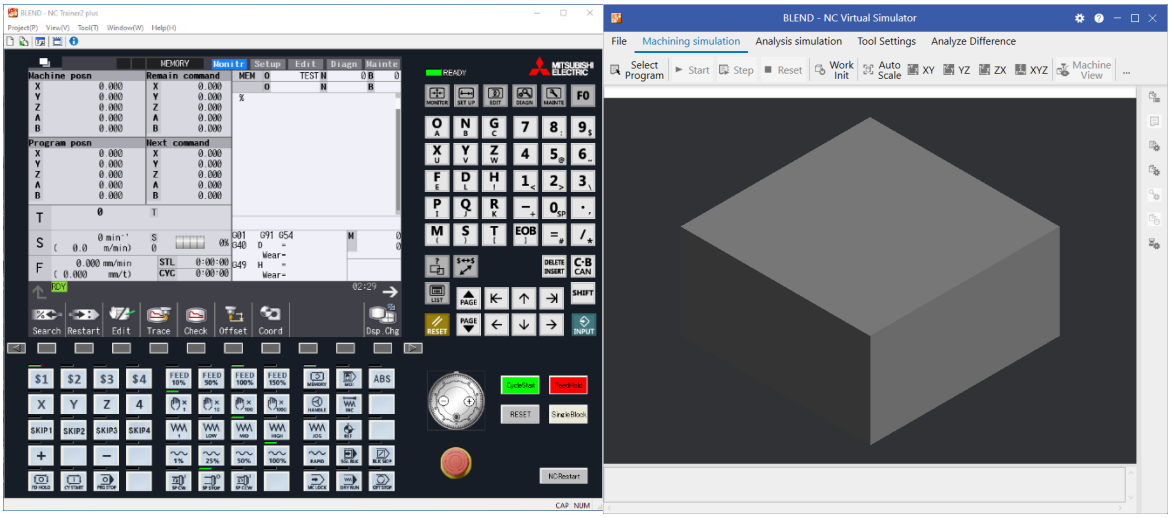
(*) If the machine model is imported during the simulation, the simulation is reset.

5.5 NC Trainer2 plus link

NC Trainer2 plus is an application to allow you to check the operation of the user-developed PLC programs and the custom screen without using an NC control unit and a dedicated display unit. By linking with NC Trainer2 plus, you can design machines, debug the developed PLC programs, and check machine operation and interference by a machine model.

To link NC Virtual Simulator with NC Trainer2 plus, start NC Trainer2 plus to perform the operation.

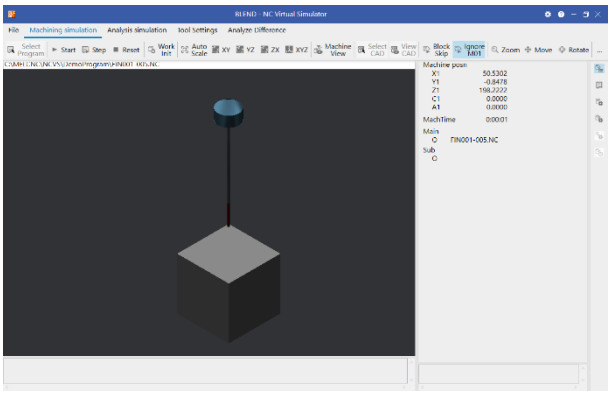
For detailed steps, refer to the NC Trainer2 plus instruction manual "IB-1501318".



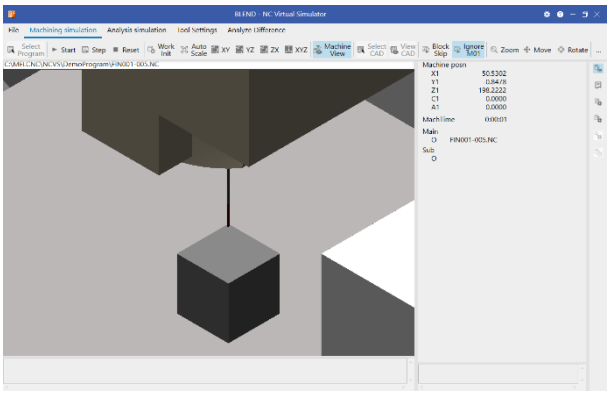
6. Machining simulation

The machining simulation function allows you to check the shape of the workpiece, the movement of the tool, and interference in the cutting process. You can toggle between workpiece simulation and machine simulation in this function.

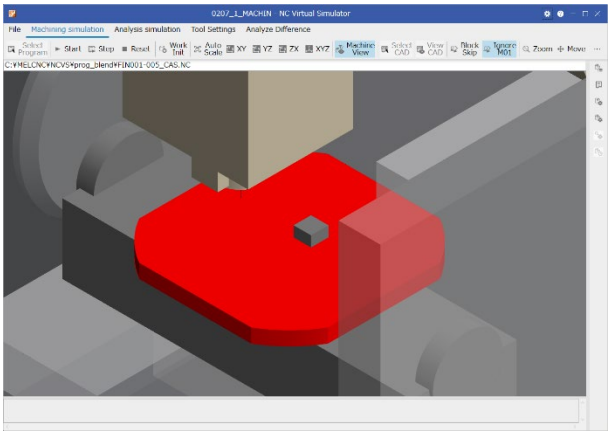
(*) To perform machining simulation and interference check, you need to import the machine model. (Refer to "5.4 Importing machine model to NC Virtual Simulator".)



Workpiece simulation



Machine simulation

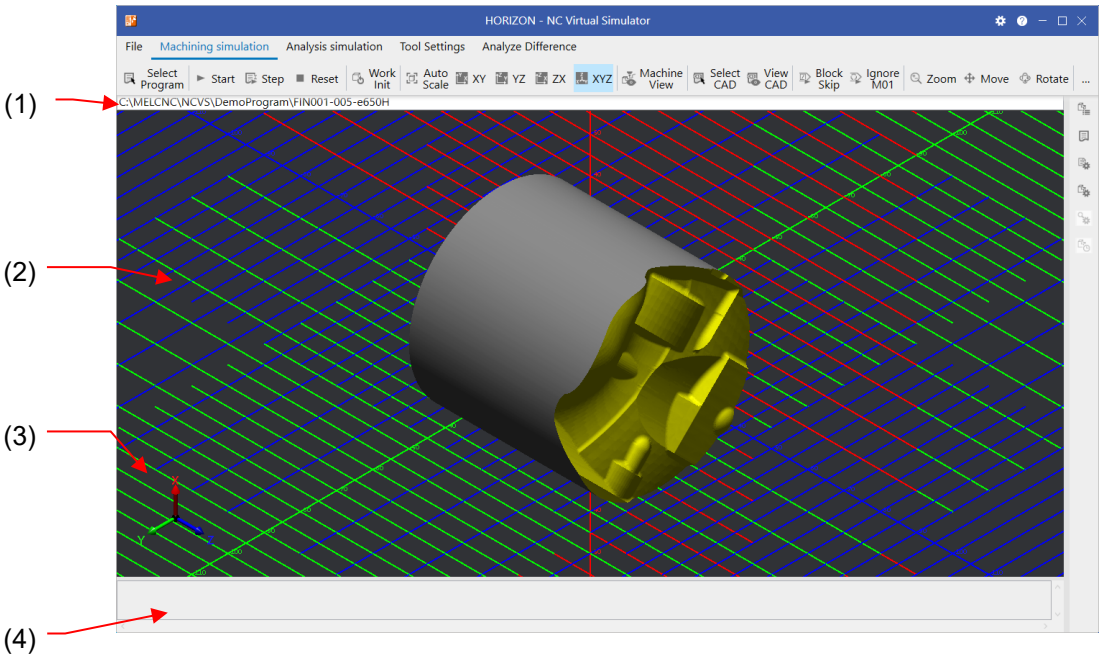


Interference check

6.1 Machining simulation window

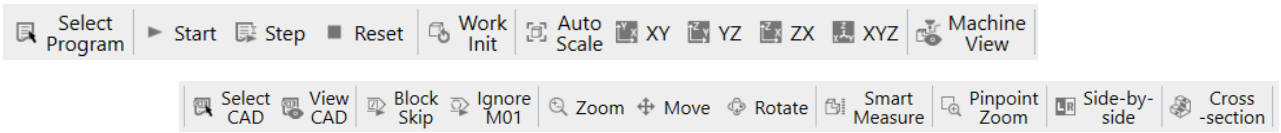
6.1.1 Screen configuration and display items

The machining simulation window displays the simulation of the currently selected machining program.



Item	Description
(1) Program path	Displays the file path of the selected machining program. The path combined with the file name by a backslash (\) or a Yen symbol (¥) is displayed.
(2) 3D graphics area	Displays the in-process workpiece shape and the tool. You can change the point of view and zoom the workpiece shape.
(3) Coordinate system	Indicates the directions of the coordinate system. The axis directions of base axes I, J, K are indicated by arrows. The interval between tick marks depends on the size of the workpiece.
(4) Message area	Displays the alarms that occur while machining simulation is running.

6.1.2 Machining simulation commands



The following lists the commands that can be used in machining simulation.

	Command	Description	Reference
1	Select Program	Select a program. Choose a program to simulate from the file selection dialog that pops up.	6.3.5 Select a program
2	Start	Start simulation. This command is highlighted while simulation is running. If you click this command when it is highlighted, simulation is paused at the end point of the current block.	6.3.6 Start simulation
3	Step	Run the program block by block.	6.3.6 Start simulation
4	Reset	Stop the ongoing simulation.	6.3.7 Stop simulation
5	Work Init	Initialize the workpiece shape. The shape of the unmachined workpiece is displayed.	-
6	Auto Scale	Zoom the workpiece shape so that the size is about 90% of the graphics area.	-
7	XY	Change the plane where the workpiece shape is displayed or change the display to 3D. The scale in the plane selected before switch is retained.	6.3.8 Change the point of view on workpiece
8	YZ		
9	ZX		
10	XYZ		
11	Machine View	Toggle the display between workpiece simulation and machine simulation. If you click this command when it is not highlighted, the command is highlighted and machine simulation is displayed. If you click this command when it is highlighted, the highlight of the command is removed, and workpiece simulation is displayed. The highlight state of this command is the same as that before the application is started and that before the project is selected again.	6.3.11 Switch between machine simulation and workpiece simulation
12	Select CAD	Select a product CAD model file. The selected product CAD model is superimposed on the workpiece shape. Files in STEP format (*.stp, *.step) and in STL format (*.stl) can be selected.	6.3.12 Superimpose product CAD model on workpiece shape
13	View CAD	Switch whether the product CAD model is superimposed. When this command is highlighted, the model is superimposed.	6.3.12 Superimpose product CAD model on workpiece shape
14	Block Skip (*1)	Select whether to execute blocks containing "/" (slash) in the program to be simulated. ON: Simulation is run by skipping blocks with "/". OFF: Simulation is run including blocks with "/". The highlight state of this command is the same as that before the application is started and that before the project is selected again.	-

	Command	Description	Reference
15	Ignore M01 (*1)	Specify whether to stop the program when an M01 command is issued. ON: Simulation is not stopped even when an M01 command is issued. OFF: Simulation is stopped when an M01 command is issued. The highlight state of this command is the same as that before the application is started and that before the project is selected again.	-
16	Zoom (*1)(*2)	Zoom in or out of the displayed workpiece shape. ・ With this command highlighted, press the [↑] [↓] keys on the keyboard, or left-click and drag up and down in the 3D graphics area, to zoom in and out of the displayed workpiece shape. ・ If you rotate the mouse wheel with the mouse pointer in the 3D graphics area, the displayed workpiece shape graphic is zoomed in or out regardless of whether this command is highlighted.	6.3.9 Adjust the workpiece display position, angle and size
17	Move (*1)(*2)	Move the displayed workpiece shape. ・ With this command highlighted, press the [↑] [↓] [←] [→] keys on the keyboard, or left-click and drag in the 3D graphics area, to move the workpiece.	
18	Rotate (*1)(*2)	Rotate the displayed workpiece shape. ・ With this command highlighted, press the [↑] [↓] [←] [→] keys on the keyboard, or left-click and drag in the 3D graphics area, to rotate the workpiece. ・ If you right-click and drag in the 3D graphics area, the displayed workpiece shape is rotated regardless of whether this command is highlighted.	
19	Smart Measure (*1)	Measure distance and angle between two points selected by mouse on the displayed workpiece shape or a machine model. (Optional function)	6.3.14 Measure distance on workpiece shape
20	Pinpoint Zoom (*1)(*2)	Clearly enlarge the range specified by mouse.	6.3.10 Partially enlarge workpiece shape
21	Side-by-side	Divide the simulation display area to display the workpiece shape and the machining shape of the same workpiece in the analysis simulation side by side. When the two screens are displayed right and left, this command is highlighted. When this command is clicked again, only the workpiece shape is displayed, and the highlight of this command is removed.	6.3.16 Display machining shape side by side

	Command	Description	Reference
22	Cross-section (*2)	Display the cross section of the position specified on the workpiece shape. If any of the following operation is performed while the command is highlighted, the highlight is removed. ・ Press the [Enter] key. ・ Select a command other than [Auto Scale], [XY], [YZ], [ZX] and [XYZ]. ・ Click the [Work pos update], [Redraw], or [Export Work Form] button of the sub-display "Machining simulation settings" or change the jig-related settings. ・ Select [Move the position of the product model], [Save POV info] or [Restore POV info] in the right click menu on the 3D graphics area. ・ Move to a screen other than the machining simulation window.	6.3.15 Display cross section of workpiece shape

(*1) The highlight indicates that the command is enabled. Every time the command is clicked, it is switched between highlighted and released (enabled and disabled).

(*2) The [Zoom], [Move], [Rotate] and [Pinpoint Zoom] commands cannot be enabled at the same time. When one of them is enabled, the highlight of the other command is removed.

(Note 1) The [Select Program], [Work Init], [Select CAD], [View CAD], [Smart Measure], [Pinpoint Zoom] and [Cross-section] commands are not selectable while simulation is running or paused.

(Note 2) The [Start], [Step] and [Reset] commands are not selectable when no machining program is selected.

(Note 3) All commands are highlighted when you left-click the command when it is not highlighted.

6.2 Preparation

To perform machining simulation, first set the data of the tool and workpiece to be used in simulation.
The following describes how to prepare the data.

6.2.1 Registering tool data

Register the tool data such as the shape and length of the tool and machined color to be used in simulation.
The tool data is registered in the Tool Settings window. For detailed information, see "8 Tool settings". If you select a tool whose shape is not registered when running simulation, the following tool is displayed.
When the TO command is executed, the tool currently displayed, including its holder, is hidden.

Tool data	Description
Length	Not displayed
Diameter	Not displayed
Cut. Edge L	Not displayed
Holder H	50 (mm)
Holder D	30 (mm)

6.2.2 Setting workpiece data

Set the shape, dimension, reference point position, etc. of the workpiece to be used in simulation.

You can set data such as the workpiece reference point shift amount, dimension, color, etc. from the [Machining simulation settings] sub-display.
See "(4) Workpiece" of "4.3.2.4 Machining simulation settings" for the setting range and the guide figure of each item.

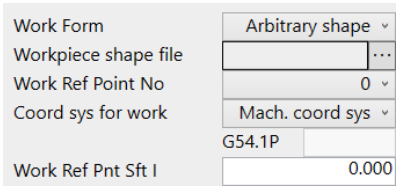
■ Set Work Form

Block/Pipe/Arbitrary shape can be set in "Work Form".
For "Arbitrary shape", a workpiece shape file created by a CAD tool according to the actual workpiece shape can be used as a simulation workpiece shape. Available workpiece shape files are those in STL format and DAT format.

The following shows an example to set an arbitrary shape.

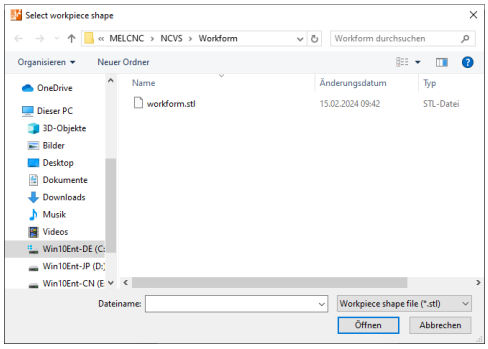
(1) Click the arrow next to [Work Form] to expand the drop-down list, and select [Arbitrary shape].

→ The "Workpiece shape file" field and the [···] button appear.
(*) They do not appear when "Block" or "Pipe" is selected in "Work Form".



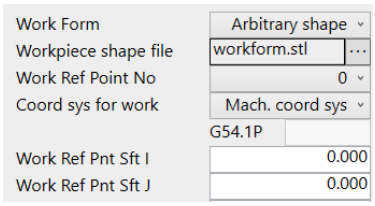
(2) Click the [···] button for "Workpiece shape file".

→ A file selection dialog appears.



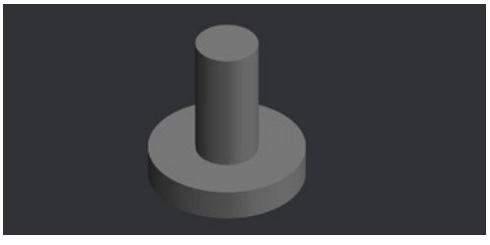
(3) Select a file to be used as workpiece shape and click [Open].

→ The progress window appears and reading of the selected file starts. To stop reading the file, click the [Cancel] button. Note that the [Cancel] button does not appear when reading a file in DAT format. When the reading is completed, the name of the selected file is displayed.



(4) Click the [Work Init] command.

→ The workpiece shape is initialized with the selected workpiece shape.
(*) When other than "Arbitrary shape" is set in "Work Form", the change of the workpiece reference point is applied by clicking the [Work pos update] button.



■ **Set Work Ref Point No.**

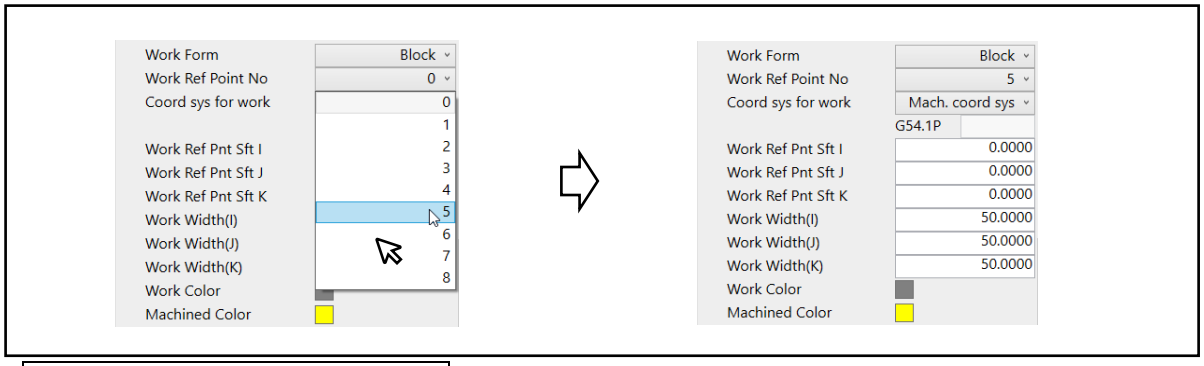
Select the reference point No. of the workpiece.

The number in the guide figure indicates the reference point number.

Decide the workpiece reference point position, and select the number at the position.

(1) Click the arrow next to [Work Ref Point No.] to expand the drop-down list.

(2) Choose Work Ref Point No. from the drop-down list.
(Example) 5 → The selected Work Ref Point No. is displayed in the field. The [Work pos update] button is enabled.



(3) Click the [Work pos update] button. → The changed reference point coordinate of the workpiece is applied to the machining simulation window.

(*) When [Block] is selected for "Work Form", the behavior is the same between when [0] is set for "Work Ref Point No." and when [5] is set.

(*) The guide figure switches according to the machine model (vertical or horizontal machining center). For detailed information, see "4.3.2.4 Machining simulation settings".

■ Set Coord sys for work

Select the coordinate system for workpiece.

(1) Click the arrow next to [Coord sys for work] to expand the drop-down list.

(2) Choose a coordinate system for workpiece from the drop-down list.
(Example) G54.1P

→ The selected coordinate system for workpiece is displayed in the field.
The [Work pos update] button is enabled.

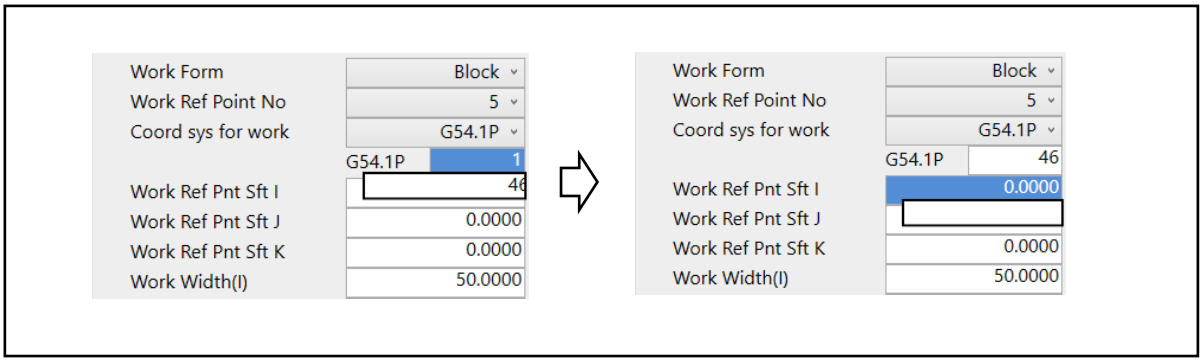
(*) When any item other than [G54.1P] is selected, the number of sets of G54.1P cannot be entered.

(3) Click the field for the number of sets of G54.1P.

→ The field is highlighted and an input dialog appears.

(4) Enter the number of sets of G54.1P in the input dialog and press the [Enter] key.
(Example) 46

→ The entered number of sets of G54.1P is displayed in the field, and the setting target moves to the next item.



(5) Click the [Work pos update] button.

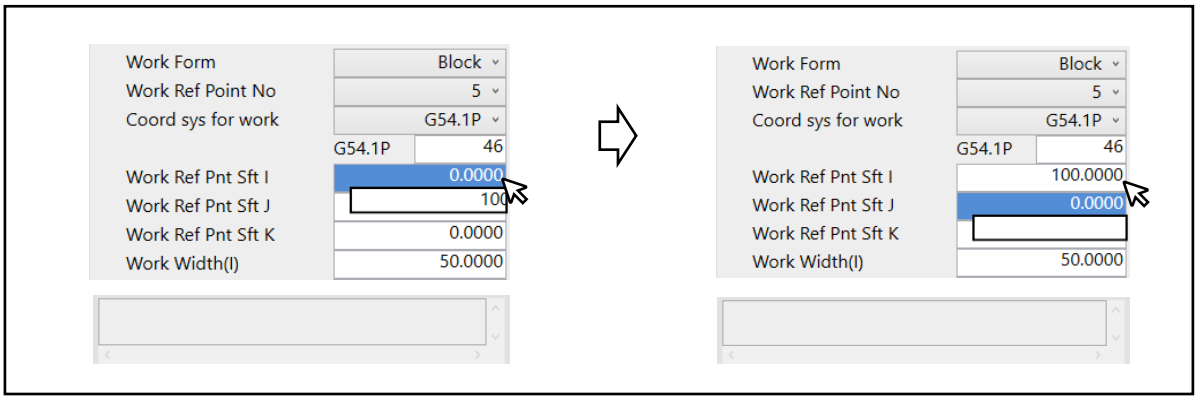
→ The changed reference point coordinate of the workpiece is applied to the machining simulation window.

■ Set Work Ref Pnt Sft

For Work Ref Pnt Sft, specify the shift amount from the reference point of the coordinate system selected with [Coord sys for work] to the reference point of the workpiece.

(1) Click the "Work Ref Pnt Sft I" field. → The field is highlighted, and an input dialog appears.

(2) Enter shift amount to the reference point in the input dialog and press [Enter].
(Example) 100 → The input shift amount to the reference point is displayed in the field, and the selection for setting moves on to the next field.
The [Work pos update] button is enabled.



(3) Set the other Work Ref Pnt Sft data in the same manner.

(4) Click the [Work pos update] button. → The changed reference point coordinate of the workpiece is applied to the machining simulation window.

(*) If the position of the workpiece graphic origin (Work Ref Point No 5) will be out of the range as a result of the settings of Work Ref Pnt Sft and Work Width, the entered value is not set, and the "Out of range" message appears.

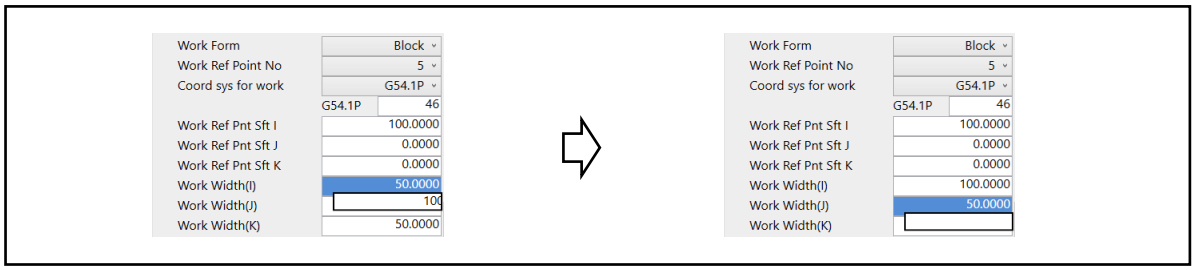
If you press [Esc], the input dialog disappears.

■ **Set Work Width (I)**

Set Work Width in mm when mm system is in use; in inch when inch system is in use.

(1) Click the "Work Width (I)" field.

(2) Enter Work Width.
(Example) 100 → The input Work Width is displayed in the field.



(3) Set the other Work Width data in the same manner.

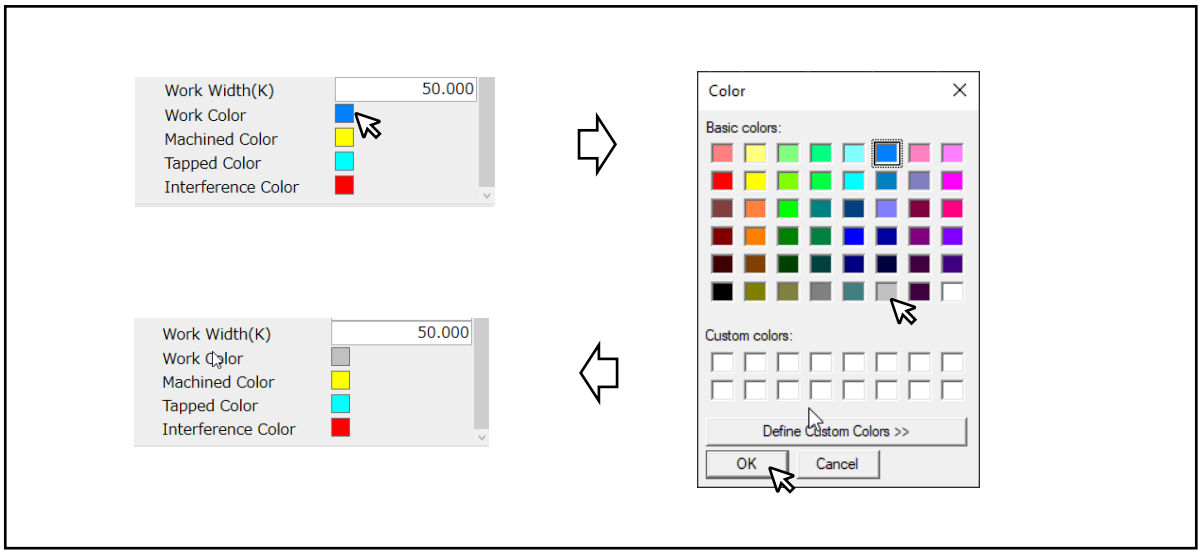
(*) If the position of the workpiece graphic origin (Work Ref Point No 5) will be out of the range as a result of the settings of Work Ref Pnt Sft and Work Width, the workpiece graphic origin is converted to the maximum/minimum value of the range.

■ **Set Work Color**

Set the display colors of workpiece, machining surface and interference surface.
When the display colors of them are not changed, this setting is not required.

(1) Click the "Work Color" field. → The "Color" dialog appears.

(2) Choose a color from the "Color" dialog, and click [OK]. → The "Work Color" field is displayed in the color you selected.



(3) Set the other colors in the same manner.

(*) If workpiece data is not set when simulation is run, the workpiece is displayed based on the following data.

Workpiece data	Description
Work Form	Block
Work Ref Point No.	0
Coord sys for work	Machining coordinate system
Work Ref Pnt Sft I	0 (mm)
Work Ref Pnt Sft J	0 (mm)
Work Ref Pnt Sft K	0 (mm)
Work Width (I)	100 (mm)
Work Width (J)	100 (mm)
Work Width (K)	50 (mm)
Work Color	Gray
Machined Color	Yellow
Tapped Color	Light blue
Interference Color	Red

6.3 Operation

Click [Machining simulation] in the function menu.

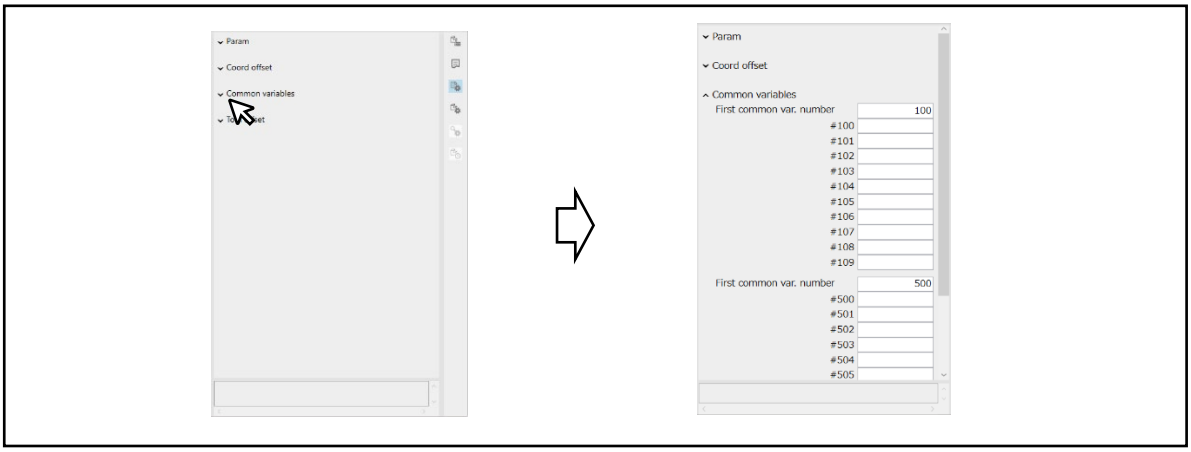
6.3.1 Set NC data (coordinate system, parameters, etc.)

You can set common variables and coordinate system offset to be used in machining program.

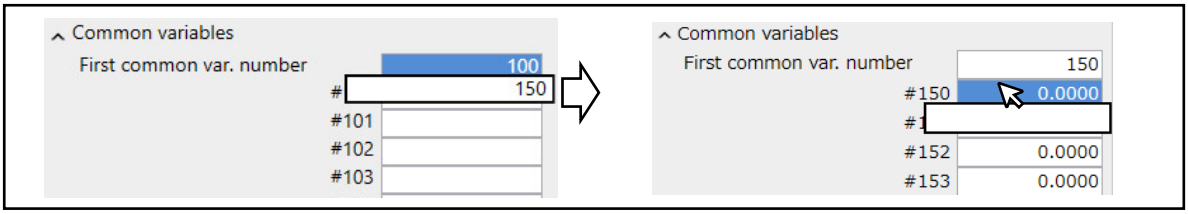
The following is an example of how to set a value to the common variables.

(1) Click the icon of the sub-display switch button to open "NC Data List".

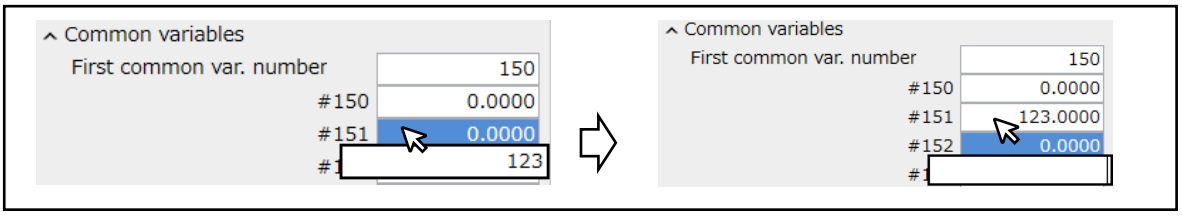
(2) Click ⌄ to expand "Common variables".



(3) Click the input field of "First common var. number" to enter a desired common variable number.



(4) The common variable of the entered number is displayed.
Click the input field of the desired variable to enter a value for the variable.



(Note) If an out-of-range value is input, the message "This value cannot be set." appears.

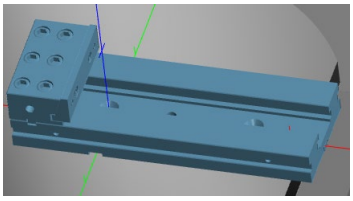
6.3.2 Register jig model

By creating a jig used in the actual machining by CAD and registering it as a jig model, an interference check can be performed in a state closer to the actual machining. The jig model that can be registered is a file in STEP format or STL format. Up to two models can be registered. Registration of jig model and interference check can be used even when there is no machine model.
Register a jig model by [Jig setting] of the [Machining simulation settings] sub-display.

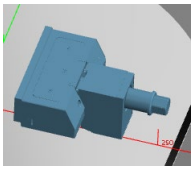
■ When there are two jigs

Up to two jig models can be registered. For a jig such as vise to hold a workpiece, create a file by regarding a piece as a jig model. Even when the workpiece size is changed, the jig model can be used only by adjusting the jig position without changing the jig model.

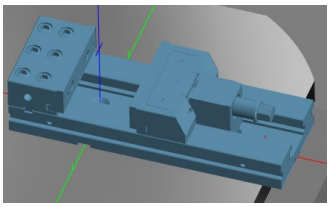
[Jig model 1]



[Jig model 2]



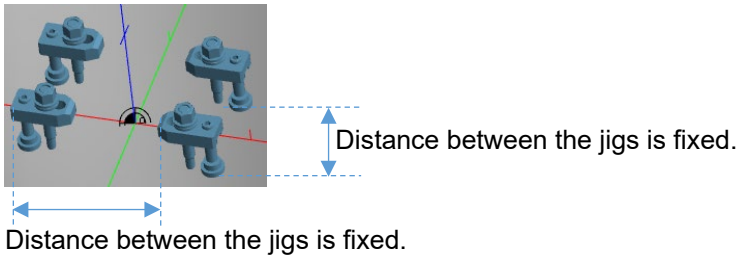
[Combined jig model]



■ When there are three or more jigs

When three or more jigs are used, output two or more jigs as a model file by CAM. They must be combined in up to two jig files. When the jigs are combined in a model file, distance between the jigs is fixed. Adjust the positions before outputting the model file.

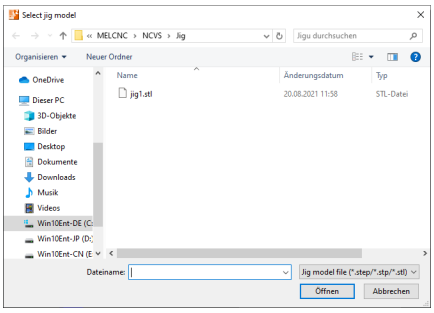
Example: When two or more jigs are combined in a jig model file



■ Register jig model

(1) Click the […] button for jig model selection.

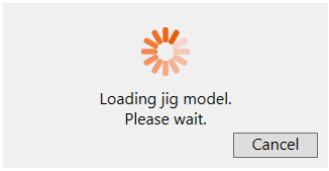
A file selection dialog appears.



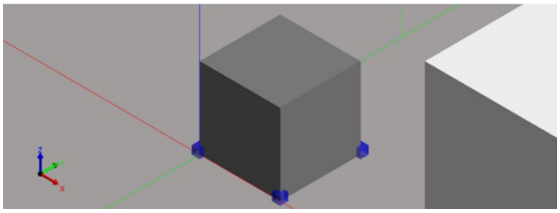
(2) Select a jig model file to be registered, and click [Open].

The progress window appears and reading of the jig model file starts.

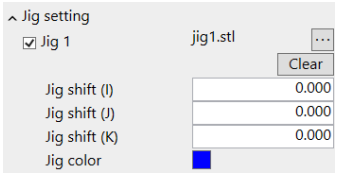
To stop the reading before it completes, click the [Cancel] button.



When the reading is completed, the selected jig model is displayed.



The checkbox of the jig model is checked, and the file name of the registered jig model is displayed.



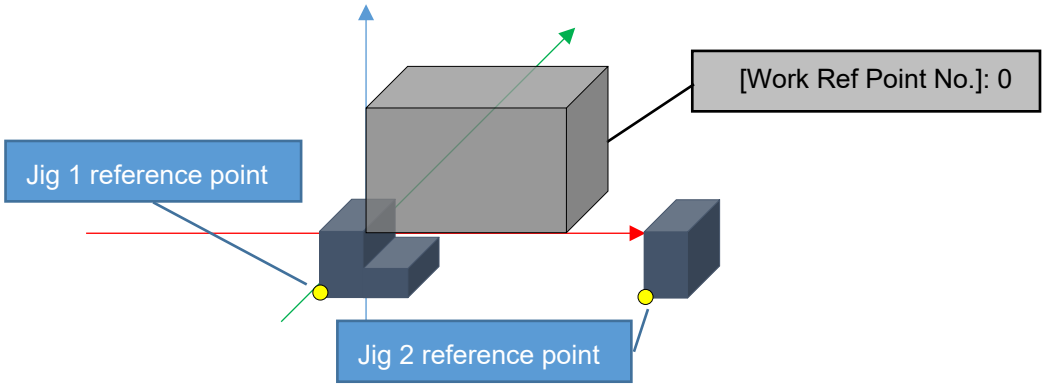
(*) When the checkbox of the jig model is unchecked, the jig model is hidden.

(3) When the display position of the jig model is deviated from the workpiece, set "Jig shift (I/J/K)" to adjust the jig position.

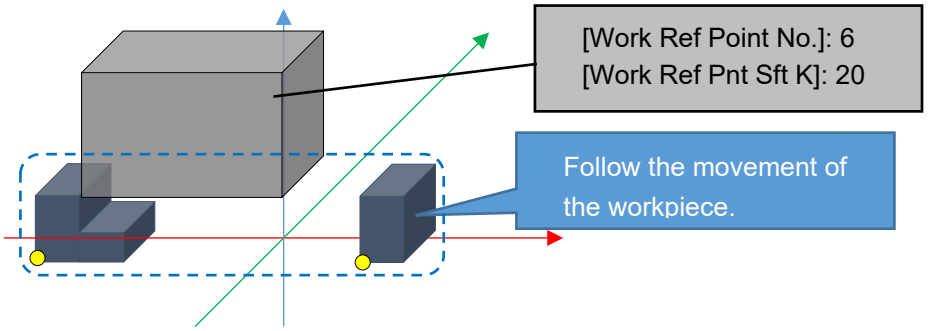
The jig model is moved by the entered shift amount.

(Note) The jig model moves following the movement of the workpiece position. When two jig models are registered, the model positions can be moved respectively.

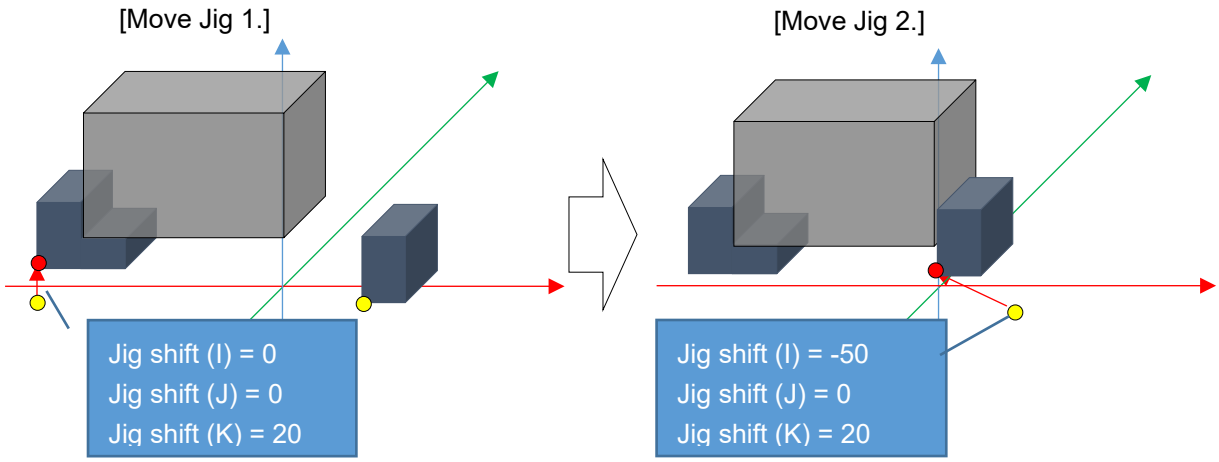
(1) Register a jig. (It is displayed at where the jig shift amount is "0".)
Create the jig by regarding the point where [Work Ref Point No.] is "0", as the reference point.



(2) Change the workpiece position.
Example: Set [Work Ref Point No.] to "6" and [Work Ref Pnt Sft] to "20".

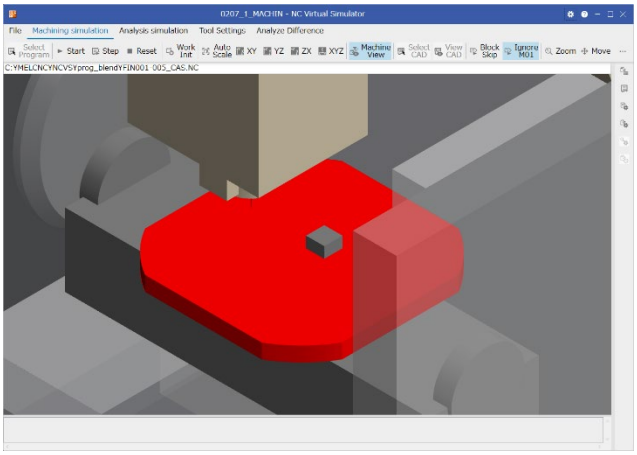


(3) Adjust the jig position.
When the display position of the jig is deviated from the workpiece, set "Jig shift (I/J/K)" to adjust the jig position.
Example: Set [Jig shift (K)] of [Jig 1] to "20", [Jig shift (I)] of [Jig 2] to "50" and [Jig shift (K)] to "20".



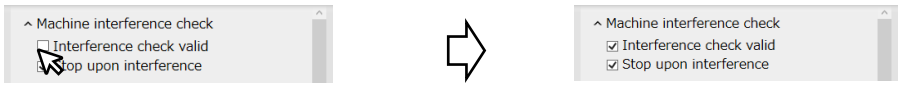
6.3.3 Check interference

Interference check is a function to detect interference between any of tool, workpiece, jig, and machine part, interference between machine parts, etc. If the tool, workpiece, jig, and machine parts come into contact with one another, the contacting parts are indicated with the interference color specified in the "Machining simulation settings".
When the [Interference check valid] checkbox is checked in the sub-display "Machining simulation settings", this function is enabled. When an interference is detected, whether to stop or not to stop the simulation can be switched by [Stop upon interference] in the sub-display.



(1) Click the [Interference check valid] checkbox. ➡ The [Interference check valid] checkbox is checked, and the interference check function is enabled.

Click the [Interference check valid] checkbox



If you click the [Interference check valid] checkbox again, the checkbox is cleared, and interference check is disabled.

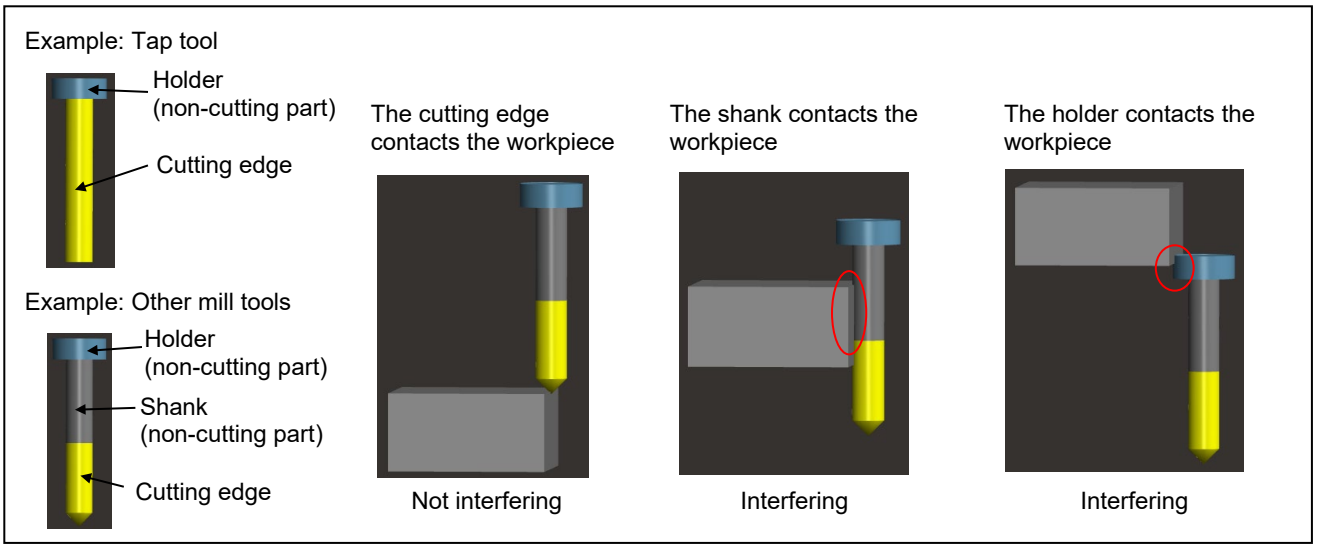
The following table summarizes the combination of the parts between which interference is detected.

Part 2 \ Part 1		Workpiece	Machine part (*2)	Jig (*2)
Tool	Cutting edge	- (*1)	✓	✓
	Holder Shank	✓	✓	✓
Machine part (*2)		✓	✓	✓

✓: Interference is detected. - : Interference is not detected.

(*1) Interference is detected if the workpiece and the cutting edge interfere with each other due to a movement other than cutting feed.

(*2) Even when the jig/machine model is hidden, interference check is performed.



(Note 1) If the tool comes into contact with an object other than the workpiece, interference is detected regardless of whether it is the cutting edge or non-cutting part of the tool.

(Note 2) When the [Interference check valid] checkbox is checked, alarm messages are displayed in the message display area when an interference occurs.

(Note 3) If simulation is resumed after it stopped because of interference, simulation continues in interference state until the interference is resolved.

(Note 4) Executing steps consecutively during interference will cancel the interference status and clear the alarm messages. Alarm messages are not displayed until interference occurs again.

6.3.4 Change the simulation priority

The simulation speed can be changed in three levels according to the purpose. Select the speed from the following modes by "Simulation priority" of the sub-display "Machining simulation settings".

High-speed mode (high): Simulation is performed at the highest speed.
This operation mode is mainly used to grasp the machining time prediction, check whether machine interference exists, etc.

High-speed mode (medium): Simulation is performed at high speed.
This operation mode is mainly used to check the analysis results that require accuracy, grasp the machining time prediction, check whether machine interference exists, etc.

Normal mode: Simulation is performed at normal speed.
This operation mode can be used for the most purposes to check the analysis results that require accuracy, check the behavior by step operation, identify the interference block, etc.

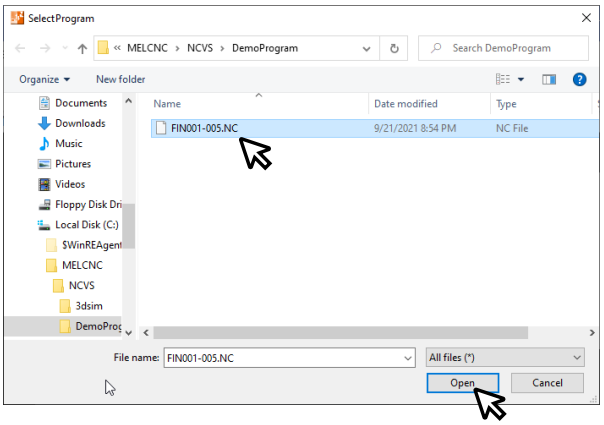
(Note) Accuracy, behavior upon interference, etc. are different depending on the operation mode.
For details, see the following table.

<table>
<tr><th rowspan="2">Item</th><th colspan="3">Simulation priority</th></tr>
<tr><th>High-speed mode (high)</th><th>High-speed mode (medium)</th><th>Normal mode</th></tr>
<tr><td>Simulation speed</td><td>◎</td><td>○</td><td>△</td></tr>
<tr><td>Simulation accuracy</td><td>○</td><td>◎</td><td>◎</td></tr>
<tr><td>Machining time prediction</td><td>○</td><td>○</td><td>○</td></tr>
<tr><td>Step operation</td><td colspan="2">Impossible</td><td>Possible</td></tr>
<tr><td>Stop upon interference</td><td colspan="2">Simulation is stopped when interference occurs.
However, the simulation is stopped at a position ahead of the program position where the interference occurs.</td><td>Simulation is stopped when interference occurs.</td></tr>
<tr><td>Tool compensation input (G10L10, etc.)</td><td colspan="2" rowspan="7">When a machining program including these commands is executed, they are applied to the simulation shape. After the simulation is executed, the setting returns to the state before the execution. They are not applied to "NC Data List" in the sub-display, and the setting before the execution remains.</td><td rowspan="7">When a machining program including these commands is executed, they are applied to the simulation shape, and the setting remains even after the simulation is executed. They are also applied to "NC Data List" in the sub-display.</td></tr>
<tr><td>Parameter input (G10L70)</td></tr>
<tr><td>Workpiece offset input (G10L2, etc.)</td></tr>
<tr><td>Other G10 commands</td></tr>
<tr><td>Common variable command (#100, #500, etc.)</td></tr>
<tr><td>Machining condition selection I (G120.1)</td></tr>
<tr><td>Tool change command (T01M06, etc.)</td></tr>
<tr><td>G code modal</td><td colspan="2">After the simulation is executed, the modal state returns to the one before the execution.</td><td>The modal state remains even after the simulation is executed.</td></tr>
<tr><td>NC Trainer2 plus link</td><td colspan="2">Impossible</td><td>Possible</td></tr>
<tr><td>Ignore M01 / Block Skip command</td><td colspan="2">Possible
(*) The response of change during simulation is slower than in normal mode.</td><td>Possible</td></tr>
<tr><td>Machine position at simulation start</td><td colspan="2">Machine coordinates reference point</td><td>Machine position when the execution of simulation in normal mode is completed</td></tr>
</table>

6.3.5 Select a program

Choose the machining program to simulate.

(1) Click the [Select Program] command. → A file selection dialog appears.



(2) Choose the machining program to simulate, and click [Open]. → The file name of the selected machining program is displayed in the program path area.

(Note 1) A file name must not exceed 32 characters; a folder name must not exceed 48 characters. If the file name is longer than 32 characters, or the folder name is longer than 48 characters, an error message dialog appears.

(Note 2) The main program and subprograms must be stored in the same folder.

6.3.6 Start simulation

Simulate the cutting process of the selected program, and render the graphic of the workpiece. Start simulation with the [Start] or [Step] command.
To run the program continuously, click [Start]. To run the program block by block, click [Step].
If no program is selected, simulation cannot be started. Select a program by following the steps in "6.3.5 Select a program" before starting simulation.

(1) Click the [Start] or [Step]command. → The command is highlighted, and the simulation starts. The counters and machining time in the sub-display "Processing status" are updated.

(2) When M02 or M03 in the program is issued, simulation is completed. → The highlight of the [Start] command is removed.

(*) If you click [Start] during continuous operation, simulation is paused.
If you click [Start] while simulation is paused, simulation resumes.

(*) If you click [Step] during continuous operation, simulation is run block by block from the next block.
If you click [Start] during step operation, the simulation is switched to continuous operation.

(*) If you click [Reset] while simulation is running, simulation is stopped after the current block completes.

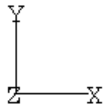
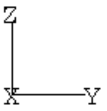
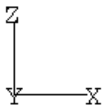
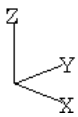
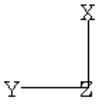
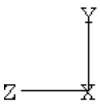
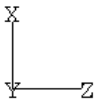
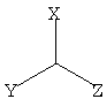
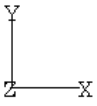
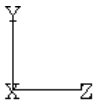
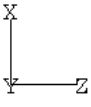
6.3.7 Stop simulation

Stop the ongoing simulation.

(1) Click the [Reset] command. → Simulation stops.
The highlight of the [Start] command is removed.

6.3.8 Change the point of view on workpiece

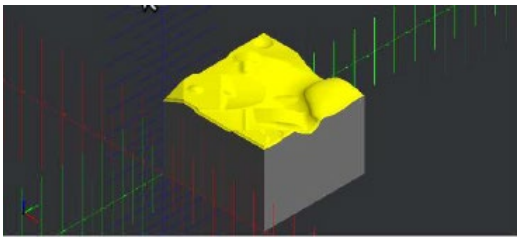
You can change the point of view of the workpiece shape (or machining shape) to be displayed in the simulation display area.
The point of view is changed as specified according to whether the machine is a vertical or horizontal machining center, which is determined based on the machine configuration of the selected project. If a project to which no machine model has been imported is selected, the point of view is for a vertical machining center.

	Machine coordinate system			
	XY plane	**YZ plane**	**ZX plane**	**XYZ (3D display)**
Vertical machining center				
Horizontal machining center (X axis is vertical)				
Horizontal machining center (Y axis is vertical)				

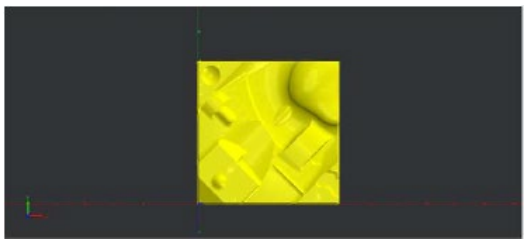
The view depends on the machine configuration and the command as shown below.
(The figure is an example of the machining simulation window.)

■ **Point of view for a vertical machining center**

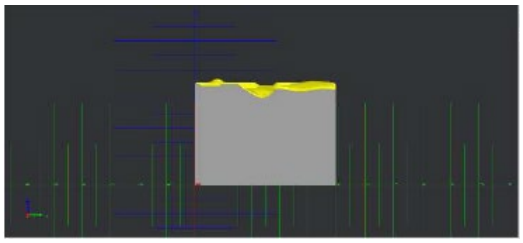
[XYZ] command



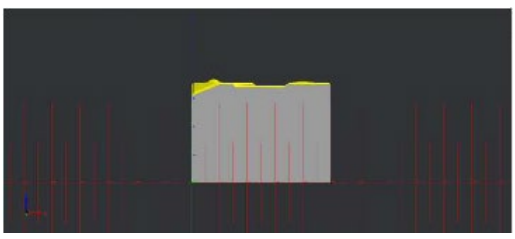
[XY] command (View from the plus side of Z axis)



[YZ] command (View from the plus side of X axis)

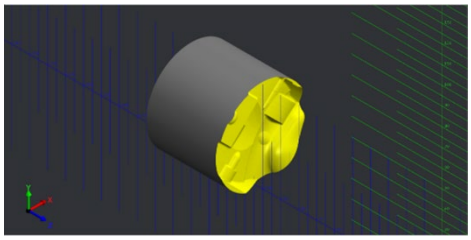


[ZX] command (View from the minus side of Y axis)

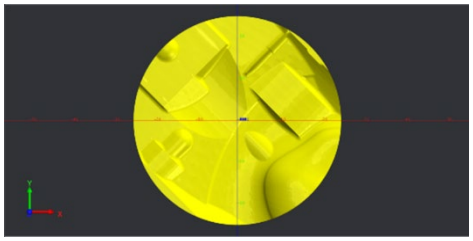


■ **Point of view for a horizontal machining center (for machine model with vertical Y axis)**

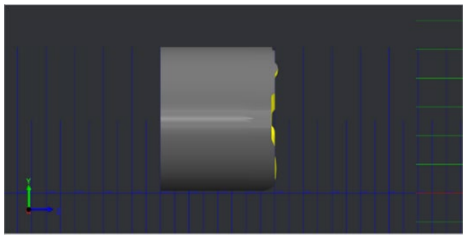
[XYZ] command



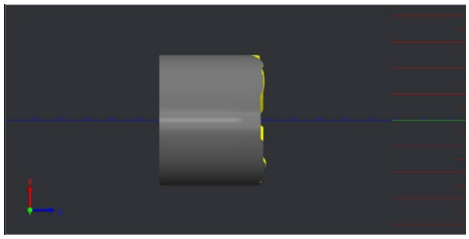
[XY] command (View from the plus side of Z axis)



[YZ] command (View from the plus side of X axis)



[ZX] command (View from the plus side of Y axis)

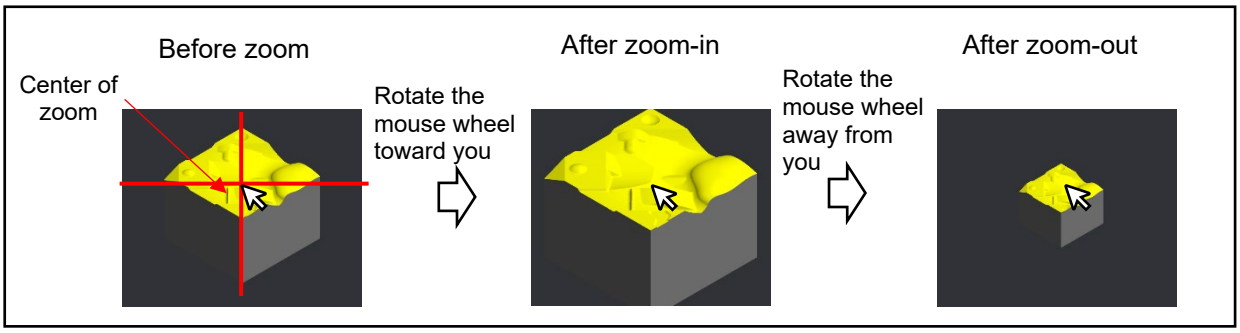


6.3.9 Adjust the workpiece display position, angle and size

You can zoom, move, and rotate the displayed workpiece shape.

■ **Zoom in/out the workpiece shape (1)**

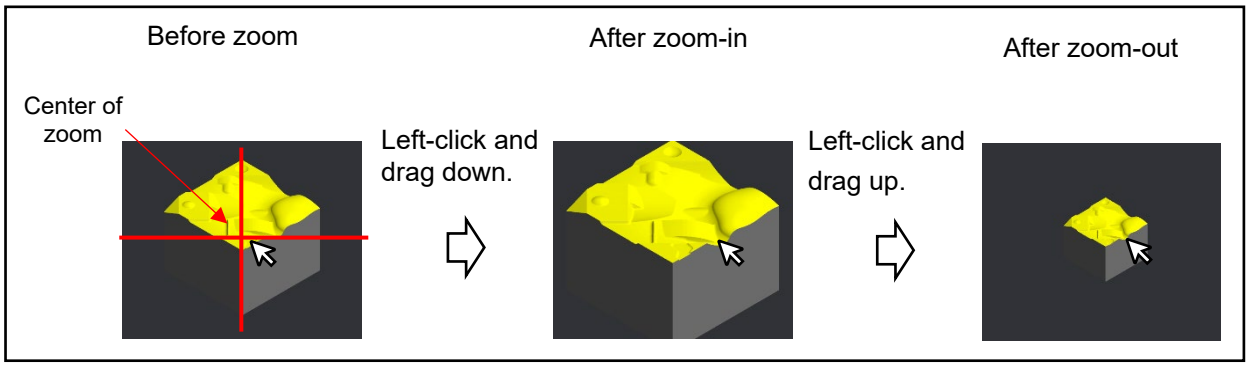
(1) With the mouse pointer in the 3D graphics area, rotate the mouse wheel. → The workpiece shape is zoomed in/out with the position of the mouse pointer as the center.
To zoom in, rotate the mouse wheel toward you, and to zoom out, rotate it away from you.
This operation is available regardless of whether the [Zoom] command is ON or OFF.



■ **Zoom in/out the workpiece shape (2)**

(1) Click the [Zoom] command. → The Zoom mode is turned on, and the [Zoom] command is highlighted.
The highlight of the [Move] or [Rotate] command is removed.

(2) With the mouse pointer in the 3D graphics area, left-click and up or down. → The workpiece shape is zoomed in/out with the position of the mouse pointer where drag was started as the center.
To zoom in, drag down, and to zoom out, drag up.

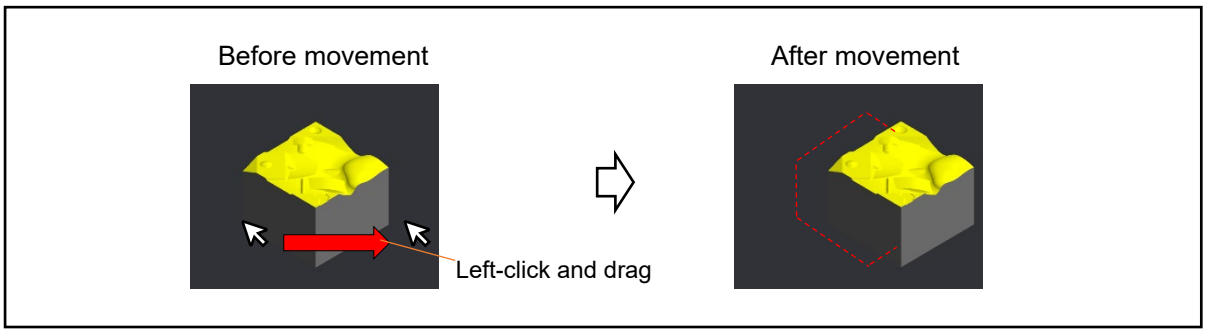


The [↑] and [↓] keys on the keyboard are available to zoom in and out the workpiece shape instead of the mouse left button operation.

(3) After the zoom-in/out operation, click the [Zoom] command to turn off the Zoom mode.

■ **Move the workpiece shape (1)**

(1) When the [Zoom] and [Rotate] commands are not highlighted, left-click and drag while the mouse pointer is in the 3D graphics area. → The workpiece shape moved in the direction to which you dragged is displayed again.

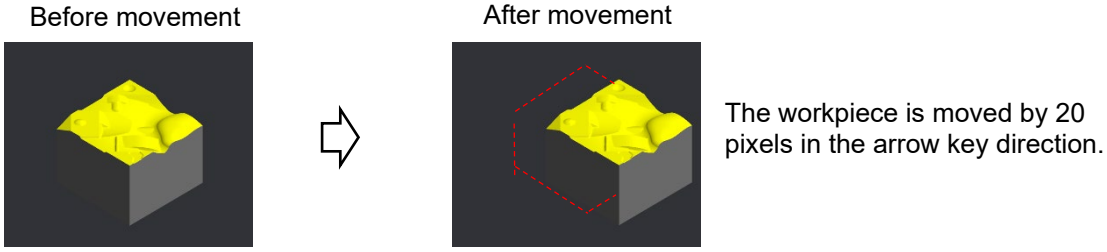


■ **Move the workpiece shape (2)**

(1) Click the [Move] command. → The Move mode is turned on, and the [Move] command is highlighted.
The highlight of the [Zoom] or [Rotate] command, if highlighted, is removed.

(2) Press the [→] key on the keyboard. → The workpiece shape moved in the right direction is displayed again.

You can also move the workpiece shape by pressing the [↑] [↓] [←] [→] keys on the keyboard.

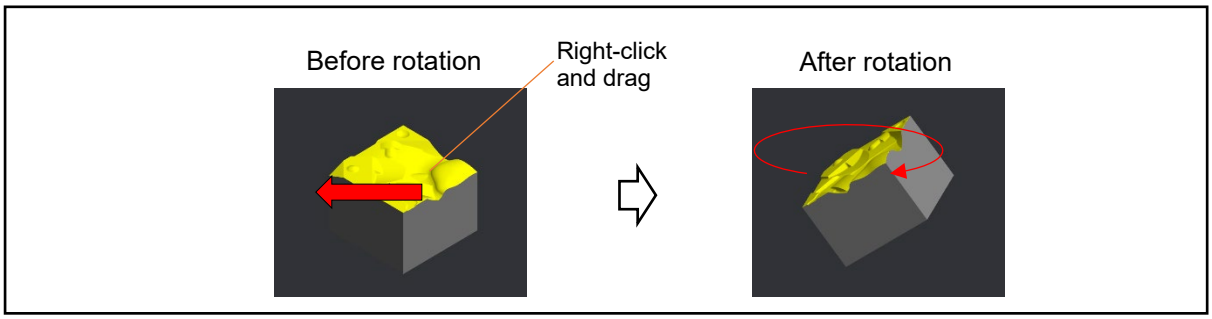


The workpiece is moved by 20 pixels in the arrow key direction.

(3) After the move operation, click the [Move] command to turn off the Move mode.

■ Rotate the workpiece shape (1)

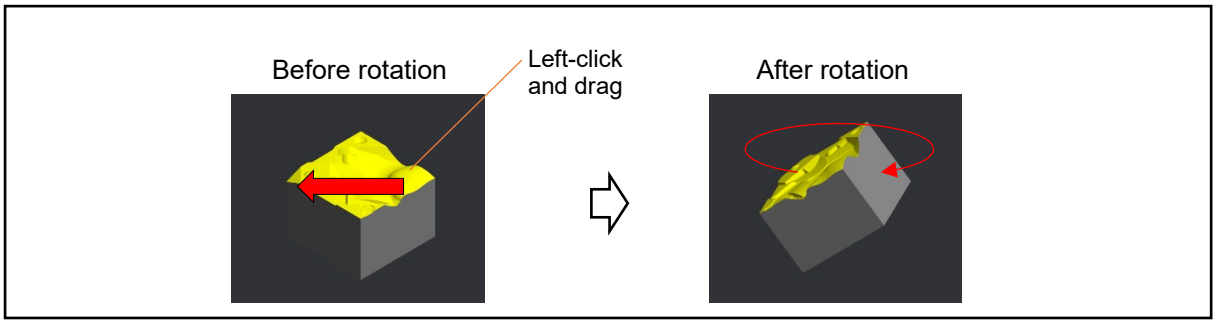
(1) With the mouse pointer in the 3D graphics area, right-click and drag. → The workpiece shape rotated in the direction you dragged is displayed again.
This operation is available regardless of whether the [Rotate] command is ON or OFF.



■ Rotate the workpiece shape (2)

(1) Click the [Rotate] command. → The Rotate mode is turned on, and the [Rotate] command is highlighted.
The highlight of the [Zoom] or [Move] command is removed.

(2) With the mouse pointer in the 3D graphics area, left-click and drag. → The workpiece shape rotated in the direction you dragged is displayed again.



The arrow keys on the keyboard are available to rotate the workpiece shape instead of the mouse left button operation.

(3) After the rotation operation, click the [Rotate] command to turn off the Rotate mode.

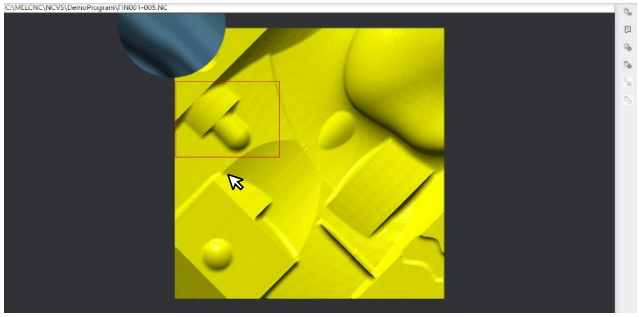
6.3.10 Partially enlarge workpiece shape

Pinpoint zoom is a function to be used when the clarity is insufficient when the normal zoom function is used, and to allow you to cut out the workpiece shape in the range specified by mouse to enlarge the graphic with a higher accuracy.
When [Cancel zoom.] in the right-click menu is selected, the workpiece shape returns to the original display.

(1) Click the [Pinpoint Zoom] command. → The [Pinpoint Zoom] command is highlighted.

(*) When the [Pinpoint Zoom] command is highlighted, the workpiece shape cannot be zoomed, moved and rotated by mouse.

(2) Move the mouse while holding the mouse left button. → A range selection rectangle is displayed so that the position where the mouse left button is clicked is the start point, and the position where the mouse is moved is the end point.

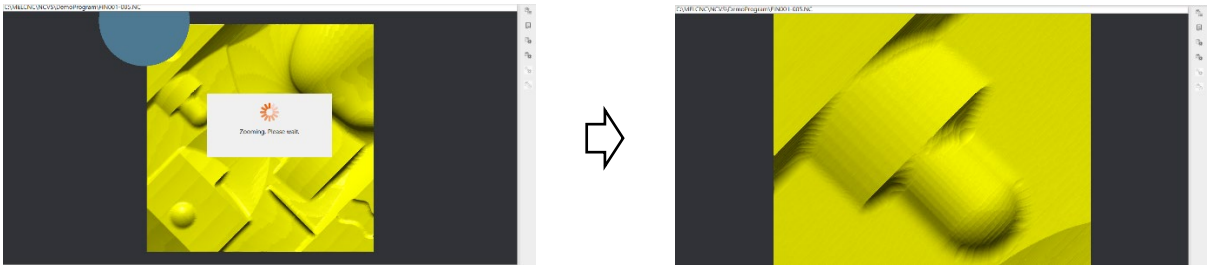


(*) Among the four vertices of the range selection rectangle, at least one of them must be on the workpiece. If all the four of them are not on the workpiece, pinpoint zoom is not enabled.
(*) If four sides of the range selection rectangle is less than 10 pixels, pinpoint zoom is not enabled.
(*) If the [Esc] key is pressed while the mouse is moved, the range selection rectangle disappears and pinpoint zoom is not enabled.

(3) Move the mouse to the end point of the pinpoint zoom range, and release the mouse left button.

The range selection rectangle is hidden and the progress window appears.
The size of the block to be partially enlarged is calculated from the range selection rectangle, and the range is enlarged.

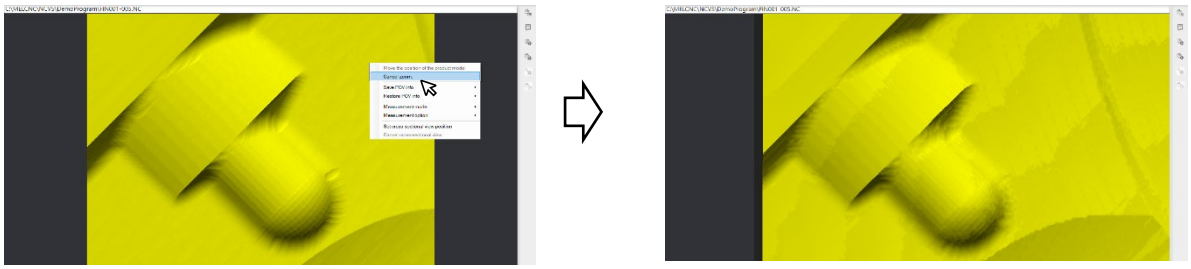
When the pinpoint zoom is completed, the progress window is hidden, and the highlight of the [Pinpoint Zoom] is removed.



(*) The partially enlarged workpiece shape can be zoomed, moved, rotated, and partially enlarged.
(*) When a project is saved while [Pinpoint zoom invalid] of the sub-display "Machining simulation settings" is checked and the workpiece shape in the project is partially enlarged, the workpiece shape in the selected range is displayed in an initialized state (the workpiece shape before the machining is displayed). For pinpoint zoom, uncheck [Pinpoint zoom invalid] of the "Machining simulation settings", and perform simulation graphic by the [Redraw] button, then perform the pinpoint zoom.

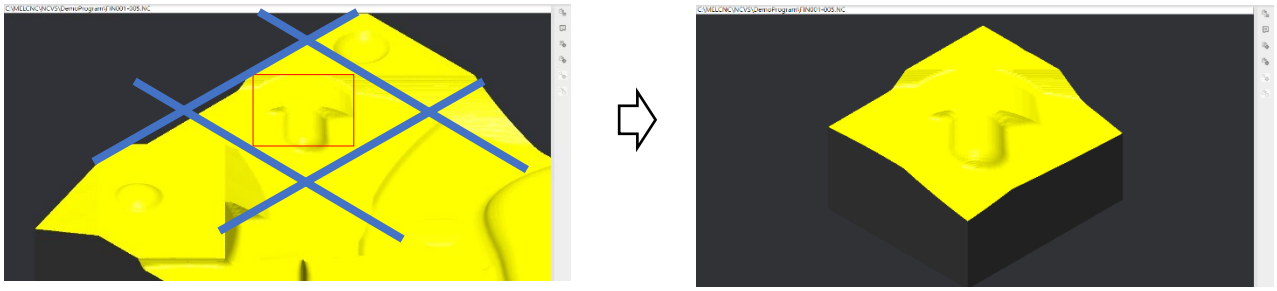
(4) Select [Cancel zoom.] in the right-click menu on the 3D graphics area.

The workpiece shape before pinpoint zoom is displayed.

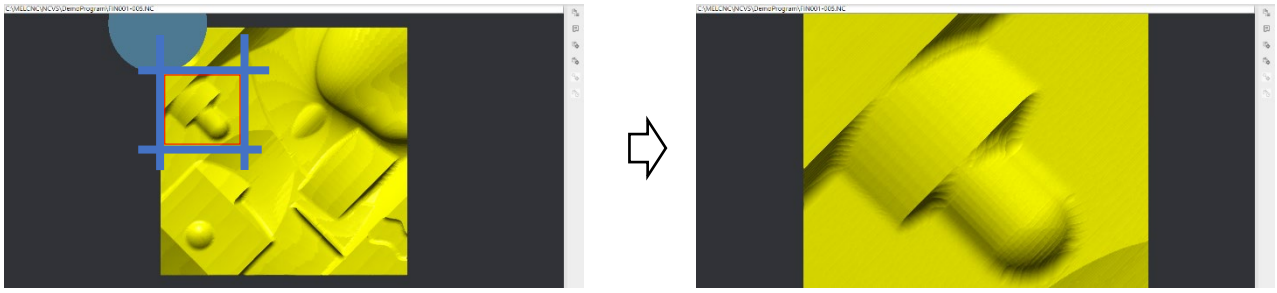


(Note 1) The size of the block to be partially enlarged is calculated from the lines parallel to the X, Y and Z axes that come in contact with the four vertices of the range selection rectangle.

[XYZ view]



[XY view]



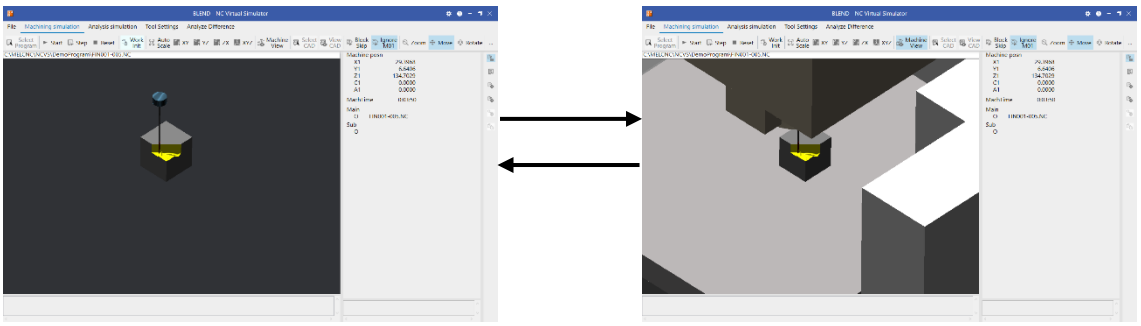
(Note 2) If simulation is started during pinpoint zoom, the workpiece shape before the pinpoint zoom is displayed and simulation is started.

(Note 3) When a project is saved during pinpoint zoom while [Geometry saving invalid] of the sub-display "Machining simulation settings" is not checked, the workpiece shape returns to the one before the pinpoint zoom. When [Geometry saving invalid] is not checked, the shape is not saved and the shape of the unmachined workpiece is displayed.

(Note 4) When [Export Work Form] is executed during pinpoint zoom, the workpiece shape before the pinpoint zoom is automatically displayed and the workpiece shape file is saved.

6.3.11 Switch between machine simulation and workpiece simulation

You can toggle between machine simulation and workpiece simulation by clicking [Machine View]. Machine simulation shows the machine and jig as well as the workpiece and tool, which helps you figure out the positional relationship in the actual machine and the position of interference. Workpiece simulation only shows the tool and workpiece, making it easy to check the shape of the workpiece and the process of machining.



Workpiece simulation　　Machine simulation

The following describes how to toggle between workpiece simulation and machine simulation.

(1) Click the [Machine View] command. → Machine simulation is displayed. The [Machine View] command is highlighted.

(2) Click the [Machine View] command again. → Workpiece simulation is displayed. The highlight of the [Machine View] command is removed.

(Note 1) Even when workpiece simulation is selected, interference involving the machine or jig is detected and simulation is stopped.

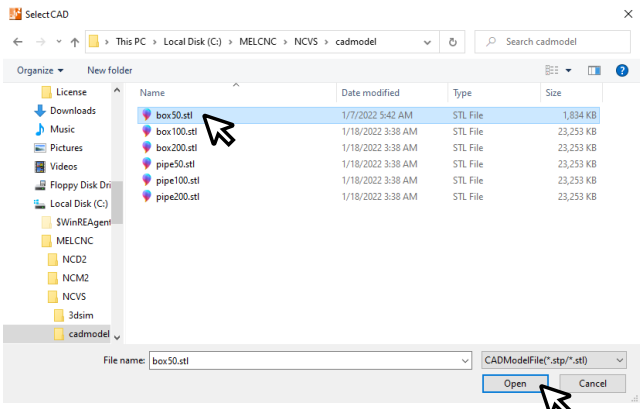
6.3.12 Superimpose product CAD model on workpiece shape

When the product CAD model is superimposed on the machining-simulated workpiece shape, you can check whether a process of the machining program is missed or the processing order is correct. The product CAD model that can be read is a file in STEP format or STL format.
Prepare data where the mm/inch unit system of the product CAD model file is in accordance with the unit system of the NC parameter (#1041 I_inch).

■ Superimpose product CAD model on workpiece shape

(Note) When a product CAD model file including an invalid shape is read, the application may end. Save the project before reading the file.

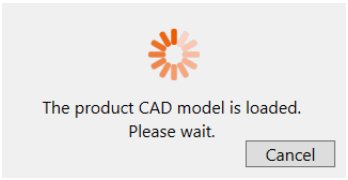
(1) Click the [Select CAD] command. → A file selection dialog appears.



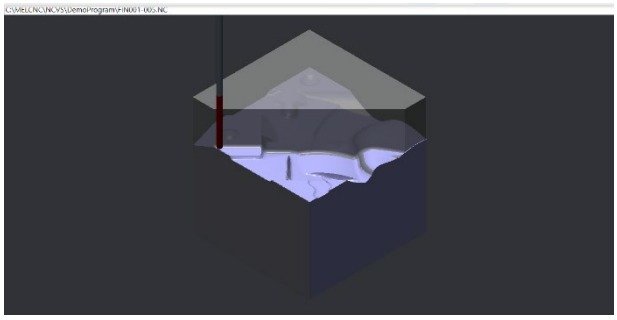
(*) When the product CAD model file is already selected in analysis simulation, the already selected product CAD model is superimposed by clicking the [View CAD] command.
(*) The selected product CAD model file is displayed by clicking the [View CAD] command in analysis simulation or on the difference analysis window.
(*) A file name must not exceed 32 characters; a folder name must not exceed 48 characters. If the file name is longer than 32 characters, or the folder name is longer than 48 characters, an error message dialog appears.

(2) Select a file of the product CAD model to be superimposed on the workpiece shape, and click [Open]. → The progress window appears and reading starts.
The reading time depends on the size of workpiece shape and CAD model.

To stop the reading before it completes, click the [Cancel] button.

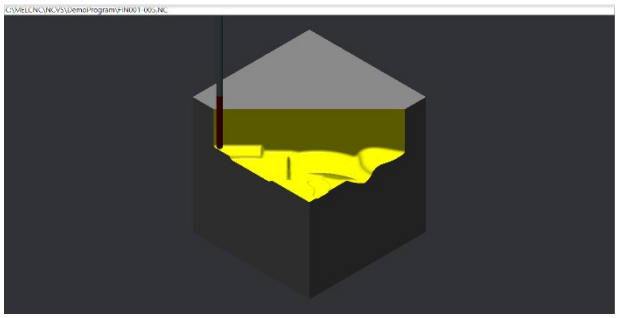


When the reading is completed, the selected product CAD model is superimposed on the workpiece shape, and the [View CAD] command is highlighted.



■ **Cancel superimposition of the product CAD model**

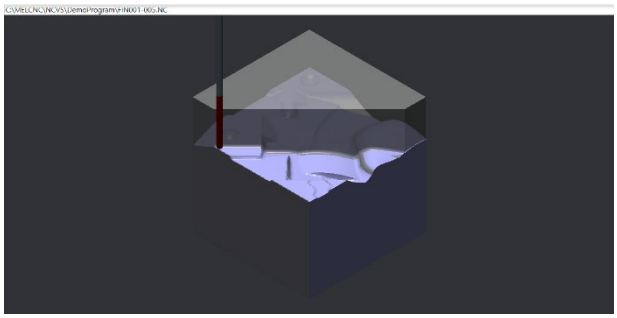
(1) When the product model is superimposed, click the [View CAD] command. → The superimposition of the product CAD model is canceled. The highlight of the [View CAD] command is removed.



(*) Also, when the [Start] or [Step] command is clicked to start simulation while the product mode is superimposed, the superimposition of the product CAD model is canceled.

■ **Superimpose the product CAD model again**

(1) When the product model is not superimposed, click the [View CAD] command. → The product CAD model is superimposed again. The [View CAD] command is highlighted.



(Note1) When the product CAD model is superimposed, the colors of the machined surface and the tapped surface are the same as the workpiece color. To return the colors to the original ones, start machining simulation again.

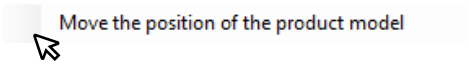
6.3.13 Adjust display position of product CAD model

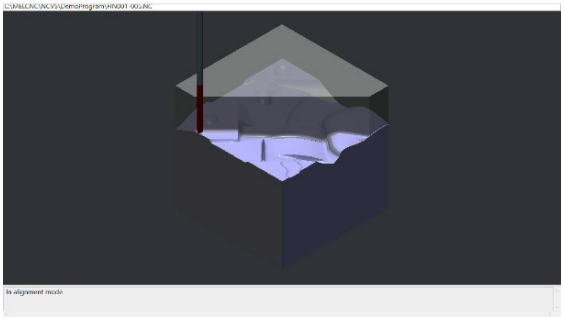
When the product CAD model is read to be superimposed, the model may be displayed at an unexpected position. In this case, adjust the superimposition position by the following operation.

■ **Adjust display position of product CAD model**

(1) Click the [View CAD] command. → The product CAD model is superimposed on the workpiece shape.
The [View CAD] command is highlighted.

(2) Click [Move the position of the product model] in the right click menu on the 3D graphics area. → The mode turns to the positioning mode of the product CAD model.
"In alignment mode" is displayed in the message display area.





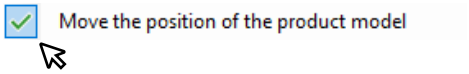
(3) Click the [XY], [YZ], [ZX] or [XYZ] command to change the point of view. → The workpiece shape and the product CAD model are displayed from the view of the clicked command.

(4) Press the arrow key of the desired direction. → The product CAD model moves to the direction of the arrow key.

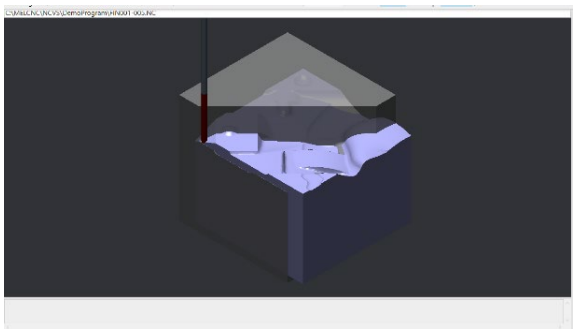
(*) When [XYZ] is clicked, the model moves to the following direction.
Vertical machining center
[↑], [↓] keys: Z axis direction
[←], [→] keys: X axis direction
Horizontal machining center (for machine model with vertical X axis)
[↑], [↓] keys: X axis direction
[←], [→] keys: Z axis direction
Horizontal machining center (for machine model with vertical Y axis)
[↑], [↓] keys: Y axis direction
[←], [→] keys: Z axis direction

■ **End adjustment of display position of product CAD model**

(1) Click [Move the position of the product model] in the right-click menu on the 3D graphics area.



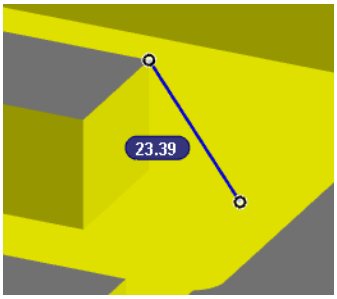
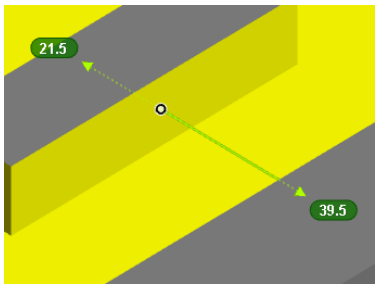
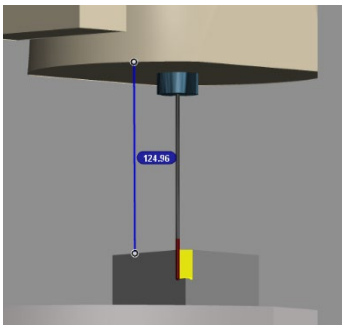
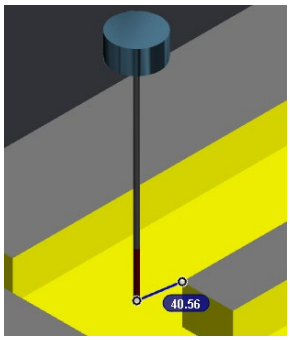
The positioning mode of the product CAD model is ended. "In alignment mode" displayed in the message display area is hidden.



(*)When the [Esc] key is pressed and when a command other than [XY], [YZ], [ZX] and [XYZ] is clicked, adjustment of the display position is ended.

6.3.14 Measure distance on workpiece shape

The smart measurement function allows you to measure the workpiece shape dimension and the distance between workpiece and tool and between workpiece and machine model. (Optional function)
When the workpiece shape dimension is measured, you can measure the length between two points, the angle of edge or arc, the radius of arc, and so on.
The smart measurement function has the measurement mode and the measurement option. For the measurement mode, you specify whether to measure the distance between two points as a straight line or decomposed orthogonal three axes. For the measurement option, you specify whether to enable or disable snap to the vertex and edge. For details, see "4.3.2.4 Machining simulation settings".

Length between two points 	Angle
Arc radius 	Distance in the normal direction 
Measurement between workpiece shape and machine model 	Measurement between workpiece shape and tool 

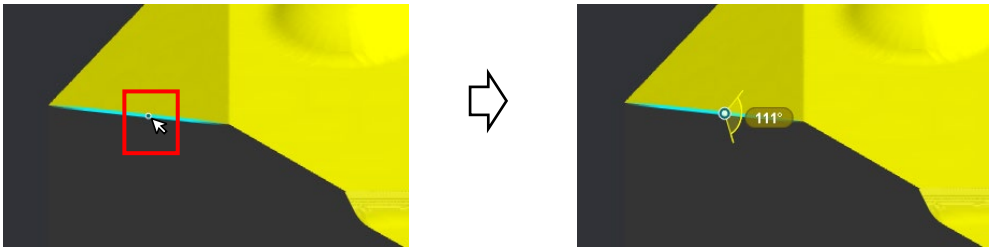
The following are the measurement target and type.

Target	・ Workpiece shape ・ Tool ・ Machine model
Type	・ Measure an angle. (Specify one point at the intersection of surfaces.) ・ Measure an arc radius. (Specify one point on the arc.) ・ Measure a distance in the normal direction. (Specify one point on the surface.) ・ Measure a distance between two points. (Specify the two points on the measurement target.)

■ **Measure the angle/arc/distance in the normal direction**

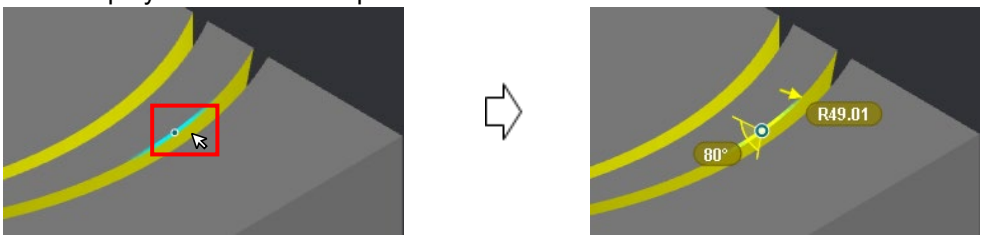
(1) Click the [Smart Measure] command. → The [Smart Measure] command is highlighted.

(2) Move the mouse cursor to the shape to measure the angle and left-click. (*1) → The measurement point (first point) and angle are displayed at the left-clicked position.

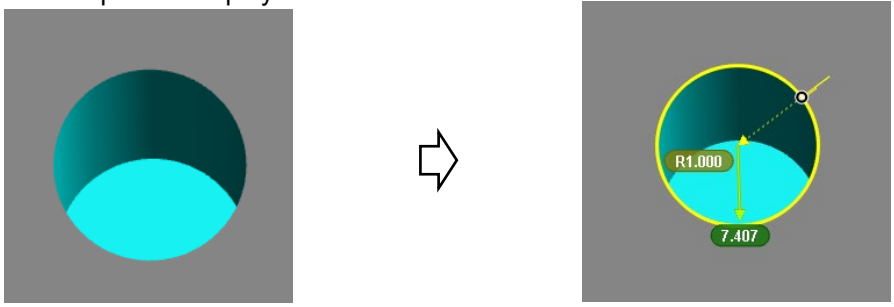


(*) To continue the measurement, move the mouse cursor to the shape you want to measure, and right-click.
(*) To delete the measurement point, left-click on the measurement point or left- and right-click at the same time.

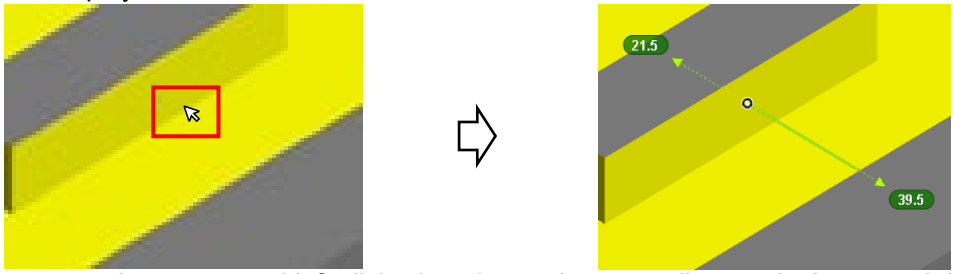
(*1) When you left-click on the shape to measure the arc, the measurement point (first point), radius and angle are displayed at the clicked position.



When you click on the circumference of the hole machined, the measurement point (first point), hole radius and depth are displayed.



When you left-click on the shape to measure the distance in the normal direction, the measurement point (first point) and distance in the normal direction (boundary between workpiece inner surface and space side) are displayed.

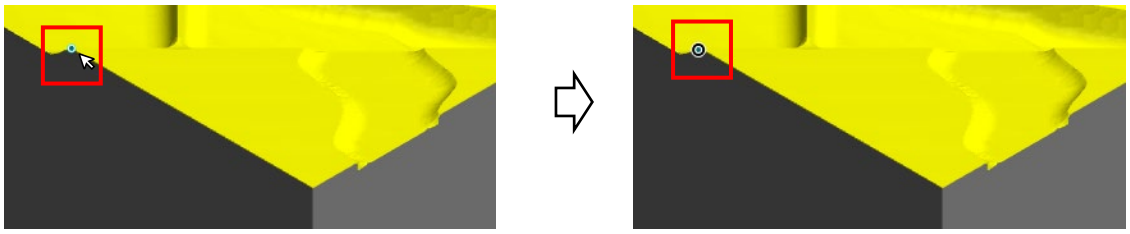


(Note) If you move the mouse and left-click when the angle, arc or distance in the normal direction is already measured (the measurement point is displayed), a distance between two points is measured.

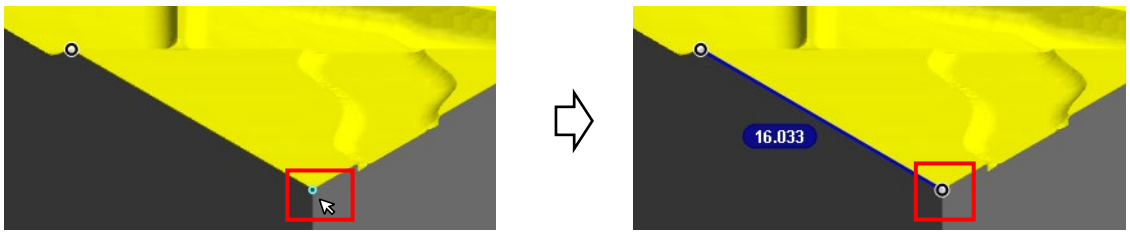
■ **Measure the distance between two points**

(1) Click the [Smart Measure] command. → The [Smart Measure] command is highlighted.

(2) Move the mouse cursor to the measurement start point and left-click. → The measurement point (first point) is displayed at the left-clicked position.

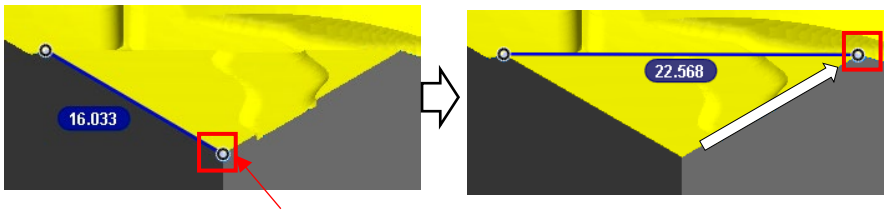
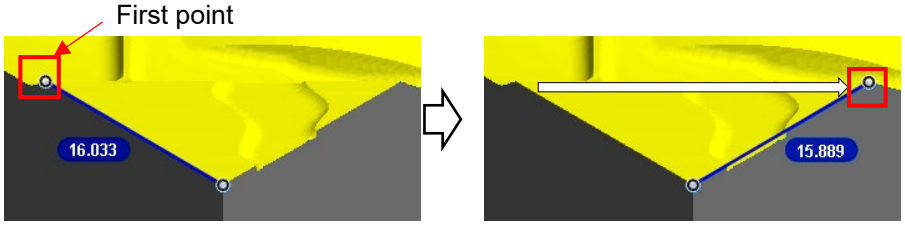


(3) Move the mouse cursor to the measurement end point and left-click. → The measurement point (second point) is displayed at the left-clicked position, and the measured distance is displayed between the measurement points.



(*) To delete the two measurement points, left-click on the measurement point (first point) or left- and right- click at the same time.

(Note) When the distance between two points is already measured, the measurement points can be changed with the following operation.

Operation	Behavior
Left-click	The second measurement point is moved to the clicked position. 
Right-click	The first measurement point is moved to the clicked position. 

6.3.15 Display cross section of workpiece shape

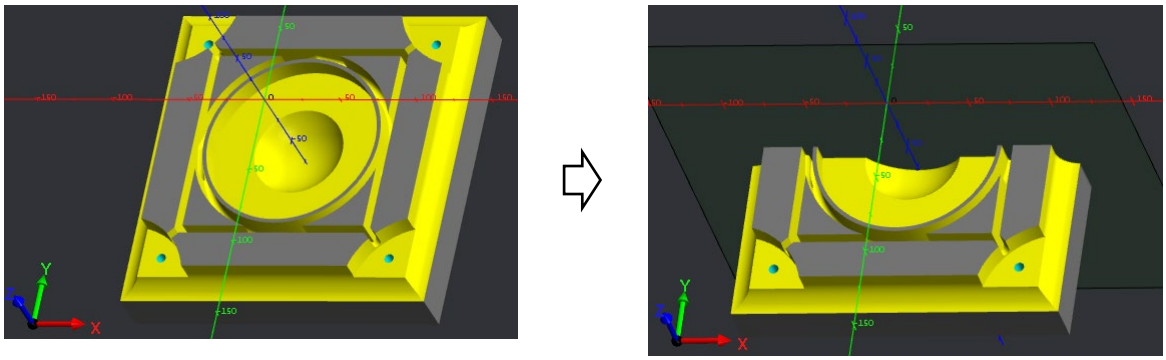
By displaying the cross section of the workpiece shape, the cutting state, etc. of the machining surface can be checked for details. The cross section orthogonal to the I or J axis of the workpiece shape is displayed. Set the display direction of the cross section by the "Machining simulation settings" sub-display.

■ Specify the display direction of the cross section

(1) Click the [Cross-section] command. Or click [Set cross-sectional view position] in the right click menu on the 3D graphics area.

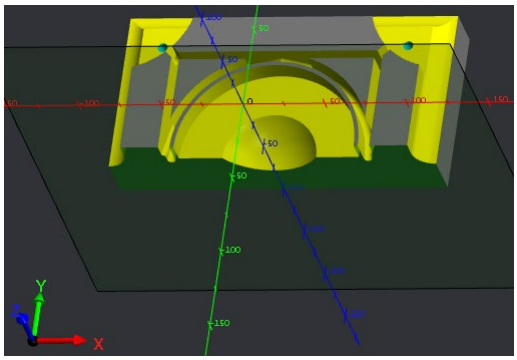
The [Cross-section] command is highlighted, and the plate to adjust the cross section display position is displayed in transparent green. When it is clicked for the first time, the square is displayed in the center of the workpiece shape.

The highlight of the [Zoom], [Move] or [Rotate] command is removed. The workpiece shape can be zoomed, moved and rotated by mouse.



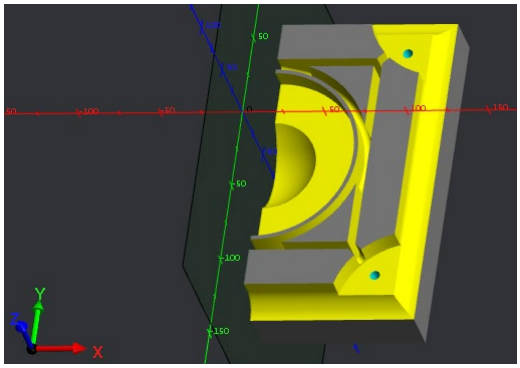
(2) To reverse the direction of the cross-sectional view, check "Reverse cross-sectional direction" in the "Machining simulation settings".

The direction of the cross-sectional view is reversed.



(3) To change the cross-sectional plane, change "Select cross-section" in the "Machining simulation settings".
(Example) Change from XZ plane to YZ plane.

The cross section orthogonal to the workpiece J axis is changed to that to the I axis.



■ Adjust the display position of the cross section

(1) Click any point in the 3D graphics area.

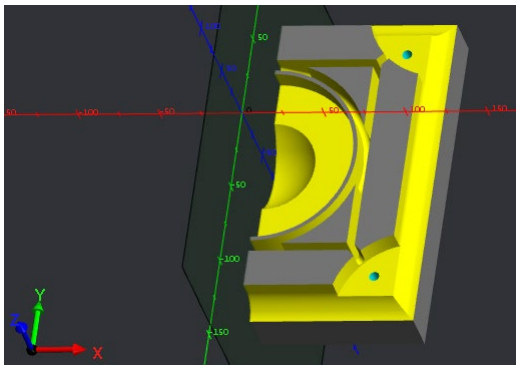
(2) Press the [↑][↓][←][→] key on the keyboard.

The display position of cross section is moved.

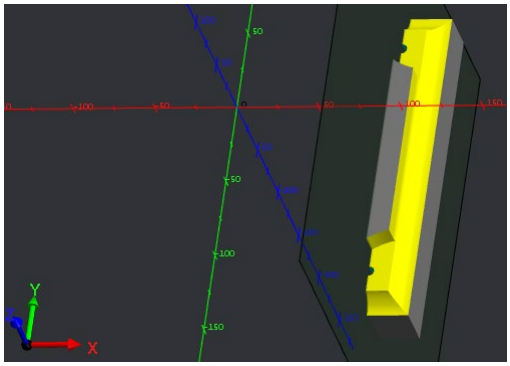
(*) One key press moves the display position by 0.1 mm. Holding down the key moves the display position by the movement amount determined depending on the workpiece size.

(*) When "Select cross-section" in the sub-display "Machining simulation settings" is "YZ", if the displayed point of view is XYZ, the display position is moved in the following direction.
[↓], [→] keys: The cross section is moved to the positive direction of the X axis.
[↑], [←] keys: The cross section is moved to the negative direction of the X axis.

(*) The display position of the cross section cannot be moved to a position which exceeds the workpiece shape size before the simulation is executed.

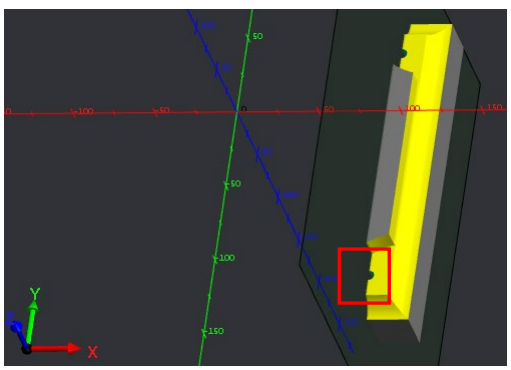


Press the [→] key.

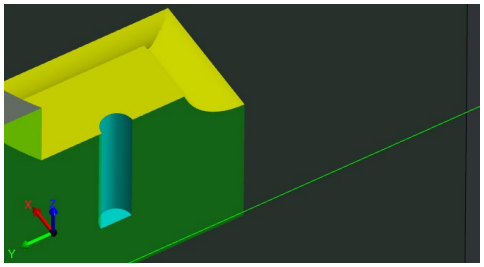


(3) Change the point of view and zoom to adjust the display of the workpiece shape.

Adjust the display of the workpiece shape with the command to change the point of view or by mouse.



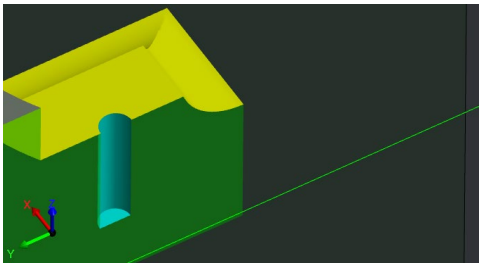
Adjust the display of the workpiece view to the red frame.

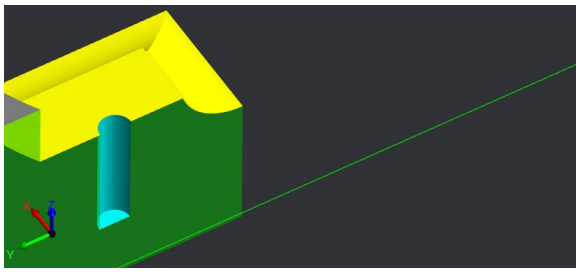


■ Determine the display position of the cross section

(1) After the cross section is displayed and the display position, etc. is adjusted, press the [Enter] key.

The display position of the cross section is determined, and the plate to adjust the display position of the cross section is hidden.
The highlight of the [Cross-section] command is removed.
After the display position of the cross section is determined, other functions can be used while the cross-sectional view is enabled.
When [Cross-section] is clicked again, the display position of the cross section can be changed.



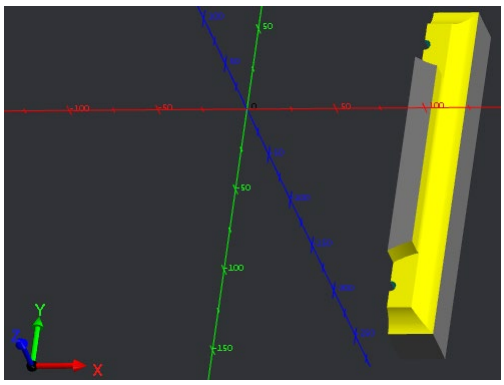


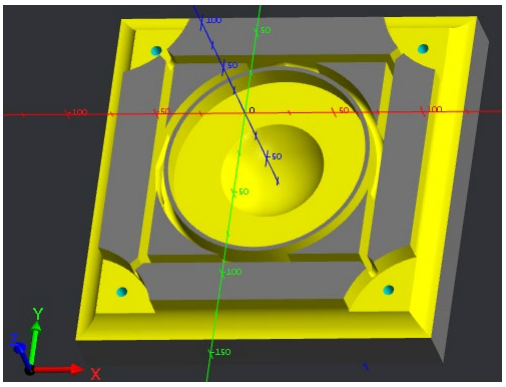
(Note) If the highlight of the [Cross-section] command is removed by other than the [Enter] key, the workpiece shape returns to the state before the cross-sectional view is enabled.

■ Cancel the cross-sectional view

(1) Select [Cancel cross-sectional view] in the right-click menu on the 3D graphics area.

The workpiece shape returns to the state before the cross-sectional view is enabled.

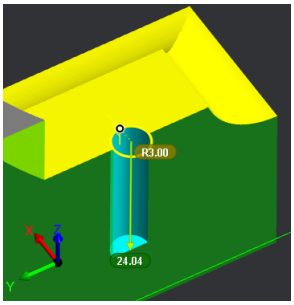




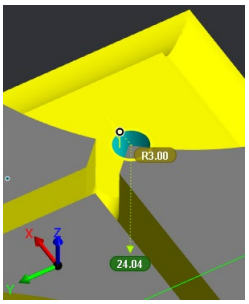
(*) When the [Cross-section] command is clicked again, the movement amount of when the cross-sectional view is enabled last time is taken over to display the cross section.
However, when the [Work Init] command is clicked or when the project is closed, the display position of the cross section returns to the center of the workpiece shape.

When a smart measurement is performed while the cross-sectional view is enabled, the measurement can be performed while the inside of the workpiece is checked.
When the depth of a tapped hole is measured, if the cross-sectional view is enabled, the measurement can be performed while the hole state is checked.

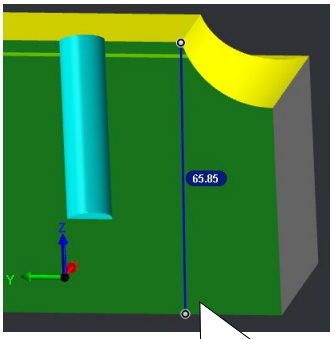
[Cross-sectional view is enabled]



[Cross-sectional view is disabled]

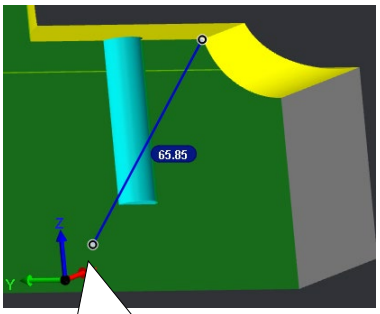


Hidden points of the workpiece shape are also considered as measurement points; therefore, the measurement might not be performed correctly at a point on the cross section.



Measure the height from the edge of the cross section.

Change the point of view.



A length between unintended points is measured.

6.3.16 Display machining shape side by side

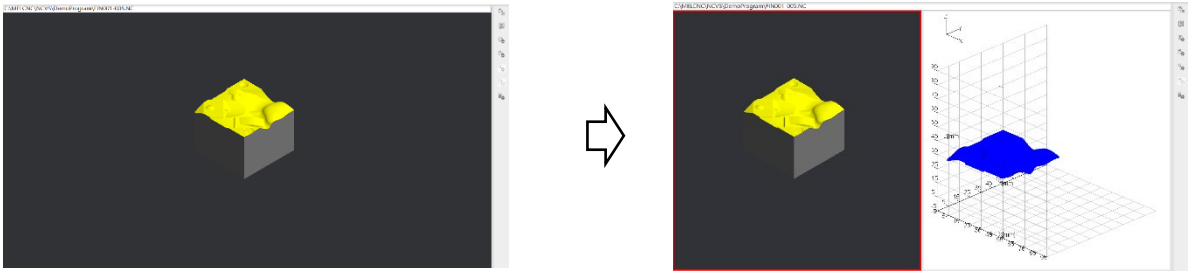
The analysis simulation can be displayed in parallel with the workpiece shape of machining simulation. This makes it easier to compare the analysis simulation and the workpiece shape of machining simulation from various angles.

(1) Click the [Side-by-side] command.

The [Side-by-side] command is highlighted.
The simulation display area is divided into two screens, and following shapes are displayed.

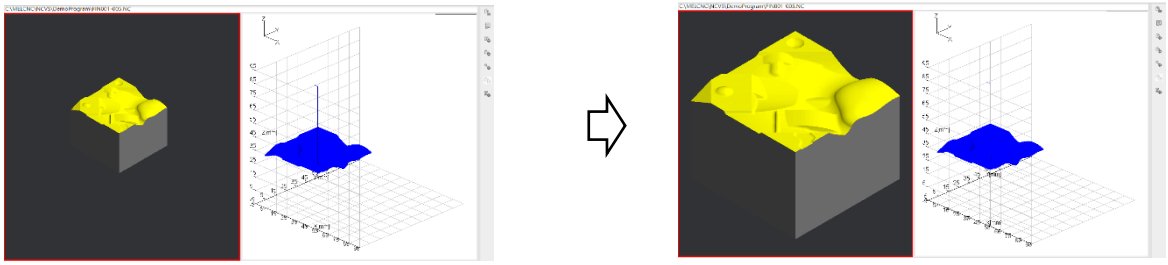
Simulation area (left): workpiece shape
Simulation area (right): machining shape of analysis simulation

The simulation area (left) is selected and enclosed in a red frame.
The sub-display switch button [Analysis simulation settings] is enabled to be selected.



(2) Click the [Auto Scale] command.

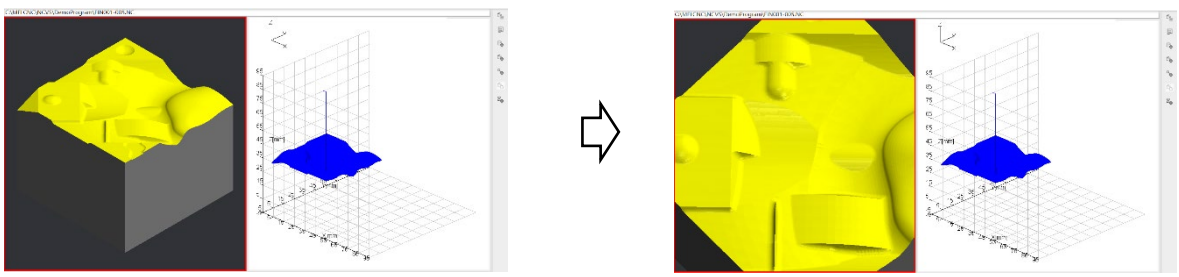
Zoom the workpiece shape in the simulation area (left) so that the size is about 90% of the graphics area.



(3) Adjust the workpiece shape view with the [Zoom], [Move] and [Rotate] commands.

The workpiece shape in the simulation area (left) is displayed according to the command operation.
(Note) When the selected simulation area is not displayed in XYZ (3D), the [Rotate] command is not selectable.

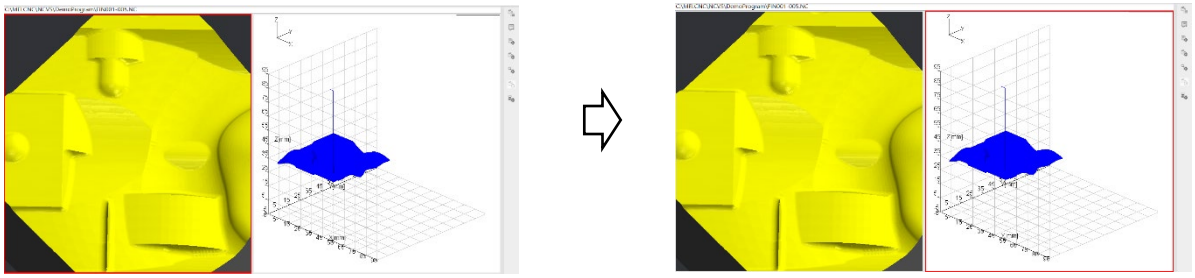
For information about how to adjust the view, see "6.3.9 Adjust the workpiece display position, angle and size".



(4) Left-click the simulation area (right), or press the [Tab] key.

The simulation area (right) is selected, and enclosed in a red frame.

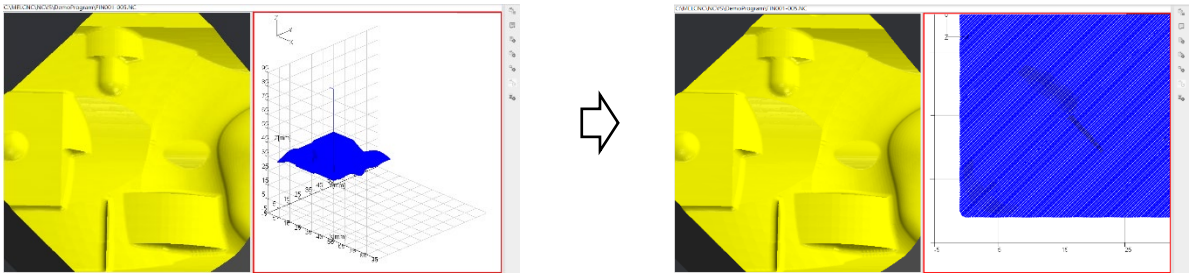
While the simulation area (right) is selected, the [Work Init] and [Machine View] commands are not selectable.



(5) Click the [XY], [YZ], [ZX] or [XYZ] command to change the point of view.
Example: [XY] command

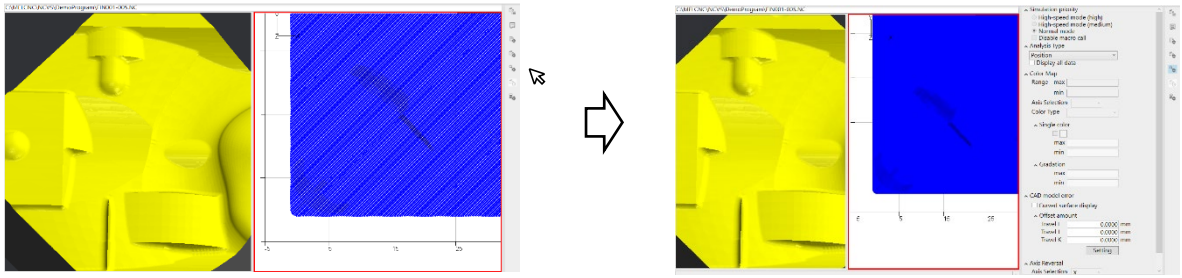
The machining shape of analysis simulation in the simulation area (right) is displayed from the view of the clicked command.

(Note) When the selected simulation area is not displayed in XYZ (3D), the [Rotate] command is not selectable.



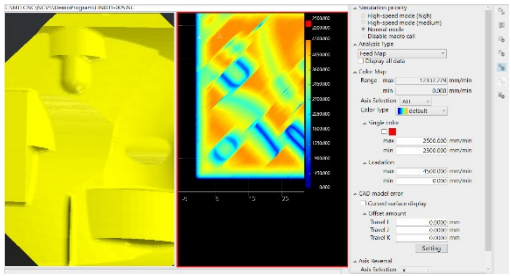
(6) Select [Analysis simulation settings] from the sub-display switch buttons.

The "analysis simulation settings" are displayed in the sub-display.



(7) Select [Feed Map] in Analysis Type, and click [Update].

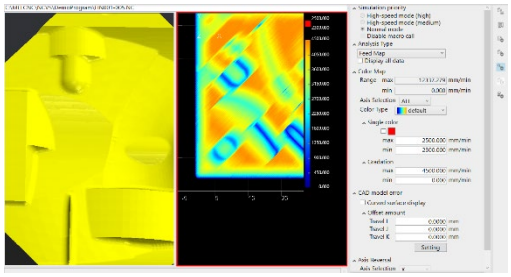
After the progress window appears, the analysis is completed.
The feedrate map is displayed in the simulation area (right).

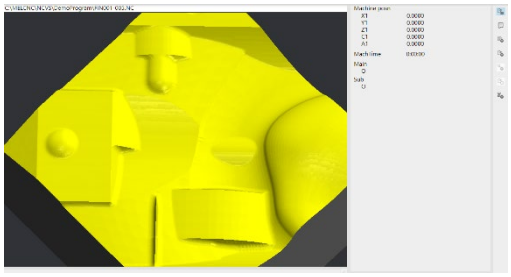


(8) Click the [Side-by-side] command.

The highlight of the [Side-by-side] command is removed.
The workpiece shape displayed in the simulation area (left) is displayed in one screen.

The sub-display switch button [Analysis simulation settings] is disabled to be selected. If Analysis simulation settings is displayed in the sub-display, it is switched to Display processing status.





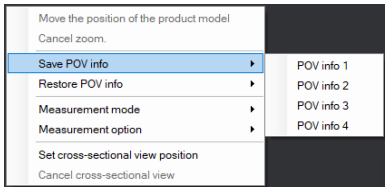
6.3.17 Save/restore the point of view

You can save the point-of-view information such as point of view and zoomed state at a check point of the machining shape, and easily restore the point of view at that time. Up to four points of point-of-view information can be saved.

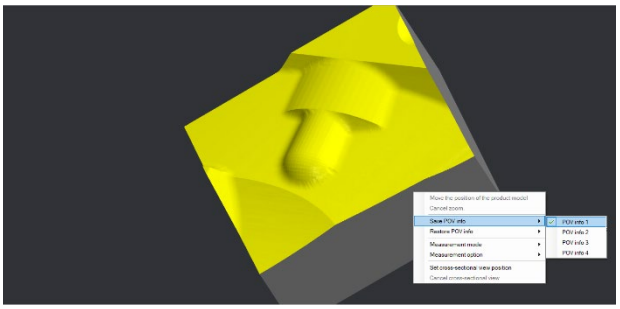
Point-of-view information that can be saved/restored
・ Point of view, zoomed/moved/rotated state ・ Light source state (specular light ON/OFF, light source position) ・ Pinpoint zoomed state ・ Cross-sectional view state (cross-sectional view enabled/disabled, cross-sectional view position, cross-section selection, [Reverse cross-sectional direction] enabled/disabled)

■ **Save the point-of-view-information**

(1) Adjust the point of view of the shape. Select any of [POV info 1] to [POV info 4] of [Save POV info] in the right-click menu on the 3D graphics area.
Example) Select [POV info 1].



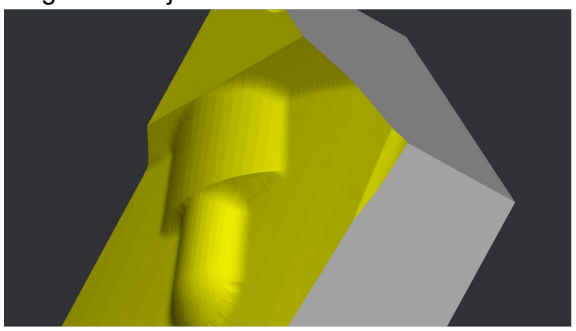
➡ The point-of-view information is saved in the selected [POV info 1].



(*) When a point-of-view information is already saved in [POV info], a check mark is displayed at the left of the menu.
(*) When [POV info] with a check mark is selected, the point of view is overwritten.

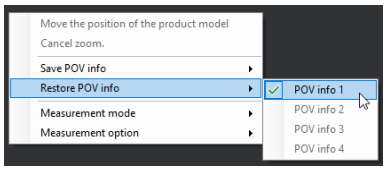
(2) To check another part of the shape, move, zoom and adjust the point of view.

The workpiece shape in the simulation area is displayed according to the adjustment.

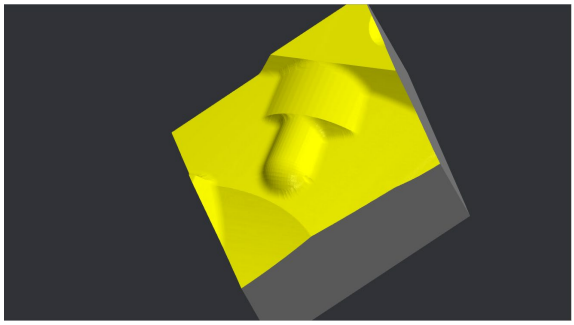


■ **Restore the saved point-of-view information**

(1) Select any of [POV info 1] to [POV info 4] of [Restore POV info] in the right-click menu on the 3D graphics area.
Example) Select [POV info 1].



➡ The workpiece shape returns to be displayed with the saved point of view.



(*) It takes time to restore the information during pinpoint zoom.
(*) The [Restore POV info] menu cannot be selected while simulation is running. (Grayed out)
(*) [POV info] where no information is saved cannot be selected.

6.3.18 Show or hide scale

Show or hide the scale.

(1) Click the icon of the sub-display switch button to open [Machining simulation settings].

(2) The direction of the coordinate system and the scale bars for the view range are displayed by checking the [Display coordinate/scale] checkbox.

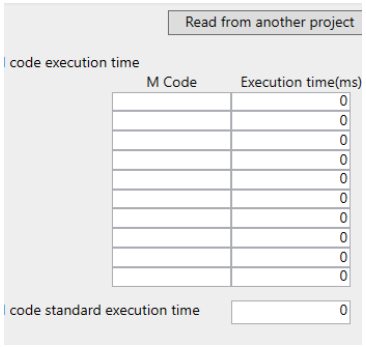
(*) If you click the [Display coordinate/scale] checkbox again, the checkbox is cleared, and the directions of the coordinate system and the scale in the graphics area are hidden.

(Note) The directions of the coordinate system and the scale are not displayed right after NC Virtual Simulator is restarted.

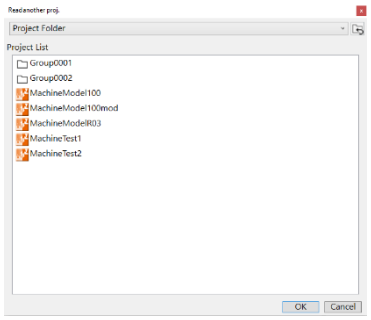
6.3.19 Read machining time prediction setting from another project

You can read the machining time prediction setting from another project and set it in the current project.

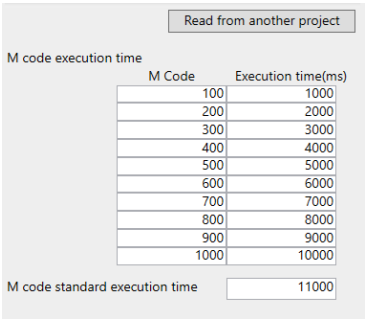
(1) Click [Machining time prediction setting] from the sub-display switch buttons. → The "machining time prediction setting" is displayed in the sub-display.



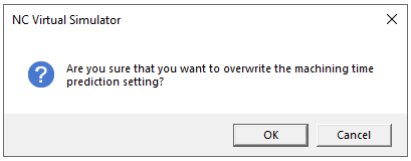
(2) Click the [Read from another project]. → The "Read from another project" dialog appears.



(3) Select a project and click the [OK] button. → The value of another project is set.



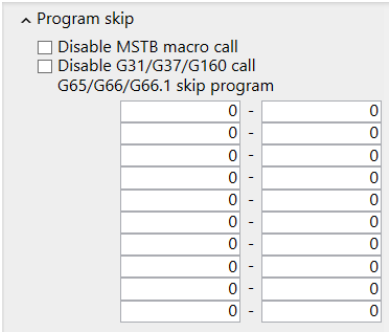
When the machining time prediction setting is already set, the confirmation dialog appears.



6.3.20 Specifying machining programs to be skipped

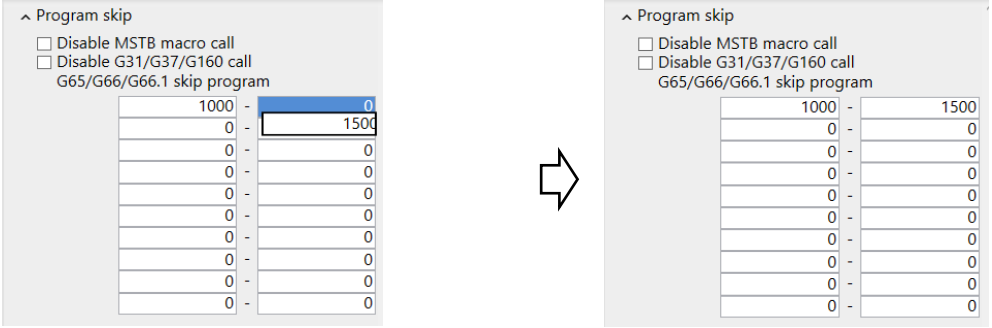
In NC Virtual Simulator, subprograms such as tool measurements prepared by machine manufacturers are simulated differently from the actual machine operation. To exclude program ranges that differ from the actual machine operation, specify the range of the macro programs to skip and run the simulation to the end.
The program skip function cannot be used when NC Virtual Simulator is linked with NC Trainer2 plus.

(1) Click [Machining simulation settings] from the sub-display switch buttons. → [Machining simulation settings] - [Program skip] is displayed in the sub-display.



(2) Set the first number of the range to skip in the left column. (Example) 1000 → The number of the set macro program is displayed.

(3) Set the last number of the range to skip in the right column. (Example) 1500 → The number of the set macro program is displayed.



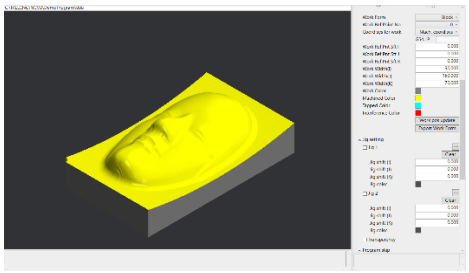
(4) Run the simulation. → The simulation runs while skipping the set macro programs.
(Example) 1000 ≤ Skipped macro programs ≤ 1500

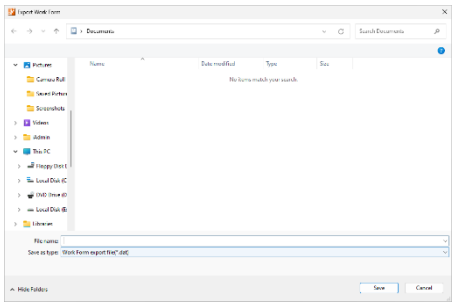
6.3.21 Exporting/importing a workpiece shape

Workpiece shapes with cutting drawings can be exported as DAT format files.
By exporting and importing a workpiece shape that has been completed up until just before the finishing process, you can change the machining conditions and perform only the finishing process repeatedly.

■ Exporting a workpiece shape

(1) Click the [Export Workpiece Shape] button from the "machining simulation settings" sub-display. → The dialog for selecting the workpiece shape export destination is displayed.





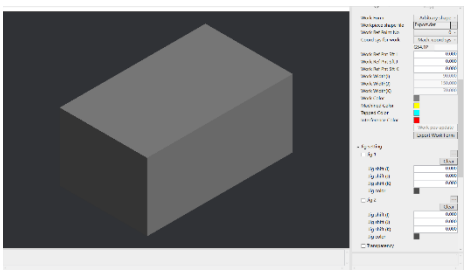
(*1) A message will be displayed when a simulation is running and the dialog will not appear.
(*2) If the width of one side of the workpiece shape before cutting is one of the following, a message is displayed and the dialog will not appear.
- Larger than 99999.999 (mm) (mm system)
- Larger than 9999.999 (inch) (inch system)
- Less than 1.0 (mm) (mm system)
- Less than 0.1 (inch) (inch system)

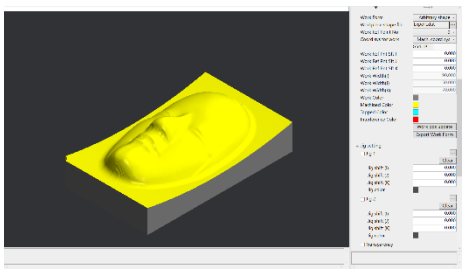
(2) Select the export destination, enter the file name, and click the [Save] button. → A progress window is displayed and the exporting of the workpiece shape begins.
The progress window is hidden when exporting is complete.

■ Importing a workpiece shape

For detailed procedures on importing a workpiece, see "■ Set Work Form" of "6.2.2 Setting workpiece data".

(1) After importing an exported workpiece shape file, click the [Work Init] command. → The workpiece shape is the shape when it was exported.





6.4 Caution

(1) The simulation function uses only the result of the internal operation of the NC. It is not possible to simulate commands that require the movement of the machine.
(2) If an M02 or M30 command is issued while simulation is running, the processing status of the program is canceled; however, the simulation continues until the workpiece cutting is completed.
(3) Even when you zoom, move or rotate the graphic, or switch the simulation mode during simulation, the shape of the workpiece is not initialized, nor is the program stopped.
(4) While simulation is running, the current program (ONB display) in the sub-display "Display processing status" displays the content of the program in automatic operation.
Because what is displayed in the current program in the sub-display "Display program" is the content of the program with which the workpiece is being cut, the update of buffer display may delay.
(5) Among the common variables, only #100 to #199 and #400 to #999 can be set and displayed in NC Data List. It is also not possible to display or set variable names for #500 to #599.
(6) When tool compensation type III is selected (#1037=3/#1046=1), tool offsets are not displayed.
(7) If you attempt to set data in NC Data List while simulation is running (excluding pause in step operation), the message "Simulation is running." appears, and you cannot set data.
(8) It is not possible to select the product CAD model and switch whether the product CAD model is superimposed while simulation is running.
Perform those operations after the simulation is completed or reset.
(9) Robot command (G198) is not supported. When G198 is issued, simulation remains stopped at the block. To end the simulation, click the [Reset] command.
(10) The simulation state may not be updated in the following case: the analysis simulation screen is opened during machining simulation, the simulation is reset, and the machining simulation screen is opened again while the simulation is running. In this case, reset and start the simulation.
(11) When the side-by-side display is performed for the second and subsequent times after the project is opened, the last set analysis type is displayed on the right side of the window. However, when the side-by-side display is performed during simulation while [Posit. Diff Map] and [Feed Diff Map] are set to the right side of the window, the position is displayed on the right side of the window.
(12) When simulation graphic is performed by the [Redraw] button in the "Machining simulation settings", the machine interference check is disabled, and the redraw is performed by using the current workpiece setting and tool setting data. Therefore, when information of workpiece and tool is changed, execute the simulation again.
(13) When simulation graphic is performed by the [Redraw] button in the "Machining simulation settings", the redraw is performed by using the operation mode selected during the simulation.
(14) The machining time prediction setting is not applied to the machining time of simulation graphic by the [Redraw] button in the "Machining simulation settings".
(15) A machining program cannot be selected through the network. Select a machining program on the local disk.
(16) When a project is saved while [Pinpoint zoom invalid] of the "Machining simulation settings" is checked and the workpiece shape in the project is partially enlarged, the workpiece shape in the selected range is displayed in an initialized state (the workpiece shape before the machining is displayed). For pinpoint zoom, uncheck [Pinpoint zoom invalid] of the "Machining simulation settings", and perform simulation graphic by the [Redraw] button, then perform the pinpoint zoom.
(17) If the workpiece shape is too small, it may not be zoomed so that the size is about 90% of the graphics area even by clicking the [Auto Scale] command.

(18) If operations that increase the load on the graphics processor (GPU) (such as enlarge the window size of NC Virtual Simulator, enlarge the machining surface of the workpiece, display the cross section, etc.) are combined, the response may become slower. When the response becomes slower, reduce the window size of NC Virtual Simulator or reduce the machining surface of the workpiece to reduce the load on the graphics processor (GPU).

(19) When simulation is completed in the analysis simulation screen, the status prior to simulation is displayed when the machining simulation screen is opened. To confirm the simulation results in the machining simulation screen, click the [Redraw] button of the sub-display "Machine simulation settings".

(20) When the checkbox of [Disable MSTB macro call] in the sub-display "Machine simulation settings" is checked and a program with MSTB code macros of the G66 and G66.1 macro call type is executed, a "P276 Illegal G67 command" message is displayed at G67 command execution blocks, and the simulation stops. In this case, comment out the G67 command and execute the simulation again.

(21) When the G66 and G66.1 commands are disabled by specifying the program number ranges with the [G65/G66/G66.1 skip program] checkbox of the sub-display "Machine simulation settings", a "P276 Illegal G67 command " message is not displayed at G67 command execution blocks.

(22) When the G66 and G66.1 commands are disabled by specifying the program number ranges with the [G65/G66/G66.1 skip program] checkbox of the sub-display "Machine simulation settings", in some cases an error may not occur at programs where a "P273 Macro call nesting over" occurs when the number of G66 and G66.1 nests exceeds the limit.

(23) When [Machined color] of the tool used in a simulation is changed with the Tool Settings window after executing the simulation, the machined color of the machining shape is displayed in the color changed in the Tool Settings window when returning to the machining simulation screen. Additionally, when tool data used in a simulation is deleted, the machined color is displayed as the color selected in [Machined Color] of the sub-display "Machining simulation settings".

(24) When cutting a workpiece in the same place using tools that have different Tool No. and machining color but the same Tool Type, Length, etc., the second cutting tool does not contact the workpiece and therefore the machining color of the first cutting tool remains.

(25) When importing a workpiece shape file that was exported from a project different to the project to which the workpiece shape is being imported, the machined color may be different from when the file was exported. In this case, read the tool data of the project from which the file was exported.

(26) An exported workpiece shape does not have units. When importing an exported workpiece shape, it is imported at a size that aligns with the units of the project currently opened.

(27) When the machining time prediction setting in the sub-display "Machining time prediction setting" is read from another project, there may be cases where the result is not as expected. This can occur when a project created with a new version of NC Virtual Simulator is read with an older version of NC Virtual Simulator.

(28) When fixed cycle for drilling (G80 to G89) is commanded in normal mode or high-speed mode (medium), the tool may not reach the hole bottom under the following conditions. Therefore, the shape machined in normal mode or high-speed mode (medium) may be different from that in high-speed mode (high).
- The fixed cycle for drilling (G85, etc.) without dwell command at the hole bottom position is executed.
- When the parameter "#1153" is "0" (Perform no deceleration check and in-position check).

(29) When the project name is changed on the screen to open a project, the project folder name and path are changed. When a workpiece shape file is stored in the project folder and this operation is performed, the location of the workpiece shape file is changed. This causes a failure in reading the workpiece shape. In this case, specify the workpiece shape file again.

(30) Tools used in the simulation are those corresponding to the T command immediately before the M06 command, regardless of the setting of the parameter "#1327" (Tool change method specification).

(31) When a product CAD model file, etc. including an invalid shape is read, the application may end. Save the project before reading the file.

7. Analysis simulation

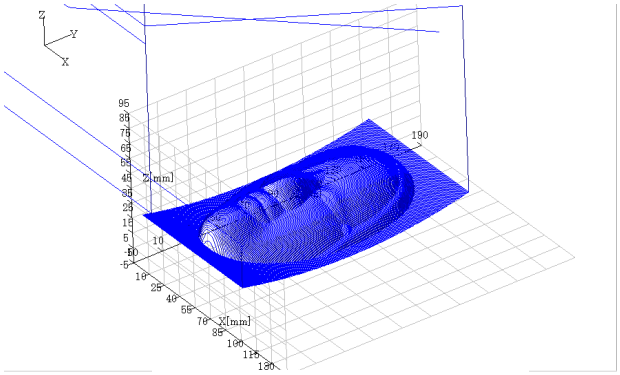
Analysis simulation can be used to identify possible portions of machining faults and causes of machining errors.

The following is types of analysis simulation display.

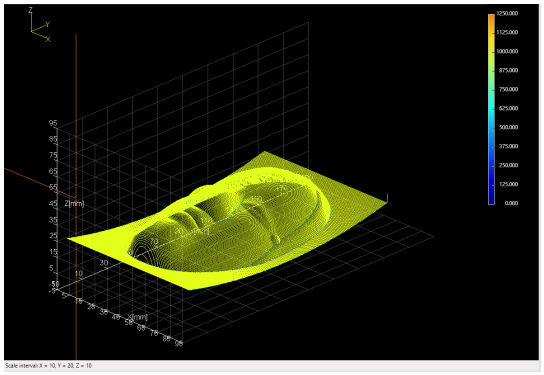
(1) Line segment path display: Display line segments of position data.

(2) Color map display: Visualize the change in feedrate and acceleration rate with colors.

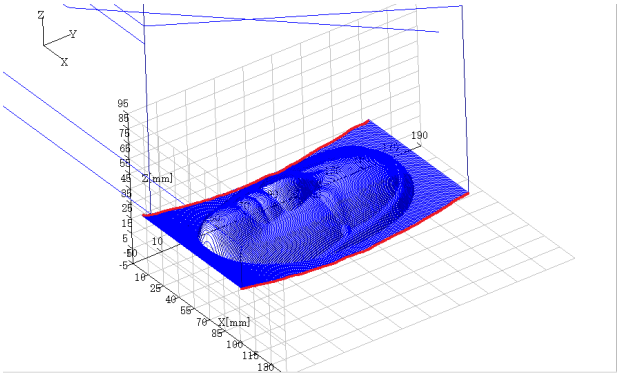
(3) Plot point highlighting: Highlight a specific position such as axis reversal with plot points.



(1) Line segment path display



(2) Color map display



(3) Plot point highlighting

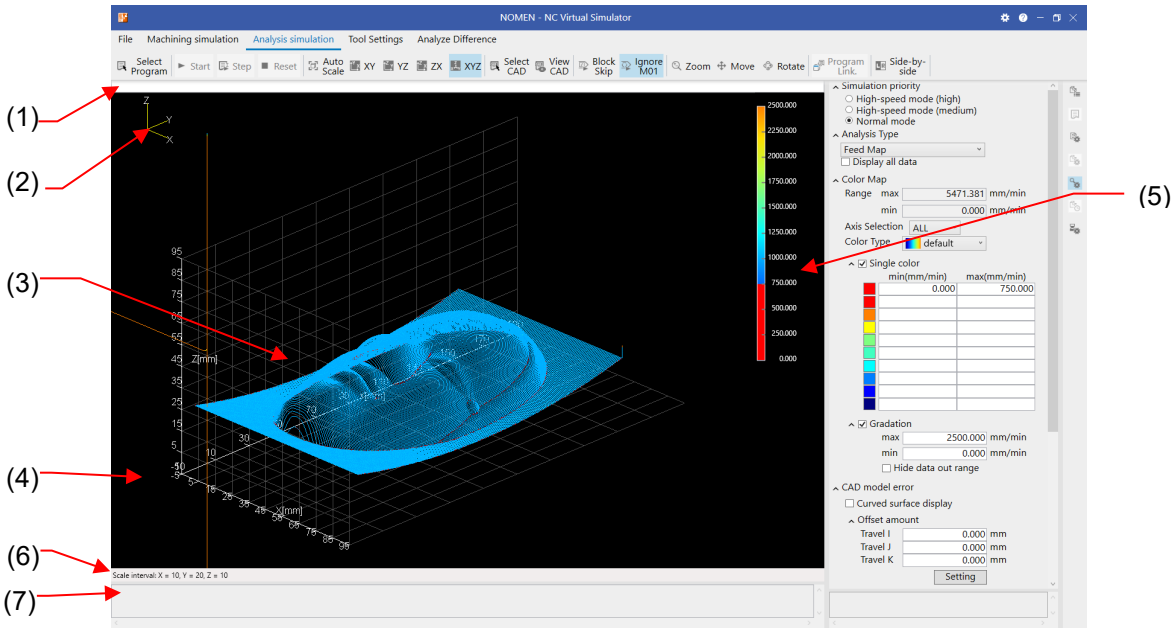
The relationship between "Analysis Type" of the sub-display "Analysis simulation settings" and the display type.

Display type	Analysis type
Line segment path display	Position
Color map display	Feed Map Acceleration Map Posit. Diff Map Feed Diff Map CAD model error
Plot point highlighting	Axis Reversal

7.1 Analysis simulation window

7.1.1 Screen configuration and display items

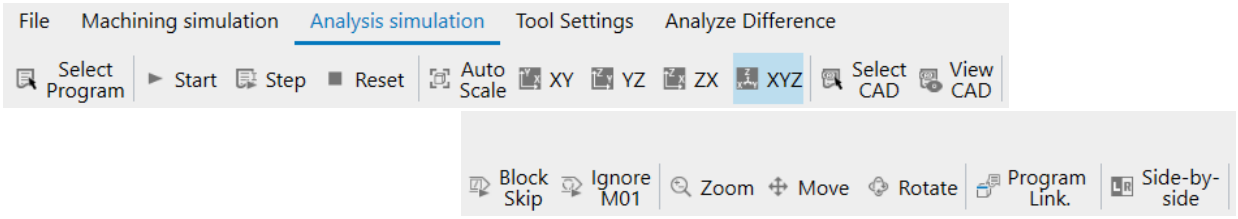
In the analysis simulation window, you can view the analysis display of the currently selected machining program.



Item		Description
(1)	Program path	Displays the file path of the selected machining program.
(2)	Coordinate axes	Displays the coordinate axes X, Y, Z in the upper-left corner of the graphics area.
(3)	Graphics area	Displays the machining shape based on the position data obtained from the simulation of the selected machining program. You can zoom, rotate, and move the machining shape.
(4)	Scale	The scales for base axes I, J and K are displayed around the machining shape. The tick marks for each axis divide the workpiece width and extra value into 10 to 1000 segments.
(5)	Gradation bar display	This item is displayed when color map display is selected as the analysis type. The relationship between display color and value is displayed in the range between the maximum and minimum values of gradation and single color set in the sub-display "Analysis simulation settings" with a gradation bar.
(6)	Status bar	Displays the drawing area information.
	Scale interval (*1)	Displays the scale interval of the scale display.
(7)	Message area	Displays the status during simulation and errors that occurred.

(*1) For side-by-side display, the scale interval is displayed on the left screen only.

7.1.2 Analysis simulation commands



The following lists the commands that can be used in analysis simulation.

	Command	Description	Reference
1	Select Program	Select a program. Choose a program to simulate from the file selection dialog that pops up.	6.3.5 Select a program
2	Start	Start simulation. This command is highlighted while simulation is running. If you click this command when it is highlighted, simulation is paused at the end point of the current block.	6.3.6 Start simulation
3	Step	Run the program block by block.	6.3.6 Start simulation
4	Reset	Stop the ongoing simulation.	6.3.7 Stop simulation
5	Auto Scale	Zoom the workpiece so that the size is about 90% of the graphics area.	-
6	XY	Change the plane where the machining shape is displayed or change the display to 3D. The scale in the plane selected before switch is retained.	6.3.8 Change the point of view on workpiece
7	YZ		
8	ZX		
9	XYZ		
10	Select CAD	When a product CAD model is selected from the file selection dialog, the product CAD model is displayed on the workpiece reference point. Files in STEP format (*.stp or *.step) can be selected. This command cannot be selected while simulation is running.	6.3.12 Superimpose product CAD model on workpiece shape
11	View CAD	Toggles between showing and hiding the product CAD model. This command cannot be selected while simulation is running.	6.3.12 Superimpose product CAD model on workpiece shape
12	Block Skip (*1)	Select whether to execute blocks containing "/" (slash) in the program to be simulated. When optional block skip is ON, this command is highlighted. ON: Simulation is run by skipping blocks with "/". OFF: Simulation is run including blocks with "/". The highlight state of this command is the same as that before the application is started and that before the project is selected again.	-

	Command	Description	Reference
13	Ignore M01 (*1)	Specify whether to stop the program when an M01 command is issued. When the Ignore M01 is ON, this command is highlighted. ON: Simulation is not stopped even when an M01 command is issued. OFF: Simulation is stopped when an M01 command is issued. The highlight state of this command is the same as that before the application is started and that before the project is selected again.	-
14	Zoom (*1)(*2)	Zoom the displayed machining shape. When zooming, the scale interval of the scale display is adjusted automatically. • With this command highlighted, press the [↑] [↓] keys on the keyboard, or left-click and drag up and down in the 3D graphics area, to zoom in and out of the displayed machining shape. • If you rotate the mouse wheel with the mouse pointer in the 3D graphics area, the displayed machining shape is zoomed in or out regardless of whether this command is highlighted.	6.3.9 Adjust the workpiece display position, angle and size
15	Move (*1)(*2)	Move the displayed machining shape. • With this command highlighted, press the [↑] [↓] [←] [→] keys on the keyboard, or left-click and drag in the 3D graphics area, to move the displayed machining shape.	
16	Rotate (*1)(*2)	Rotate the displayed machining shape. • With this command highlighted, press the [↑] [↓] [←] [→] keys on the keyboard, or left-click and drag in the 3D graphics area, to rotate the displayed machining shape. • If you right-click and drag in the 3D graphics area, the displayed machining shape is rotated regardless of whether this command is highlighted.	
17	Program Link.	Show which block in the machining program corresponds to the line segments selected in the displayed machining shape and vice versa. • If you click a line segment in the machining shape, the line segment is highlighted in red, and the corresponding block in the machining program in the sub-display is highlighted. • While Program Link is enabled, you can zoom, move or rotate the workpiece.	7.3.1 Enable program link
18	Side-by-side	Divide the simulation display area to display the displayed machining shape and a machining shape in a different analysis type side by side. When two screens are displayed, this command is highlighted. When this command is clicked again, only the machining shape displayed in the left side is displayed, and the highlight of this command is removed.	7.3.2 Display machining shape side by side

(*1) The highlight indicates that the command is enabled. Every time the command is clicked, it is switched between highlighted and released (enabled and disabled).

(*2) The [Zoom], [Move] and [Rotate] commands cannot be enabled at the same time. When one of them is enabled, the highlight of the other command is removed.

(Note 1) While simulation is running or paused, the [Select Program] command is not selectable.

(Note 2) If no machining program is selected, the [Start], [Step] and [Reset] commands are not selectable.

(Note 3) All commands are highlighted when you left-click the command when it is not highlighted.

(Note 4) When the [XY], [YZ] or [ZX] command is selected, the [Rotate] command is not selectable.

(Note 5) When [Posit. Diff Map], [Feed Diff Map] or [CAD model error] is selected in "Analysis Type", the [Start], [Step] and [Reset] commands are not selectable.

7.2 Preparation

To perform analysis simulation, first set the data of the tool and workpiece to be used in simulation.
For detailed information about the preparation, see "6.2 Preparation".
(*) The data of the tool and workpiece is common to "6 Machining simulation". When it is already set in Machining simulation, it does not need to be set in Analysis simulation.

7.3 Operation

Click [Analysis simulation] in the function menu.

(*) Simulation is run based on the simulation priority and analysis type that were selected in the sub-display "Analysis simulation settings" before simulation is started.

The following describes how to run analysis simulation.

(1) Select "Analysis Type".
The following are the analysis types that can be selected.
- Position
- Feed Map
- Acceleration Map
- Axis reversal
- Post. Diff Map (*)
- Feed Diff Map (*)
- CAD model error (*)

(*) When they are selected after the simulation and [Update] is clicked, analysis is performed and the result is displayed. (The [Start], [Step] and [Reset] commands cannot be selected.)

(2) How to run analysis simulation and how to adjust workpiece display position are the same as those in machining simulation.
See the following chapters.
"6.3.4 Change the simulation priority"
"6.3.5 Select a program"
"6.3.6 Start simulation"
"6.3.7 Stop simulation"
"6.3.9 Adjust the workpiece display position, angle and size"

(3) To continue analysis after "Analysis Type" is changed, run simulation by following the steps in the chapters below.
- When [Position] is selected
  "7.3.3 Turn to line segment path (position)"

- When [Axis reversal] is selected
  "7.3.6 Turn to plot point highlighting (axis reversal)"

- When [Feed Map], [Acceleration Map], [Post. Diff Map], [Feed Diff Map] or [CAD model error] is selected
  "7.3.4 Turn to color map display"

(4) For the other functions useful for analysis, see the following chapters.

- Find a machining program block corresponding to the line segment in the machining shape.
  "7.3.1 Enable program link"

- Compare simulation results in different analysis types.
  "7.3.2 Display machining shape side by side"

- Compare simulation results with CAD model.
  "7.3.5 Display CAD model error"

7.3.1 Enable program link

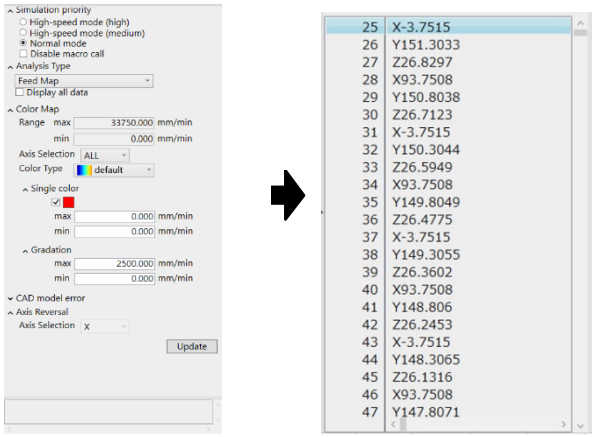
If you click a line segment in the machining shape, the line segment is highlighted in red, and the corresponding block in the machining program is highlighted.
Conversely, if you select a block in the machining program, the line segments in the machining shape corresponding to the block is highlighted.

■ Select a line segment in the machining shape

(1) Select the [Program Link.] command.

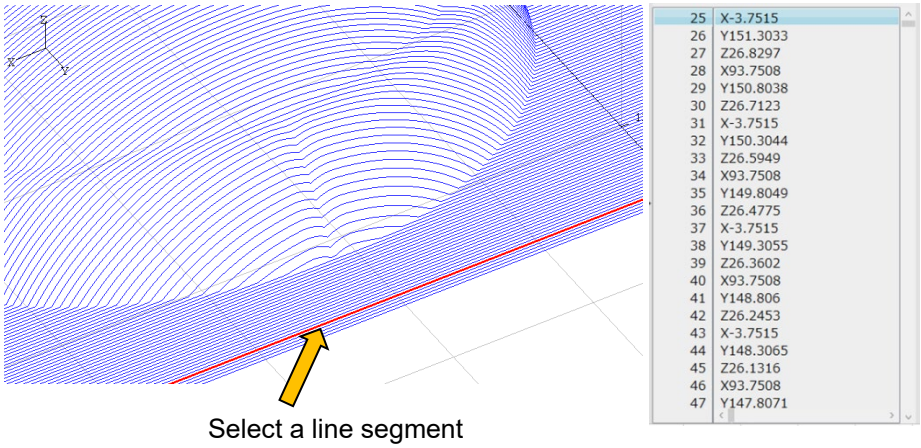
The [Program Link.] command is highlighted.
The sub switch button [Display program] is selected, and the machining program appears in the sub-display.

If Analysis simulation settings is displayed in the sub-display, it is automatically switched to the machining program.



(2) Select a line segment in the machining shape.

The line segment you selected is highlighted, and the corresponding block in the machining program is highlighted.
The line segment for one block in the machining program is highlighted.



Select a line segment

(*) Even if you switch the sub-display to a display other than Machining simulation settings during Program link, Program link remains enabled.

(*) When the selected line segment in the machining shape corresponds to a block in the main program, the main program is displayed in the program display sub-display. When the selected line segment in the machining shape corresponds to a block in the subprogram, the subprogram is displayed. You cannot switch between the main program and subprogram through operation in the program display sub-display.

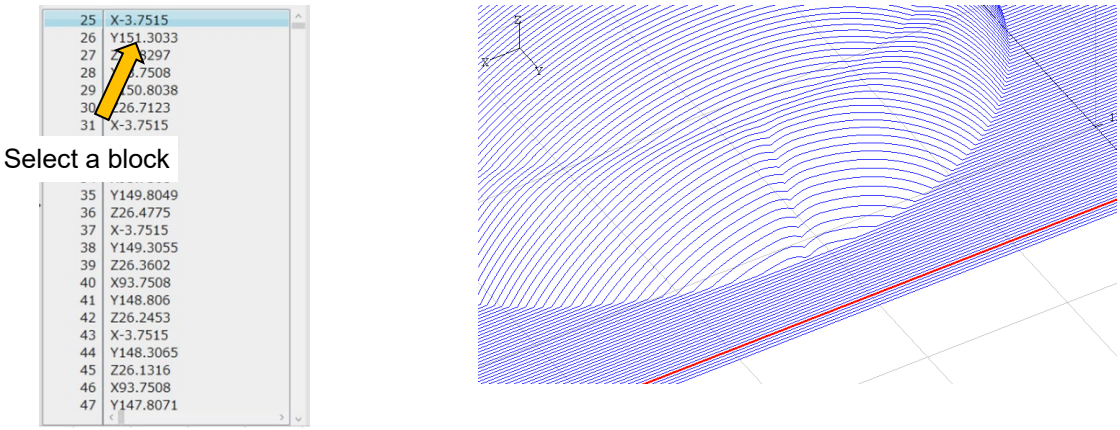
(*) Scrolling using the vertical scroll bar or by the cursor movement is only available within the displayed program.

(*) In a micro segment program, due to block superimposition control of NC, etc., the commanded block may not correspond to the highlighted segment and the segment may be divided to correspond to the previous and later blocks.

■ **Select a block in the machining program**

(1) Select the [Program Link.] command. → The [Program Link.] command is highlighted.
The sub switch button [Display program] is selected, and the machining program appears in the sub-display.

(2) Select a block in the machining program. → The line segments in the machining shape that correspond to the selected block are all highlighted.



(*) If the block you selected in Program Link. is executed repeatedly by means of GOTO, etc., the line segments corresponding to the block executed first are highlighted.

7.3.2 Display machining shape side by side

The displayed machining shape and a machining shape in a different analysis type are displayed side by side.

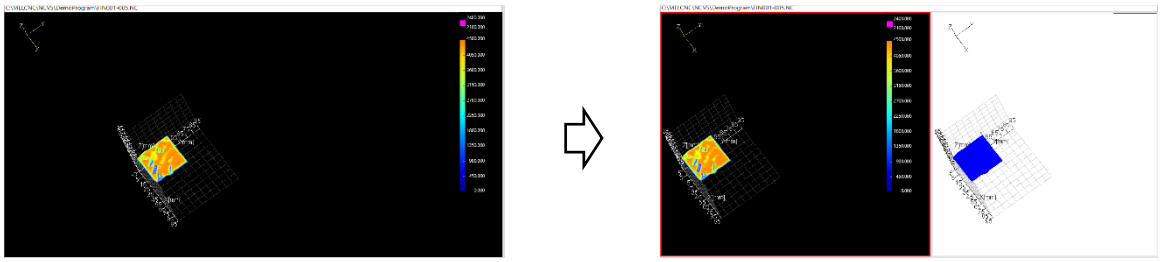
(1) Click the [Side-by-side] command.

The [Side-by-side] command is highlighted.
The simulation display area is divided into two screens, and when the side-by-side display is performed for the first time after the application is started, the following shapes are displayed.

Simulation area (left): the displayed machining shape
Simulation area (right): the displayed machining shape is displayed by "Position". (*1)

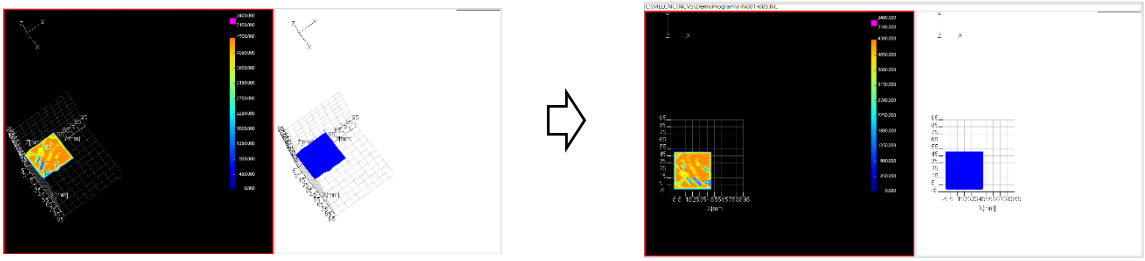
The simulation area (left) is selected and enclosed in a red frame.

(*2) When "Position" is displayed in the simulation area (left), the feed map is displayed in the simulation area (right).

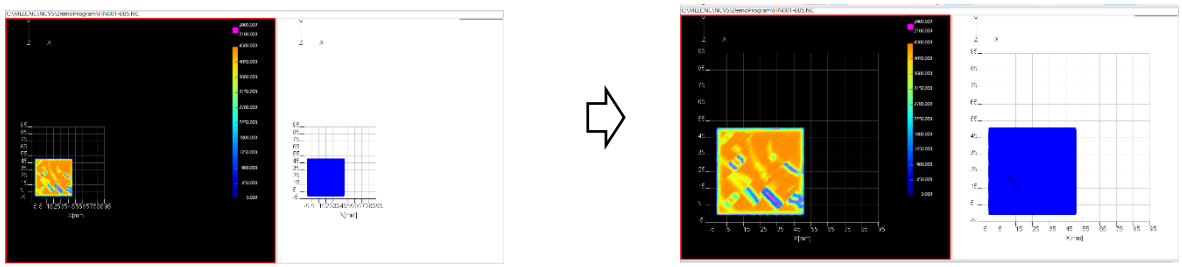


(2) Click the [XY], [YZ], [ZX] or [XYZ] command according to the view direction.
Example: [XY] command

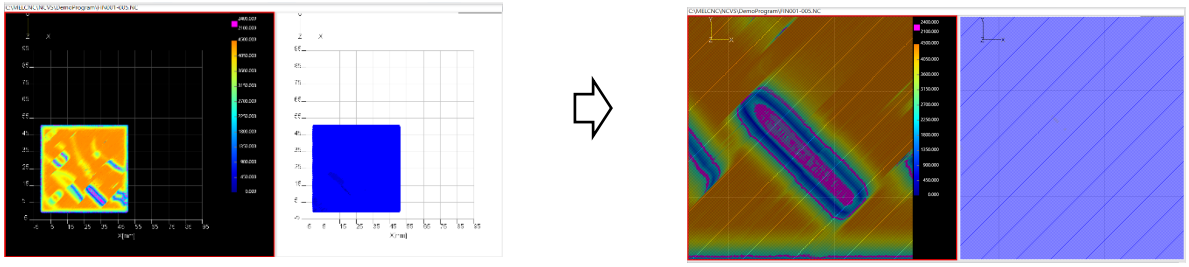
The machining shapes are displayed from the view of the clicked command on both screens.



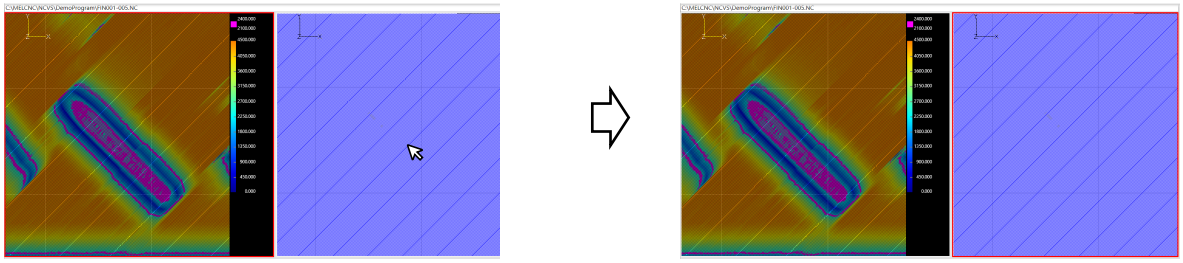
(3) Click the [Auto Scale] command. ➡ Zoom both screens so that the scale is about 90% of the graphics area.



(4) Adjust the machining shape view with the [Zoom], [Move] and [Rotate] commands. ➡ The machining shapes are displayed according to the command operation on both screens. For information about how to adjust the view, see "6.3.9 Adjust the workpiece display position, angle and size".



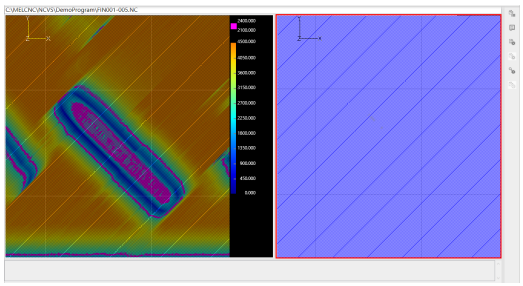
(5) Left-click the simulation area (right), or press the [Tab] key. ➡ The simulation area (right) is selected, and enclosed in a red frame.

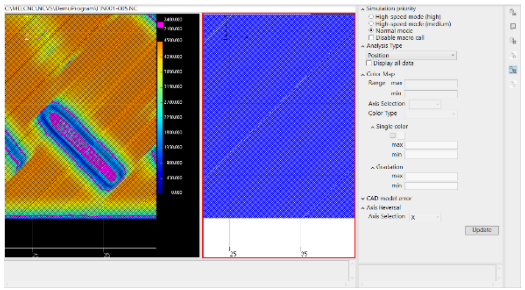


(6) Select [Analysis simulation settings] from the sub-display switch buttons.

The "analysis simulation settings" are displayed in the sub-display.
For each item, the setting values of analysis type displayed in the simulation area (right) are displayed.

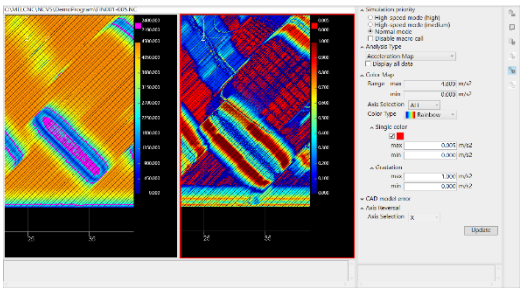
For each item of the Analysis simulation settings, the setting values of the selected simulation area (enclosed in a red frame) are displayed.





(7) Select [Acceleration Map] in [Analysis Type], and click [Update].

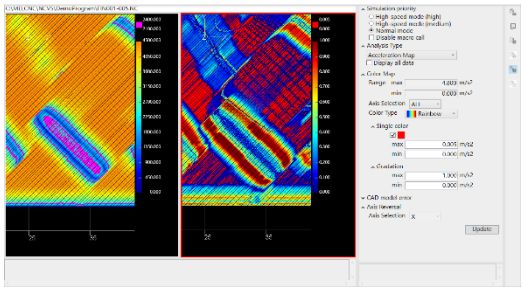
After the progress window appears, the analysis is completed.
The acceleration map is displayed in the simulation area (right).

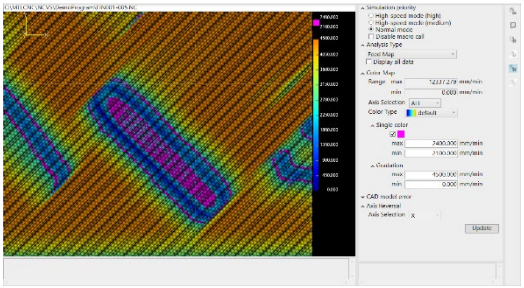


(8) Click the [Side-by-side] command.

The highlight of the [Side-by-side] command is removed.
The machining shape displayed in the simulation area (left) is displayed in one screen.

When the Analysis simulation settings is displayed in the sub-display, the setting contents of the analysis type displayed in the simulation area (left) are displayed.



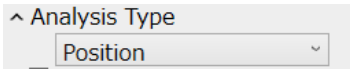


7.3.3 Turn to line segment path (position)

When [Position] is selected for "Analysis Type", the display turns to the line segment path display.

(1) Select [Position].

[Position] is displayed under Analysis Type.

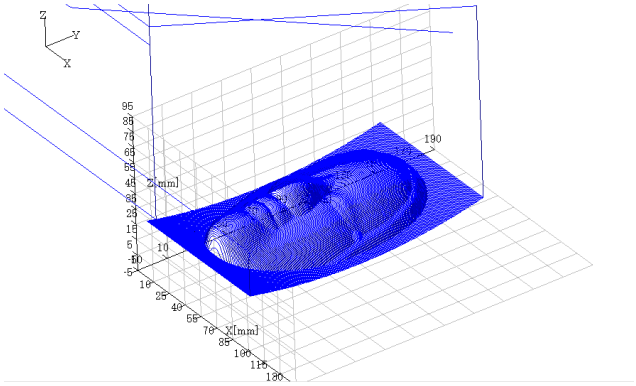


The values of [Color Map] range min and max are displayed as blank and nonconfigurable.

(2) Click [Update].

The progress window appears during the process.

When the process is completed, the machining shape is displayed with blue lines.
The scales of X, Y, and Z axes are displayed around the machining shape.
The tick marks of the scale for each axis divide the range between the largest and smallest values of the workpiece into ten segments.
Gradation bar is not displayed.



■ Change the point of view

You can change the point of view of the line segment path display.
To change the point of view, use the [XYZ], [XY], [YZ], and [ZX] commands.
For details, see "6.3.8 Change the point of view on workpiece".

■ Select axis to display in [Show Scale]

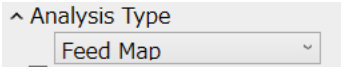
The axis displayed in [Show Scale] of the drawing area can be changed.
For details, see "4.3.2.5 Analysis simulation settings".

7.3.4 Turn to color map display

When [Feed Map], [Acceleration Map], [Post. Diff Map], [Feed Diff Map] or [CAD model error] is selected, the machining shape can be displayed with the analyzed value in a color map.

(1) Select an analysis type from [Analysis Type].

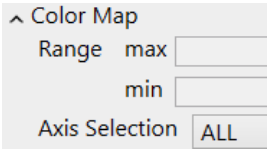
The selected type is displayed in [Analysis Type].
This is an example when [Feed Map] is selected.



For the min and max of the range for [Color map], the maximum and minimum values of each analysis result are displayed. For the min and max of [Single color] and [Gradation], the values last input are displayed.
When displayed for the first time, min and max are blank.
[Axis Selection] under [Axis Reversal] is not selectable.

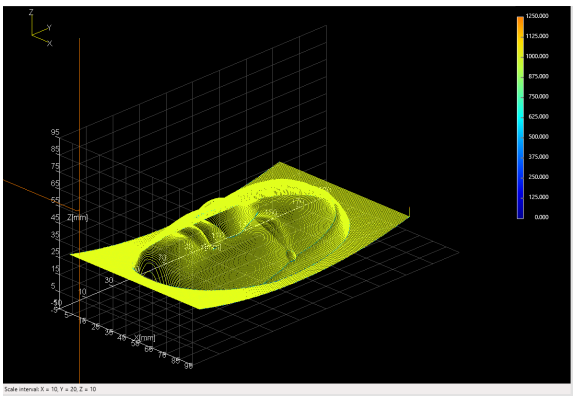
(2) Set an analysis axis in [Axis Selection].

The selected axis is displayed in [Axis Selection].
This is an example when [ALL] is selected.
When [ALL] is selected, the feed map of composition speed is displayed.



(3) Click [Update].

The progress window appears during the process.
When the process is completed, the machining shape is displayed in a color map.
The gradation bar is displayed on the right side of the machining shape.
The gradation bar is displayed with a scale that divides the combined range of [Gradation] and [Single color] from the minimum to the maximum value into ten segments.
In the gradation bar, the minimum value of [Gradation] is displayed in blue, and the maximum value in orange. In addition to this, the color set in [Single color] is displayed.



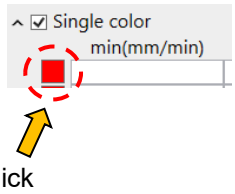
■ **Change the setting (color and range) of [Single color]**

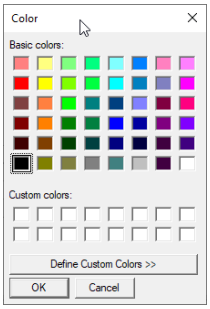
The path where the analysis type data is in the specified range can be displayed in [Single color].

Up to 10 [Single color] settings can be set.

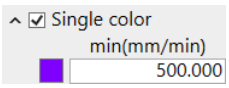
(1) Check the checkbox for "Single color".

(2) To change the color setting, click the color box of "Single color". → The Color dialog appears.

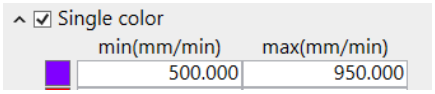




(3) Choose a color in the "Color" dialog, and click the [OK] button. → "Single color" is displayed in the color you selected.



(4) Set the range (maximum and minimum values) of the analysis data to be displayed in [Single color]. → The path where the analysis data is in the specified range is displayed in [Single color].

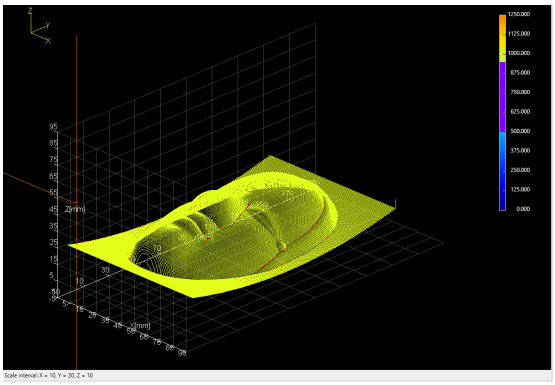


The following values cannot be set: values that are out of the range between the minimum and maximum values of the color map, and values that the minimum value is greater than the maximum value.

(5) Click [Update].

The progress window appears during the process.
When the process is completed, the path where the analysis data is in the specified range is displayed in single color.

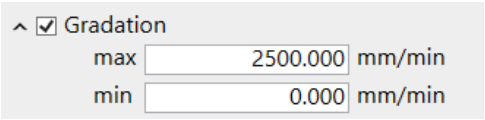
(*) When the [Gradation] checkbox is not checked, only the analysis data for the range specified by [Single color] is displayed.
(*) When the [Gradation] checkbox is checked, the [Single color] setting will be given priority and displayed for the display of areas that overlap with the gradation range.
(*) When the ranges specified by multiple [Single color] overlap, the [Single color] entered in the lower side of the range will be given priority and displayed



■ **Change the range for Gradation**

(1) Check the [Gradation] checkbox and set the range (maximum and minimum values) of the analysis data to be displayed in gradation.

The gradation bar is displayed within the range between the min and max values you set.

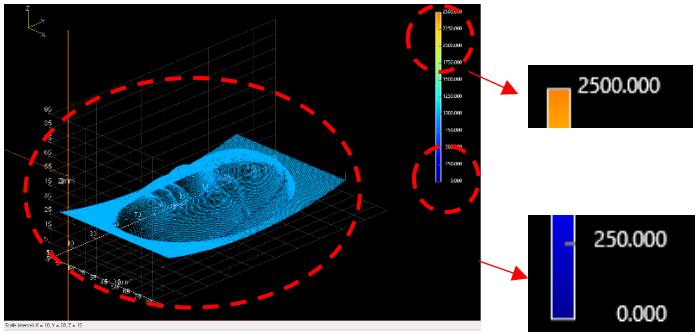


The following values cannot be set: values that are out of the range between the minimum and maximum values of the color map, and values that the minimum value is greater than the maximum value.

(2) Click [Update].

The progress window appears during the process.
When the process is completed, the machining shape is displayed in gradation.

When the [Hide data out range] checkbox is checked, the machining shape is not displayed outside of the gradation range.
When the [Hide data out range] checkbox is not checked, the machining shape is displayed in blue for the data below the gradation minimum value, and displayed in orange for the data above the maximum value.

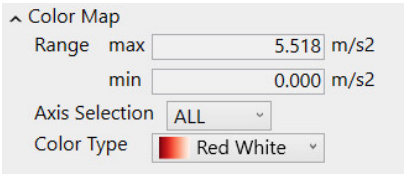


■ Change the color map type

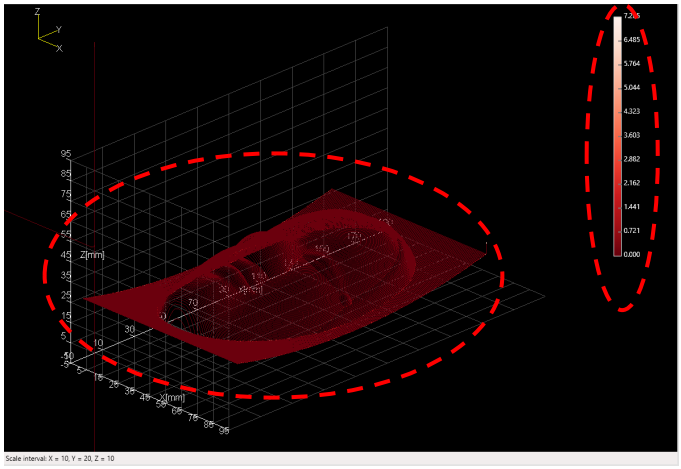
You can change the gradation color of the color map by [Color Type].

Name	min Image max	Description
default		Color map to be displayed in gradation of blue, green, yellow and orange
Rainbow		Color map to be displayed in gradation of blue, green, yellow, orange and red
Blue Red		Color map to be displayed in gradation of blue, white and red
Red White		Color map to be displayed in gradation of red and white

(1) Change the color map type. → The selected item is displayed in "Color Type".



(2) Check the [Gradation] checkbox and click [Update]. → The progress window appears during the process. When the process is completed, the machining shape is displayed in the colors of the color map type. The gradation bar is displayed with the selected color map type.



7.3.5 Display CAD model error

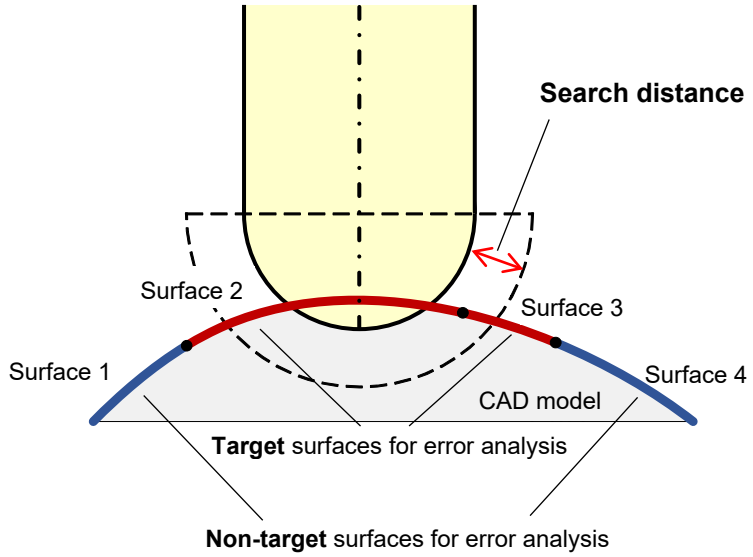
When [Analysis Type] is [CAD model error], the difference (error amount) between the product CAD model surface and the tool path is displayed in a color map. This allows you to find a portion to be excessively shaved or incompletely shaved with respect to the product CAD model.

■ **Target tools for error amount calculation**

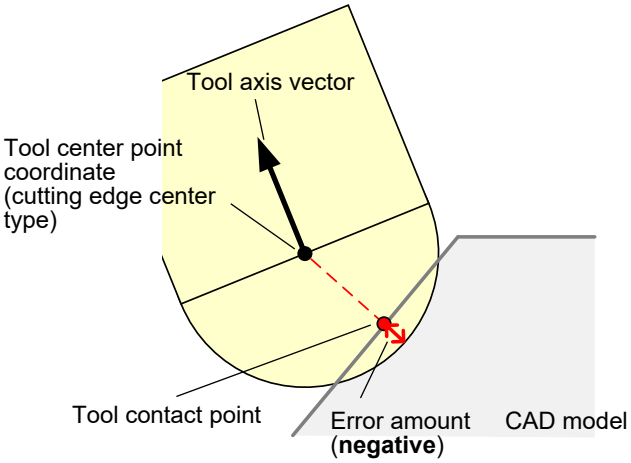
Target tools for error amount calculation are ball endmill and radius endmill.
The machining path drawn by another tool is treated in the same way as a path where the tool does not contact with the product CAD model surface. Analysis is performed by referring to tool type, diameter, etc. of the tool set on the Tool Settings window.

■ **Calculation method of error amount**

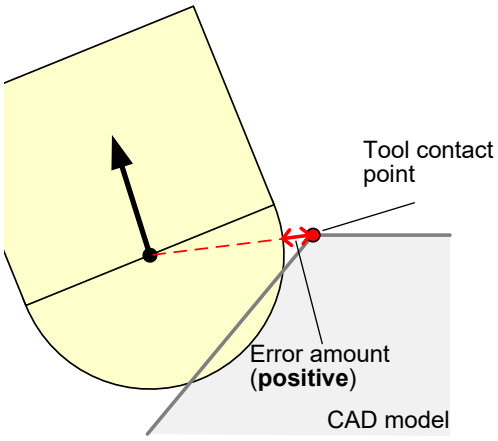
Set a 0.01 mm search distance to the cutting edge of a tool and search for target product CAD model surface for error amount calculation.
In the following figure, the surface 2 interfering with the tool and the surface 3 included in the search distance are target surfaces for error amount calculation.
The surface 1 and the surface 4 outside the search distance are non-target surfaces for error amount calculation.
When there are two or more target surfaces for error amount calculation, the minimum error amount is used among the error amounts calculated for each surface.



The surface 2 and the surface 3 are as shown in the following figures, and the distances in three-axis directions are calculated as error amounts.
The calculation result of the surface 2 is a negative value (excessively shaved) and that of the surface 3 is a positive value (incompletely shaved).

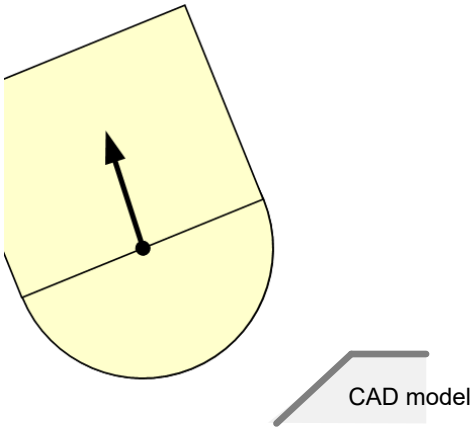


Error amount calculation image of surface 2



Error amount calculation image of surface 3

The surface 1 and the surface 4 are as shown in the following figure, and the error amount is not calculated.
This indicates that the tool does not contact with the product CAD model on the machining path.



Error amount calculation image of surface 1 and 4

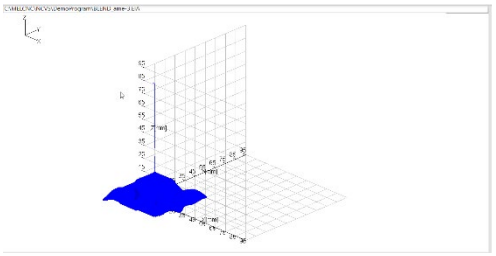
■ Types of product CAD model display

The read product CAD model display can be switched between curved surface display and wire frame display. For details, see [Curved surface display] of the sub-display "Analysis simulation settings".

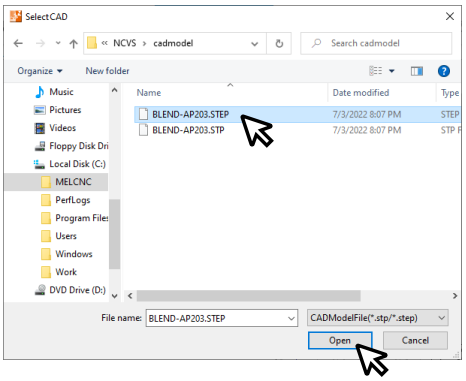
■ **Display CAD model error**

(Note) When a product CAD model file including an invalid shape is read, the application may end. Save the project before reading the file.

(1) Select a machining program and execute simulation. → The machining path is drawn.



(2) Click the [Select CAD] command. → The file selection dialog appears.



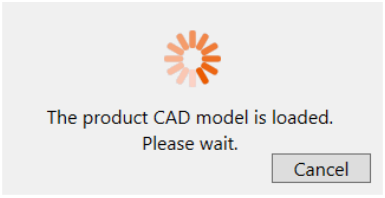
(*) When the product CAD model file is already selected in machining simulation, the already selected product CAD model is displayed by clicking the [View CAD] command.
However, only product CAD model in STEP files can be displayed. When a product CAD model in a file other than STEP file is read, an error message appears.

(*) The selected product CAD model file is displayed by clicking the [View CAD] command in machining simulation or on the difference analysis window.

(Note) A file name must not exceed 32 characters; a folder name must not exceed 48 characters. If the file name is longer than 32 characters, or the folder name is longer than 48 characters, an error message dialog appears.

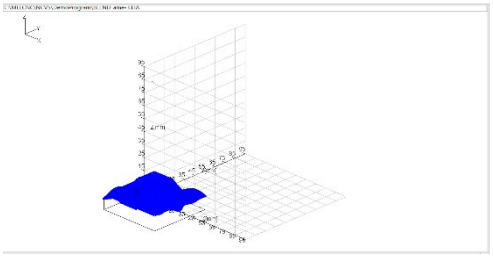
(3) Select a product CAD model file to be displayed in the simulation area, and click [Open].

The progress window appears and reading starts.
The reading time depends on the CAD model size.
To stop the reading before it completes, click the [Cancel] button.



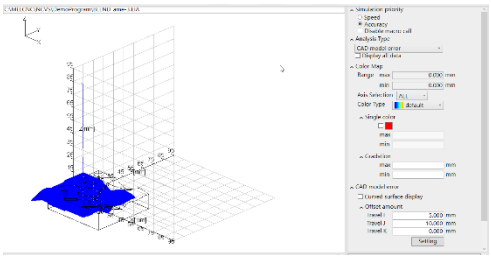
(Note) If the reading is failed, the error message "CAD model import failed." appears.

When the reading is completed, the selected product CAD model is displayed at the workpiece reference point position, and the [View CAD] command is highlighted.



(4) Enter values in [Travel] of [CAD model error] – [Offset amount] in the sub-display "Analysis simulation settings", and click [Setting].

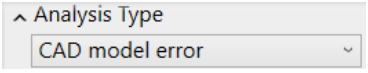
The displayed product CAD model moves by the values entered in each [Travel].



(*) When another product CAD model is selected, the values in each [Travel] of [Offset amount] are initialized to "0".

(5) Select [CAD model error] for [Analysis Type].

[CAD model error] is displayed under Analysis Type.



For the min and max of [Range] for [Color map], a blank is displayed.
For the min and max of [Single color] and [Gradation], the values last input are displayed. When displayed for the first time, min and max are blank.
[Axis Selection] under [Axis Reversal] is not selectable. (Grayed out)

(6) Click [Update].

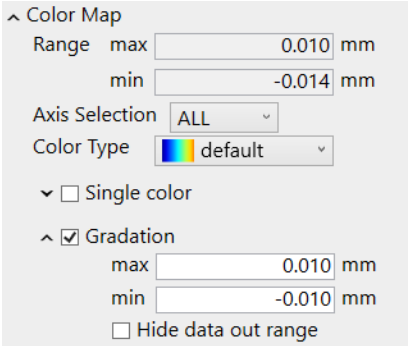
After the progress window appears, the analysis is completed.
The progress rate (%) is displayed on the progress window.

For the min and max of [Range] for [Color map], the minimum and maximum values of the analysis result are displayed.

(*) When a product CAD model file is not selected or a product CAD model file in STL format is selected, a message appears in the sub-message display area, and analysis process is not executed.

(7) Set each item for [Color Map].

Set the color of the machining path to be displayed when the analysis is completed.



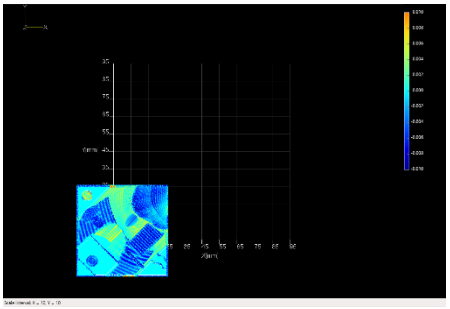
(*) If an out-of-range value is set in min or max for [Color Map], or if the min value is greater than the max value, the message "This value cannot be set." appears, and originally set value is restored.

(*) The machining path where the tool does not contact with the product CAD model surface is displayed in the same color as the [Gradation] max when the [Gradation] checkbox is checked and the [Hide data out range] checkbox is not checked. In all other cases, the machining path is hidden.

(8) Click [Update].

After the progress window appears, the analysis is completed.
The progress rate (%) is displayed on the progress window.

The machining path is displayed in gradation set in [Color Map].



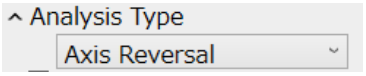
7.3.6 Turn to plot point highlighting (axis reversal)

When [Axis Reversal] is selected for "Analysis Type", the points of the axis reversal portion are highlighted in red.

■ **Display axis reversal**

(1) Select [Axis Reversal].

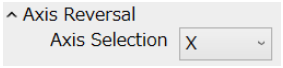
[Axis Reversal] is displayed for [Analysis Type].



The values of min and max for Color Map are displayed as blank and nonconfigurable.

(2) Select the axis to check the axis reversal in [Axis Selection].

The selected axis is displayed for [Axis Selection].

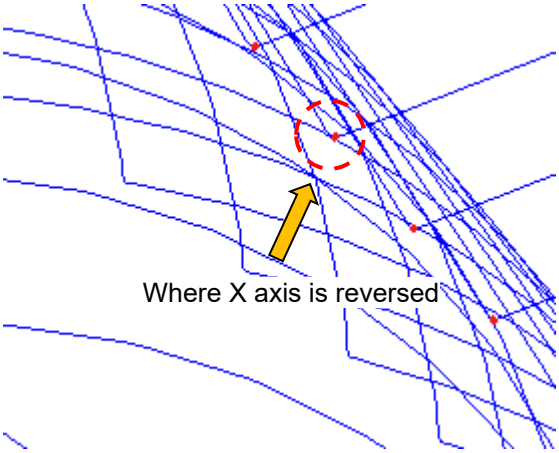


(3) Click [Update].

The progress window appears during the process.

When the process is completed, the machining path is displayed, and the positions where the selected axis is reversed are indicated with red dots.

Gradation bar is not displayed.



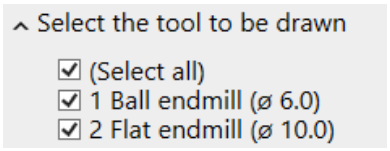
7.3.7 Selecting a tool for drawing

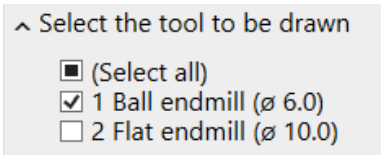
Only the machining paths that use a specific tool can be displayed. This provides the user with a visual understanding of the machining details for each process by tool selection.
Select a tool that displays a machining path from "Select tool for drawing" in the "Analysis simulation settings" sub-display.

(1) Uncheck the checkboxes of the tools to be hidden in [Select tool for drawing]
(Example) [2 Flat endmill (φ10.0)]

➡ The checkboxes are unchecked for the tools to be excluded from the drawing.
[(Select all)] becomes undetermined.

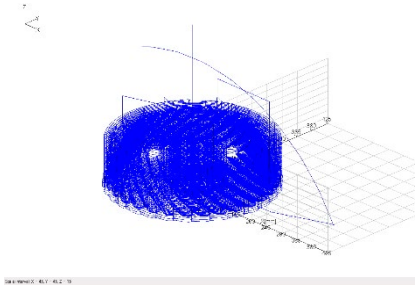
(*) The [(Select all)] checkbox is checked by default.

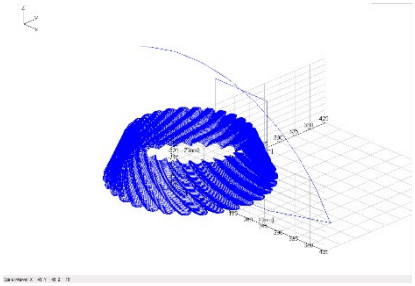




(2) Click [Update].

➡ A progress window is displayed while processing.
When processing completes, the paths of the tools with checked boxes only are displayed.





7.4 Caution

(1) If an M02 or M30 command is issued while simulation is running, the processing status of the program is canceled; however, the simulation continues until the analysis is completed.

(2) In VMware environment, if you maximize NC Virtual Simulator while simulation is running, and then minimize and expand it again, the display of the graphics area may go blank. In this case, the graphic is displayed correctly by changing the window size, switching the display of the sub-screen, switching the function menu, or performing other operations.

(3) In [Posit. Diff Map] and [Feed Diff Map], the adjacent point is searched for among only adjacent points on the XY plane.

(4) When the screen is changed to another screen while [Posit. Diff Map] or [Feed Diff Map] is selected and the screen is returned to the analysis simulation screen, the display is the same as the previous time or a blank is displayed according to the state of the simulation. To update the view, click the [Update] button.
When the screen is changed to another screen while [CAD model error] is selected and the screen is returned to the analysis simulation screen, the display is the same as the previous time or a blank is displayed according to the state of the simulation, whether or not the product CAD model file is selected again, whether or not the tool setting data is edited, and whether or not the display position of the product CAD model is changed (the workpiece reference point is moved, the offset amount is changed). To update the view, click the [Update] button.

(5) When the side-by-side display is performed for the second and subsequent times after the project is opened, the last set analysis type is displayed on the right side of the window. However, when the side-by-side display is performed during simulation while [Posit. Diff Map] and [Feed Diff Map] are set to the right side of the window, the position is displayed on the right side of the window (the feed map is displayed when the position is selected on the left side of the window).

(6) When a machining program path created by [High-speed mode (high)] is analyzed by [CAD model error], errors of incompletely shaved edges of recesses and projections are not displayed. It is recommended to create a machining program path by [High-speed mode (medium)] or [Normal mode].

(7) When the tool type, diameter, etc. are changed on the Tool Settings window after a machining program path is created, if CAD model error analysis is performed, the analysis result is different from the one at creation of machining program path.

(8) When the screen is changed to another screen while the product CAD model is displayed and the screen is returned to the analysis simulation screen, the product CAD model is hidden. The product CAD model can be displayed again by clicking the [View CAD] command.

(9) A product CAD model file is selected, and the file is deleted in Explorer, etc. and replaced with another file with the same name. Then the screen moves to another screen and returns to the analysis simulation window. When these operations are performed, the product CAD model is displayed with the replaced file contents; however, the CAD model error analysis result is displayed with the contents analyzed last time. To perform the CAD model error analysis with the replaced product CAD model file, select the product CAD model file again by clicking the [Select CAD] command.

(10) Tools used in the simulation are those corresponding to the T command immediately before the M06 command, regardless of the setting of the parameter "#1327" (Tool change method specification). However, when the T command immediately before is "0", the tool corresponding to the earlier T command is used.

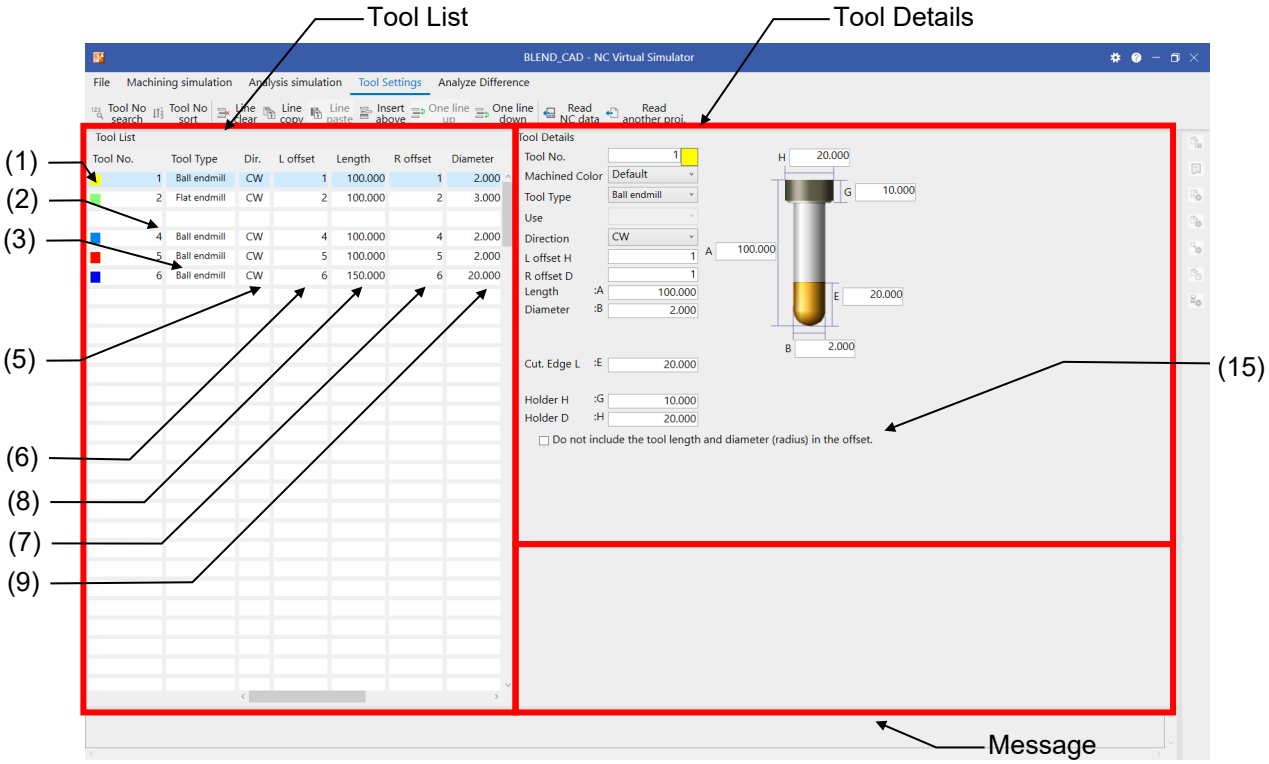
(11) When a product CAD model file, etc. including an invalid shape is read, the application may end. Save the project before reading the file.

8. Tool settings

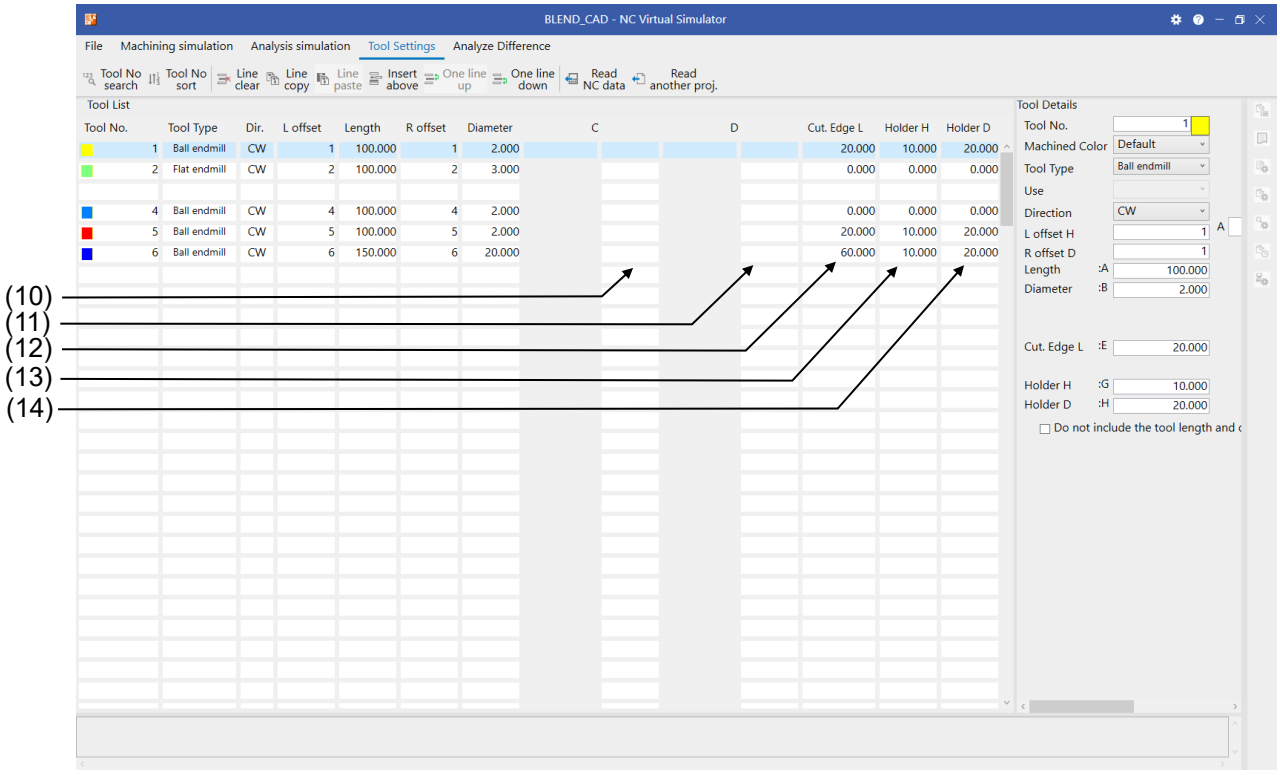
In the Tool Settings window, you set the tool data used in machining simulation and analysis simulation.

8.1 Screen configuration and display items

The Tool Settings window consists of Tool List, Tool Details, and message.



The width of the "List Display" can be changed by dragging and dropping the border between the tool details and the tool list.



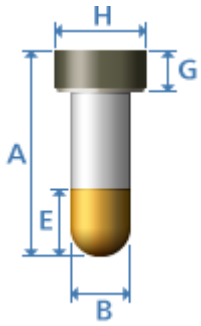
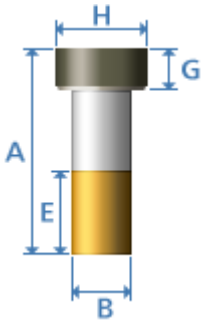
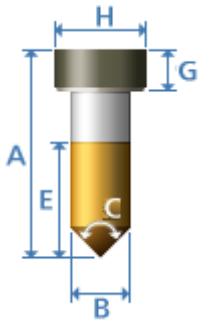
"Tool List" provides a list of detailed information about the tools.
In "Tool Details", you can set the details of the tool you selected in "Tool List". The line selected in Tool List is highlighted.
The following describes each item displayed in "Tool List" and "Tool Details" with the range of values.

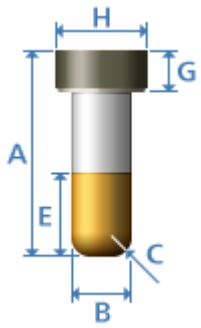
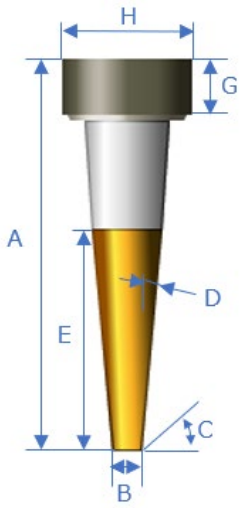
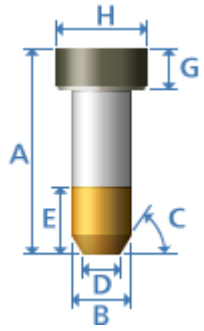
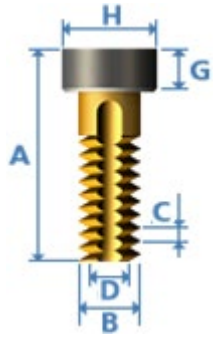
Item	Description	Range of values
(1) Machined Color	Set the machining surface color when cutting with machining simulation. When [Default] is specified, [Machined Color] or [Tapped Color] in the [Machining simulation] sub-display are set. When [User Setting] is specified, the color is selected from color setting window.	Default User Setting
(2) Tool No.	Set the tool number. Data with up to eight digits can be entered.	1 to 99999999
(3) Tool Type	Set the type of the tool.	Ball endmill Flat endmill Drill Radius endmill Taper endmill Chamfer Tap Tap - UN No. Tap - UN Frac Tap - Pipe Rc Tap - Pipe G Tap - Pipe Rp Face mill Boring Back boring
(4) Use	Not used.	-
(5) Dir: Hand/Rev	Set the hand of tool or the spindle rotation direction. Which item is to be set depends on the type of the tool. • Set as the spindle rotation direction for the following tools: Mill tool, lathing drill and lathing tap • Set as the hand of tool for the following tools: Turning tool	(Spindle rotation tool) CW CCW
(6) L offset H	Set the tool length offset number.	Depends on the number of tool offset sets.
(7) R offset D	Set the tool radius offset number. (Note) This item is not settable for compensation type I.	Depends on the number of tool offset sets.
(8) Length A	Set the tool length.	The maximum value depends on the value of #8010. 0 to 9999.999 (mm) 0 to 999.9999 (inch)
(9) Diameter B	Set the tool diameter.	
(10) C	Set the value of the displayed item. The item that is displayed varies depending on "(3) Tool Type". (*2)	Length: 0 to 9999.999 (mm) 0 to 999.9999 (inch) Angle: 0 to 180.000(°) Taper: 0 to 45.000(°) (*1)
(11) D		
(12) Cut Edge L E	Set the blade length. This item is not displayed depending on "(3) Tool Type".	
(13) Holder H G	Set the holder height.	
(14) Holder D H	Set the holder diameter.	

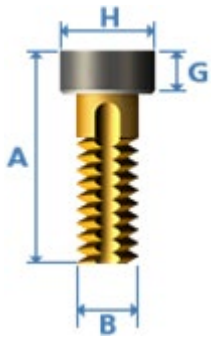
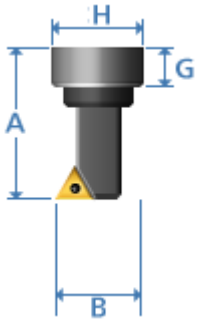
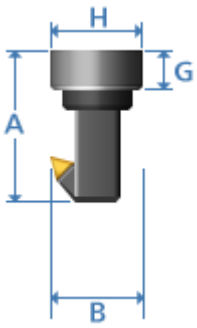
Item	Description	Range of values
(15) Do not include the tool length and diameter (radius) in the offset.	When this checkbox is checked, the tool length and tool diameter (radius) are not included in offsets.	ON/OFF

(*1) The set value may be corrected during the simulation. For details, see "11.5 Caution related to tool display".

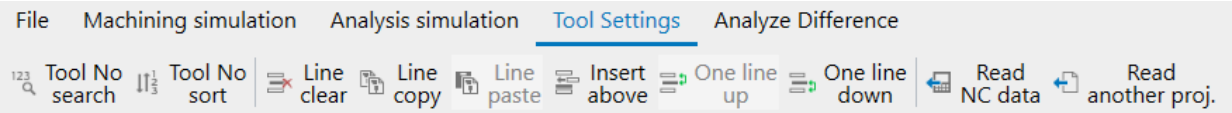
(*2) Which item is to be input varies depending on the type of the tool.
The following summarizes the tool types and items to be input.

Tool type	Image	Items to be input
Ball endmill		A: Length B: Diameter G: Holder H H: Holder D E: Cut. Edge L
Flat endmill		A: Length B: Diameter G: Holder H H: Holder D E: Cut. Edge L
Drill		A: Length B: Diameter C: Nose angle G: Holder H H: Holder D E: Cut. Edge L

Tool type	Image	Items to be input
Radius endmill		A: Length B: Diameter C: Corner R G: Holder H H: Holder D E: Cut. Edge L
Taper endmill		A: Length B: Diameter C: Corner R D: Taper G: Holder H H: Holder D E: Cut. Edge L
Chamfer		A: Length B: Diameter C: Tip angle D: Tip dia. G: Holder H H: Holder D E: Cut. Edge L
Tap		A: Length B: Diameter C: Pitch D: Root dia. G: Holder H H: Holder D

Tool type	Image	Items to be input
Tap > UN No. Tap > UN Frac Tap > Pipe Rc Tap > Pipe G Tap > Pipe Rp		A: Length B: Diameter G: Holder H H: Holder D
Face mill		A: Length B: Diameter C: Cutter L D: Shank dia. G: Holder H H: Holder D E: Cut. Edge L
Boring		A: Length B: Diameter G: Holder H H: Holder D
Back boring		A: Length B: Diameter G: Holder H H: Holder D

8.2 Tool settings commands



The following lists the commands that can be used in the Tool Settings window.

	Command	Description	Reference
1	Tool No search	Search for a tool number and display the matched tool data in the top line.	8.3.2 Search tool data
2	Tool No sort	Sort the tool numbers in ascending or descending order.	8.3.3 Sort tool numbers
3	Line clear	Delete the tool data in selected lines.	8.3.4 Delete, insert or move lines of tool data
4	Line copy	Copy the tool data in a selected line to the memory area.	8.3.5 Copy and paste one line of tool data
5	Line paste	Paste the tool data that was copied in the memory area.	
6	Insert above	Insert tool data above a selected line.	8.3.4 Delete, insert or move lines of tool data
7	One line up	Move the tool data in a selected line up by one line.	
8	One line down	Move the tool data in a selected line down by one line.	
9	Read NC data	Read the tool management data in the all-backup data to the tool data of NC Virtual Simulator.	8.3.6 Read NC tool management data
10	Read another proj.	Import the tool data saved in another project.	8.3.7 Import tool data of another project

8.3 Operation

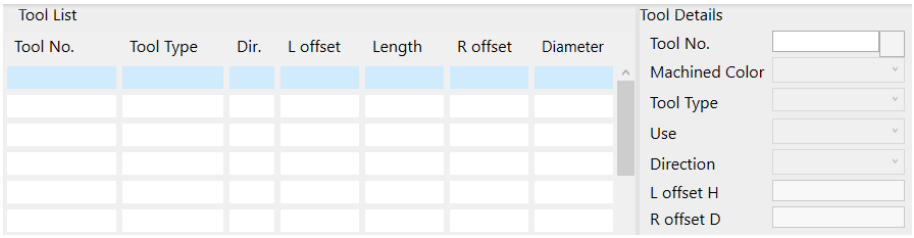
Click [Tool Settings] in the function menu.

8.3.1 Set tool data

Set the tool data after verifying the relationship between the tool numbers and offset numbers specified in the machining program to be simulated.

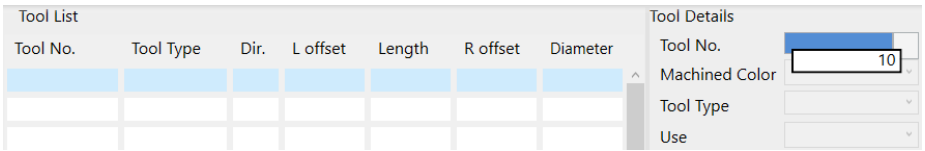
The following describes how to register new tool data.
The figures show an example where "Tool Type: Drill" is added to Tool No. 10.

(1) Click a blank line in "Tool List".
The details of the line (tool number) you clicked are displayed in "Tool Details".



(2) Click any field in "Tool Details".
The field you clicked is highlighted and an input dialog appears.
If you press [Enter] or [Tab] (+ [Shift]) without clicking a field, an input dialog appears at the "Tool No." field. You can also navigate between the fields by pressing the [↓][↑] keys.

(3) Enter a tool number in the field, and press the [Enter] key.



(4) When the tool number is set, the next item is highlighted and an input dialog appears.

(*) If an out-of-range value is entered, "Tool Number is out of range." appears in the message area, and the value is not changed.

(*) If you press [Esc], the value in the field you clicked is not updated.

(*) The same value as the tool number is also set in L offset "H" and R offset "D".

(*) If the tool number entered is not within the range of tool offset numbers, the tool number is not set in L offset "H" and R offset "D". The message "L offset "H" or R offset "D" is missing." appears. Enter a tool offset number.

(*) If a value is set in any one of the following: L offset "H", R offset "D", Length "A", Diameter "B", the length offset value and radius offset value for that offset number (H, D) are updated.

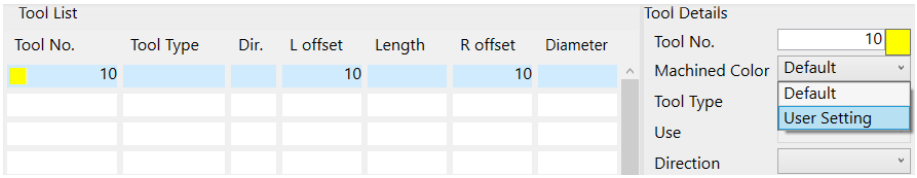
(*) If a duplicate tool number is entered, the message "Tool Number is duplicate." appears.

(5) Set the machining surface color.

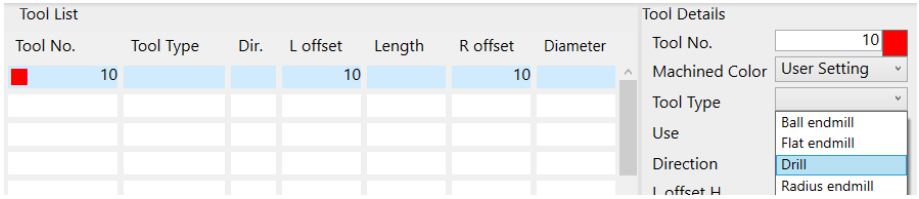
When setting the tool No., the machining surface color is set to [Default]. To change the machining surface color, select [User setting].

Changing this to [User setting] will open the "Color setting" dialog.

(*) The machining surface color can also be changed by clicking the color button beside the tool No. Changing the machining surface color from the color button while [Machined color] is set to [Default] will switch the setting to [User setting].

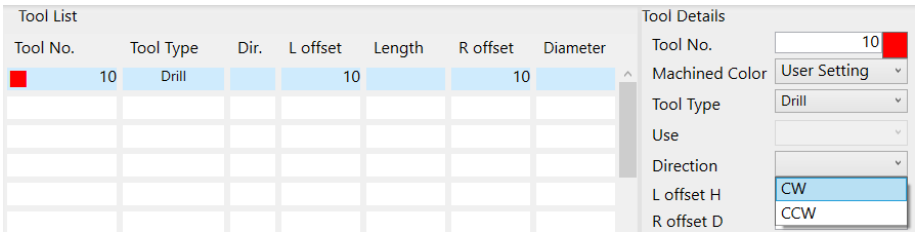


(6) Click the "Tool Type" field, and select a tool type.



(7) Click the "Direction field", and select a tool rotation direction.

Select [CW] for a tool to be used in the spindle forward rotation (clockwise), and [CCW] for a tool to be used in the spindle reverse rotation (counterclockwise).

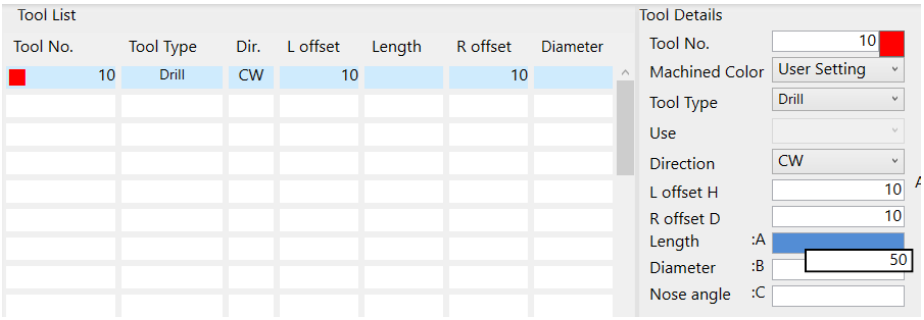


(8) Set an offset number to be used for tool length offset in the "L offset H" field.

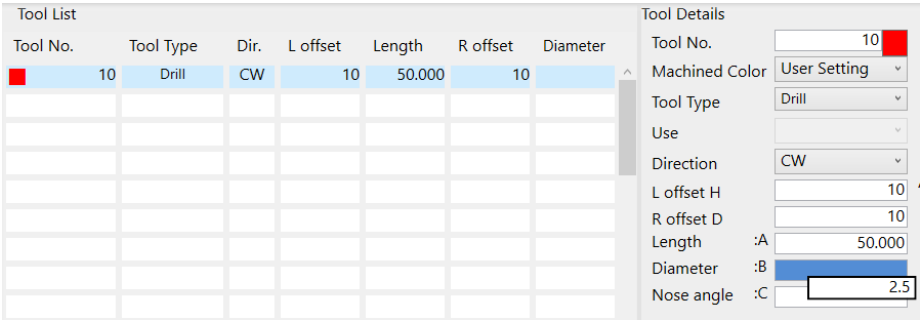
Set an offset number to be used for tool diameter offset in the "R offset D" field.

The same value as the tool number is set in both fields by default. Change the setting value to use a different number from the tool number.

(9) Click the "Length: A" field and enter a tool length ("A" dimension in the guide figure).
The value set in "Length: A" is set in the offset amount for the "L offset H" number. The value is also set for "Length A" in "Tool List".



(10) Click the "Diameter: B" field, and enter a tool diameter ("B" dimension in the guide figure).
A half value (radius value) of the entered value is set in the offset amount for the "R offset D" number. The value is also set in "Diameter" in "Tool List".



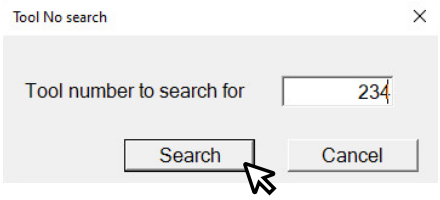
If you click the next line in "Tool List" or press [Tab] (+ [Shift]), the input dialog in "Tool Details" is hidden.

8.3.2 Search tool data

You can enter a tool number to search for and view the tool data.
The following describes how to search tool data.

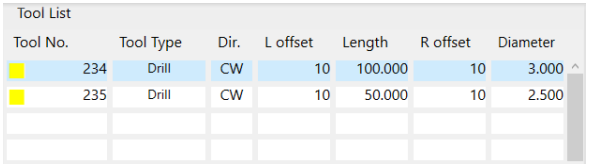
(1) Click the [Tool Settings] tab > [Tool No search] command.
(*) You can also invoke Tool No search using the shortcut key [Ctrl] + [F].

(2) The Tool No search dialog appears.
Enter a tool number to search for and click [Search].
You can only input number keys (0 to 9) in the Tool No search dialog.



If you click [Search] without entering a tool number, the error message "Tool Number is missing." appears. Be sure to set a tool number before clicking [Search].

(3) The tool number you searched for is displayed in the first line of Tool List.

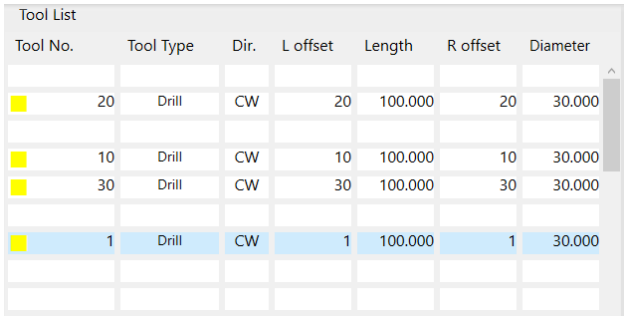


If there is no matching tool number, the error message "Tool number not found." appears.

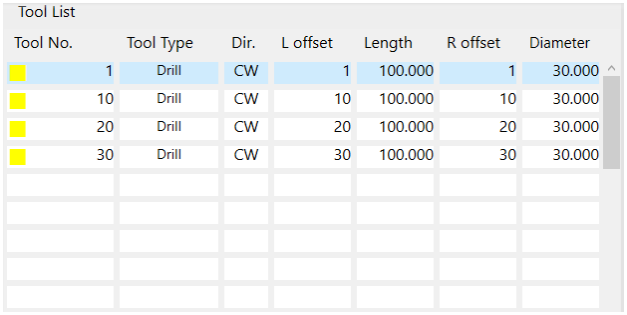
8.3.3 Sort tool numbers

You can change the order of tool data according to the tool number. (This operation is called "sort".)
The following describes how to sort the tool numbers.

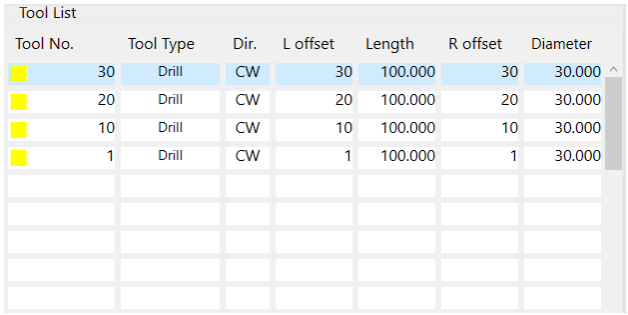
(1) Click the [Tool Settings] tab > [Tool No sort] command.



(2) The tool numbers are sorted in ascending order.
After sort, all blank lines are moved to the bottom.



(3) If you click the [Tool No sort] command again, the tool numbers are sorted in descending order.



(4) After that, each time you click the [Tool No sort] command, the tool numbers are sorted alternately: ascending > descending > ascending.
Note that if you click the [Tool No sort] command after the following operations, the tool numbers are sorted in ascending order.

- Switch to a function window such as machining simulation or analysis simulation.
- Move the cursor to Tool Details.
- Edit Tool List (delete, insert, move).
- Read tool management data or import tool data.
- Open another project.

8.3.4 Delete, insert or move lines of tool data

The following describes how to delete, insert, or move tool data.

(1) Click to select the target line in the Tool List, and click the following command in the [Tool Settings] tab.
Delete a line: [Line clear] command (*1)
Insert a line: [Insert above] command (*2)
Move a line: [One line up] or [One line down] command

(*1) Two or more lines can be selected to delete at a time.
To select consecutive lines, click the top line and then click the bottom line while holding the [Shift] key.
To select non-consecutive lines, click the target lines while holding the [Ctrl] key.
To deselect the lines, press the [ESC] key.

(*2) If tool data exists in the bottom line of the Tool List when a line is inserted, the confirmation dialog appears. If you click [OK], a blank line is added and the bottom line is deleted.

8.3.5 Copy and paste one line of tool data

The following describes how to copy and paste one line of tool data.

(1) Click the line to copy in Tool List.
Click the [Tool Settings] tab > [Line copy] command.

(*) If no line or multiple lines are selected, the [Line copy] command is not selectable (grayed out).

(*) If you perform the following operations, the copied content is canceled.

- Switch to a function window such as machining simulation or analysis simulation.
- Move the cursor to Tool Details.
- Edit Tool List (delete, insert, move).
- Read tool management data.
- Open another project.

(2) Click the line to which the copied line will be pasted.
Click the [Tool Settings] tab > [Line paste] command.
If the line is pasted to a line where data exists, "Tool No." is not changed but the other items are overwritten.
If the line is pasted to a blank line, a dialog for setting a tool number appears. Enter a unique tool number.

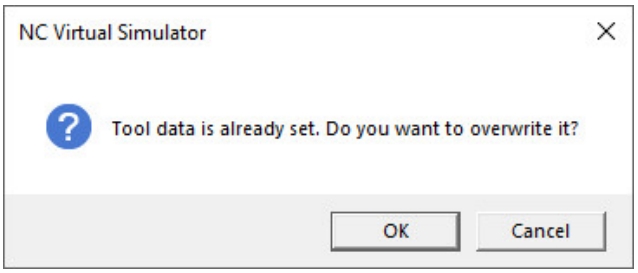
(*) If no tool number is entered or if the tool number you entered already exists or is not within the range, a warning dialog appears. Enter a tool number again.

8.3.6 Read NC tool management data

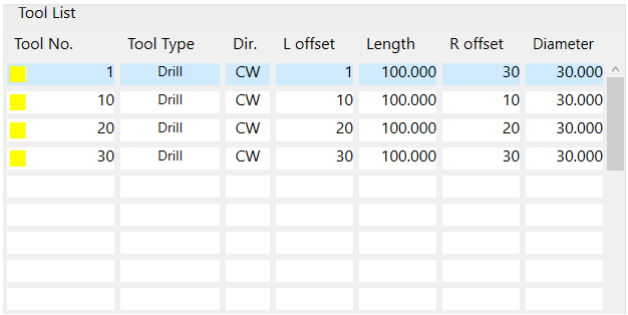
NC Virtual Simulator allows you to create tool data using the tool management data backed up by the All-backup function of the NC unit.
The following describes how to read tool management data.

(1) Click the [Tool Settings] tab > [Read NC data] command.

(*) If tool data already exists, the following confirmation dialog appears.



(2) The read tool management data is displayed.



(*) If the read tool data contains a tool whose tool type is not listed in Tool type of "8.1 Screen configuration and display items", the line corresponding to the tool displays no tool data and remains blank.

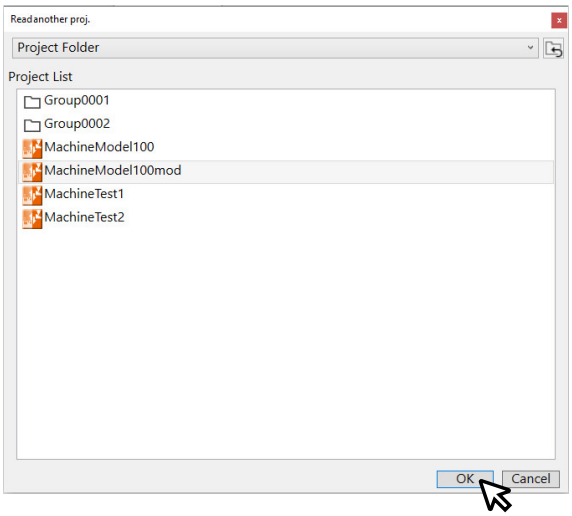
(*) The [Machined Color] of all the read tool data is set to [Default].

(Note) When "Length A" and "Diameter B" in tool management data are set, the values are set as the tool offsets for the offset numbers (L offset "H", R offset "D") same as the tool number so that the behavior of the offsets is consistent between simulation and the NC unit. For this reason, be aware that this consistency cannot be maintained if you use the [Read NC data] command after this setting is done.

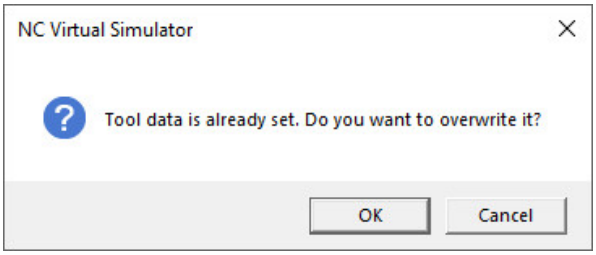
8.3.7 Import tool data of another project

The following describes how to import tool data from another project.

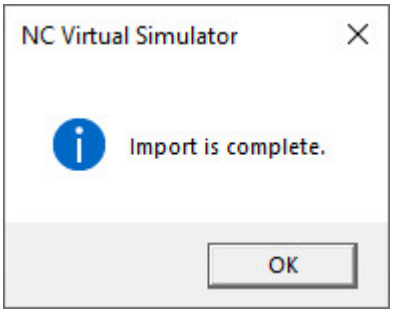
(1) Click the [Tool Settings] tab > [Read another proj.] command.
The Read another proj. dialog appears.
Choose a project and click [OK].



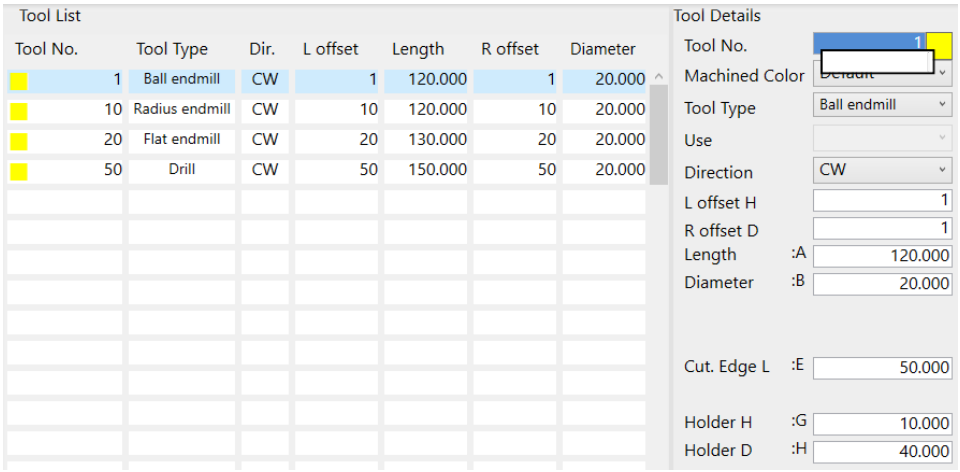
(*) If tool data already exists, the following confirmation dialog appears. Click [OK].



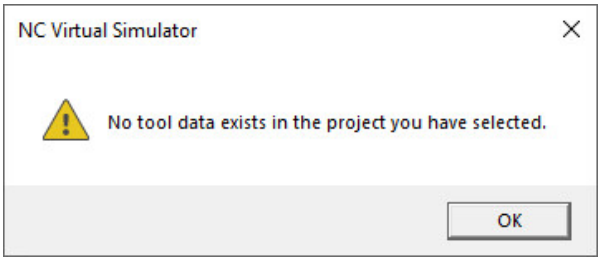
(2) The tool data of the selected project is read.
The tool data is read line by line from the top. The tool length and radius offsets corresponding to L offset "H" and R offset "D" in the read data are also updated.
When the import of the tool data is complete, the following dialog appears. Click [OK].



The tool data that is read first is displayed in the first line of Tool List.



If no tool data exists in the selected project, the following error message appears.



8.4 Caution

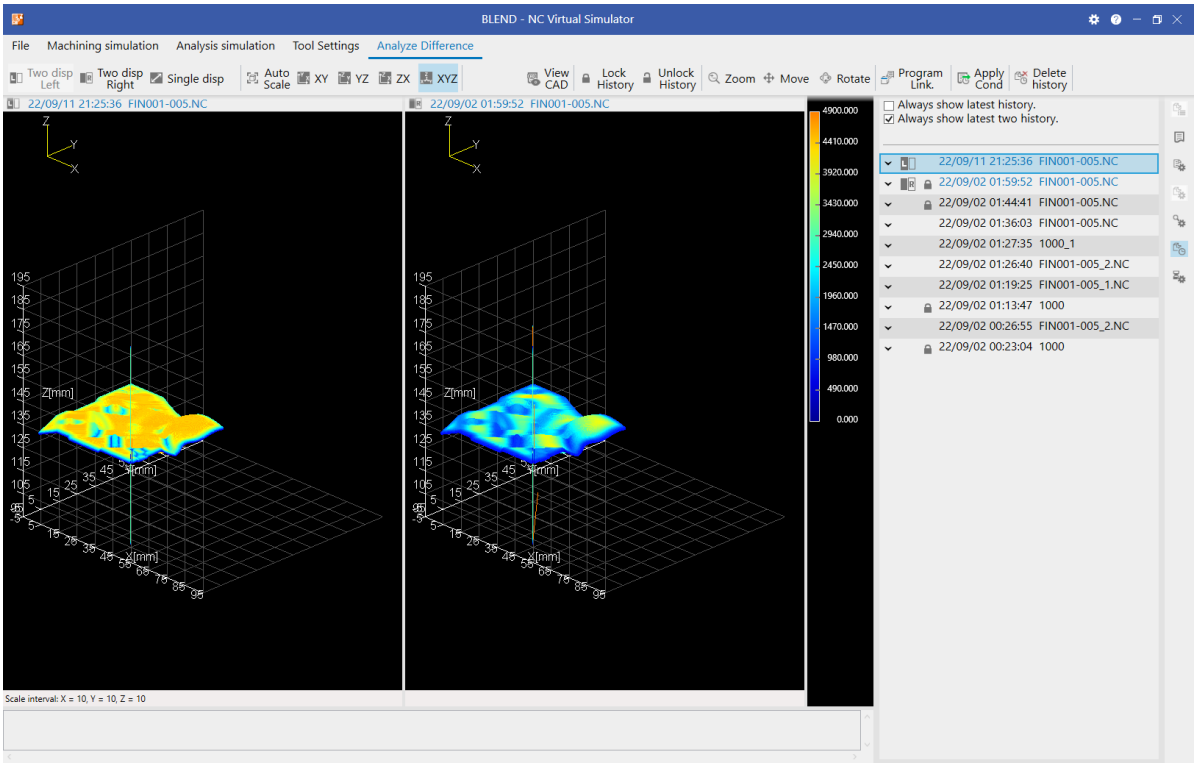
(1) Since R offset "D" cannot be specified when compensation type I is selected, the tool radius is not set in the tool radius offset. To change the tool radius offset, change it in the sub-display "NC Data List". When compensation type II is selected, ensure that the length and wear amount of the shape are taken into account in the offset values of L offset "H" and R offset "D". You cannot set the wear amount only. To change the wear offset, change it in the sub-display "NC Data List".

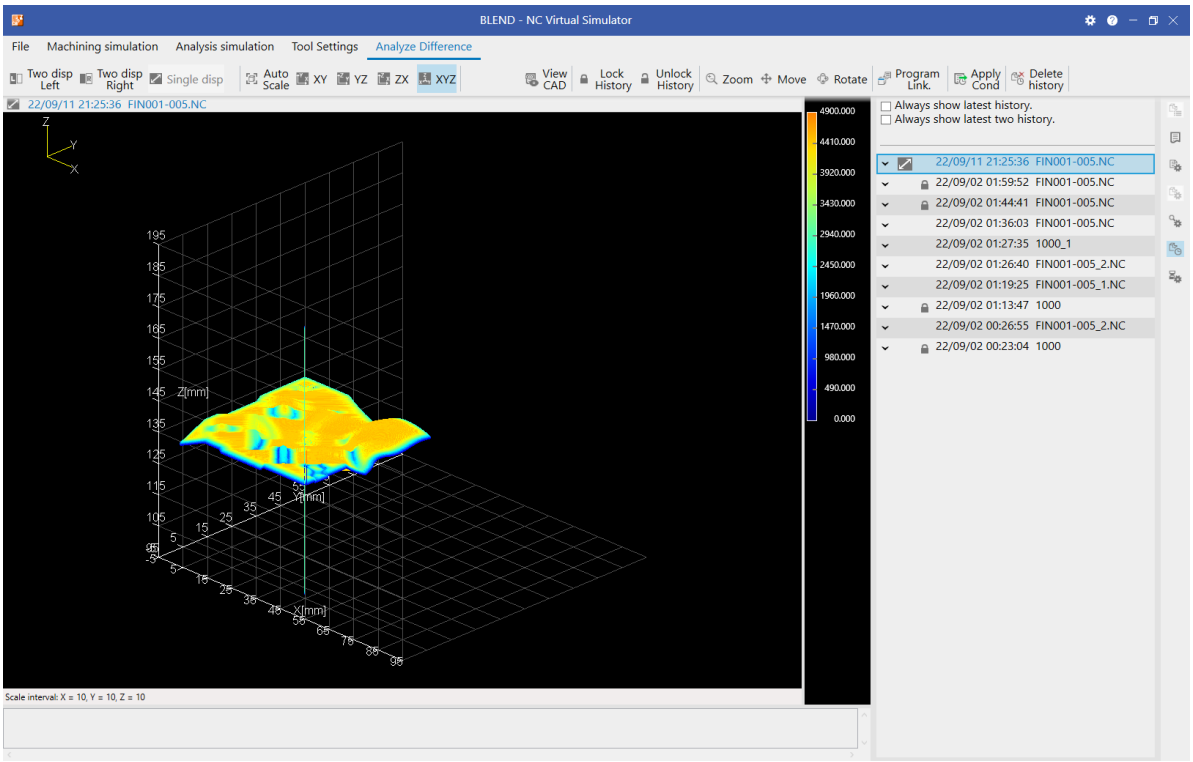
(2) If there are duplicate offset numbers, it causes unintended simulation because the offsets for the duplicate tool are changed when the tool length or tool radius is changed, which results in inconsistency between the tool length and tool radius and the offsets.

(3) In the machining simulation screen, clicking the [Work Init] command after changing [Machined Color] and [Tapped Color] in the sub-display "Machining simulation settings" will switch the machined color of the tool data for which [Default] is selected for [Machined Color] to the color after the change.

9. Simulation difference analysis

The simulation difference analysis function allows you to display the past analysis simulation results on two screens or one screen and to check machining shape differences caused by NC parameter differences.

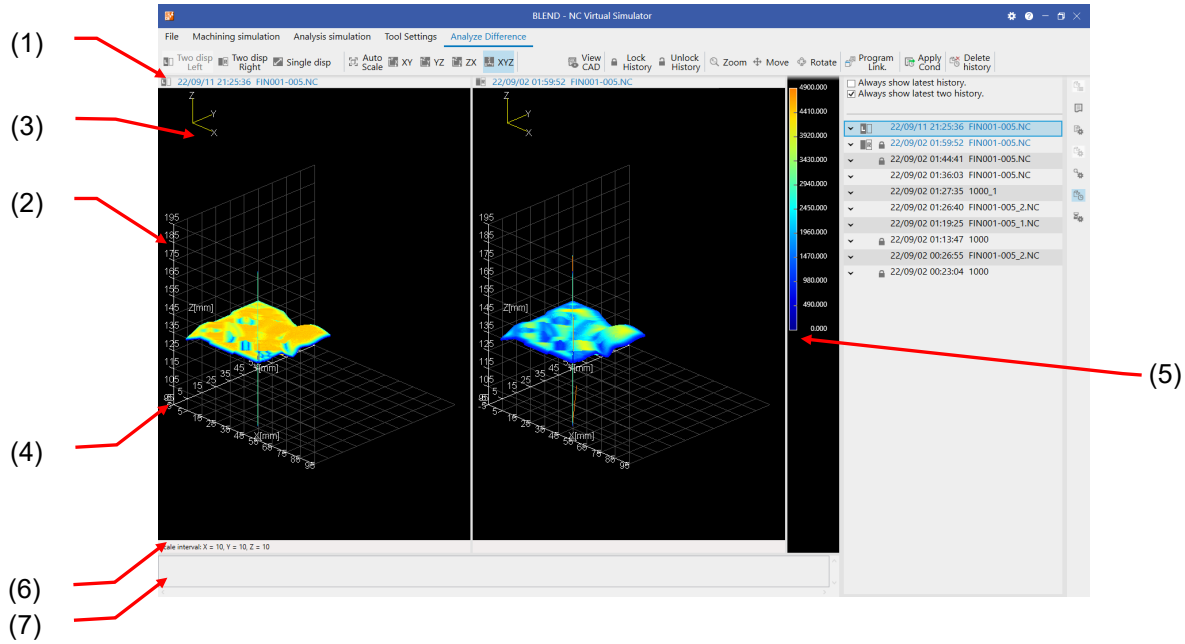




9.1 Simulation difference analysis window

9.1.1 Screen configuration and display items

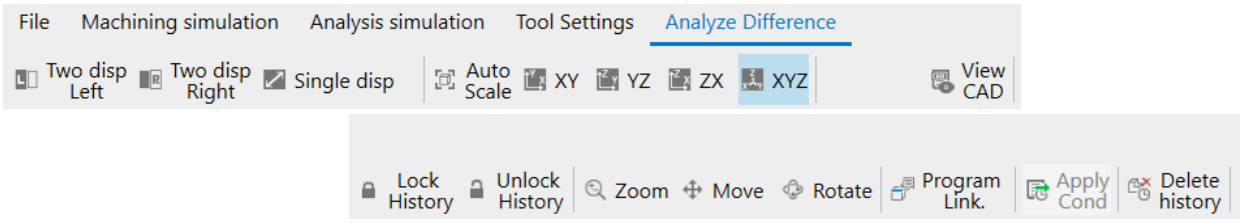
In the simulation difference analysis window, you can compare the past simulation results.
The display area of the simulation results to be compared consists of one screen or two screens of the simulation area (right) and the simulation area (left).



Item		Description
(1)	Simulation history title	Displays the time when simulation of the machining shape displayed in the graphics area is started (local time of the PC in use), and the program name. If the complete simulation history title cannot be displayed, "…" to show omission is displayed at the end.
(2)	Graphics area	Displays the machining shape of the selected simulation results. You can zoom, rotate, and move the machining shape. The display is switched between one-screen display and two-screen display.
(3)	Coordinate axes (*1)	Displays the coordinate axes X, Y, Z in the upper-left corner of the graphics area.
(4)	Scale (*1)	The scales for base axes I, J and K are displayed around the machining shape. The tick marks for each axis divide the workpiece width and extra value into 10 to 1000 segments.
(5)	Gradation bar display	This item is displayed when the analysis type (Feed Map, Acceleration Map, etc.) for color map display is selected. The relationship between the range of values set in [Gradation] and [Single color] of the sub-display "Analysis simulation settings" and the color allocation.
(6)	Status bar	Displays the drawing area information.
	Scale interval (*1)	Displays the scale interval of the scale display.
(7)	Message area	Displays error messages that occurred during machining simulation and analysis simulation. Error messages for alarms that occurred during simulation and for alarms that occurred in NC are displayed.

(*1) When two screens are displayed, the same contents are displayed on the right and left screens.
(*2) When two screens are displayed, the scale interval is displayed on the left screen only.

9.1.2 Simulation difference analysis commands



The following lists the commands that can be used in simulation difference analysis.

	Command	Description	Reference
1	Two disp Left	Displays the simulation history selected in the sub-display "Simulation History" on the left area of the two screens.	9.2.1 Compare the simulation results
2	Two disp Right	Displays the simulation history selected in the sub-display "Simulation History" on the right area of the two screens.	
3	Single disp	Displays the simulation history selected in the sub-display "Simulation History" on one screen.	
4	Auto Scale (*1)	Zoom the machining shape (cutting feed data) so that the size is about 90% of the graphics area.	-
5	XY (*1)(*2)	Change the plane where the workpiece shape is displayed or change the display to 3D. The scale in the plane selected before switch is retained.	6.3.8 Change the point of view on workpiece
6	YZ (*1)(*2)		
7	ZX (*1)(*2)		
8	XYZ (*1)(*2)		
9	View CAD (*1)(*2)	Toggles between showing and hiding the product CAD model.	-
10	Lock History	Locks the simulation history selected in the sub-display "Simulation History". The history is retained not to be deleted from the simulation history. The lock mark ( ) is displayed for the locked history.	9.2.4 Retain simulation history
11	Unlock History	Unlocks the locked simulation history.	
12	Zoom (*1)(*2)	Zoom the displayed machining shape.	6.3.9 Adjust the workpiece display position, angle and size
13	Move (*1)(*2)	Move the displayed machining shape.	
14	Rotate (*1)(*2)	Rotate the displayed machining shape.	
15	Program Link. (*2)	Shows which block in the machining program corresponds to the line segments selected in the displayed machining shape and vice versa.	9.2.7 Enable program link
16	Apply Cond	Applies the machining conditions of the simulation history selected in the sub-display "Simulation History" to the parameters of the currently open project.	9.2.5 Apply machining conditions of simulation history to current project
17	Delete history	Deletes the simulation history selected in the sub-display "Simulation History".	9.2.6 Delete simulation history

(*1) When two screens are displayed, displays of both screens are switched at the same time.

(*2) The operation is equivalent to that of analysis simulation.

(Note 1) When the [XY], [YZ] or [ZX] command is selected, the [Rotate] command is not selectable.

(Note 2) The [Apply Cond] command is not selectable while simulation is paused.

(Note 3) Some commands are not selectable according to the state of the sub-display "Simulation History".

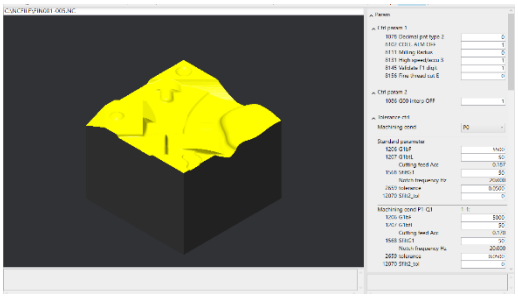
State of the sub-display "Simulation History"	Unselectable command
Hidden	Two disp Left, Two disp Right, Single disp, Apply Cond, Lock History, Unlock History, Delete history
No history exists.	Two disp Left, Two disp Right, Single disp, Apply Cond, Lock History, Unlock History, Delete history
The selected history is not one.	Two disp Left, Two disp Right, Single disp, Apply Cond
All unselected simulation histories are locked.	Lock History
The machining program of the selected simulation is different from that of the other simulation history.	Two disp Left, Two disp Right
The history displayed in the simulation area (left) is selected.	Two disp Left
The history displayed in the simulation area (right) is selected.	Two disp Right
The history displayed in the simulation area on one screen is selected.	Single disp
A history not displayed in the simulation area is selected.	Apply Cond

9.2 Operation

9.2.1 Compare the simulation results

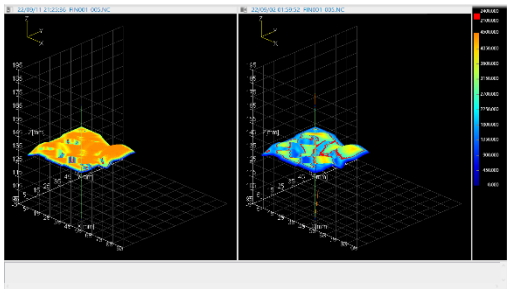
The past analysis simulation results can be displayed side by side.

(1) Click the [Start] command of [Machining simulation] or [Analysis simulation]. → Simulation is started.

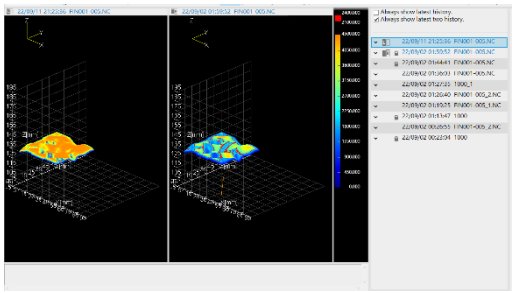


(2) After the simulation, click the [Analyze Difference] tab. → The simulation display area is divided into two screens, and the shapes are displayed as follows.

Simulation area (left): Machining shape simulated in the step (1)
Simulation area (right): Machining shape simulated last time prior to the step (1)

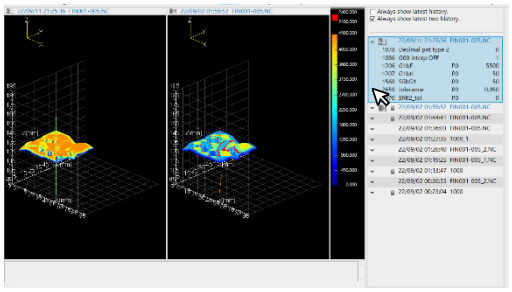


(3) Select [Simulation History] from the sub-display switch buttons. → The simulation history is displayed in the sub-display. The top history in the simulation history list is selected.



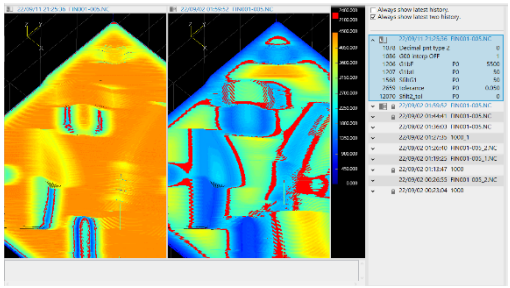
(4) To check the machining condition of the sub-display "Simulation History", click ⌄ in the history list, or press the [Enter] key.

→ The machining condition is displayed.



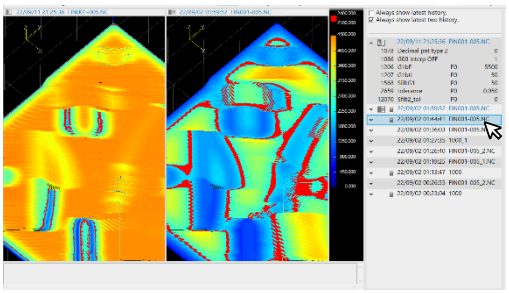
(5) To adjust the point of view and display size of the machining shape, use the [Auto Scale], [XY], [YZ], [ZX], [XYZ], [Zoom], [Move] and [Rotate] commands.

→ The machining shape is displayed according to the clicked command.



(6) To compare with another history, click the history to be compared in the history list of the sub-display "Simulation History".

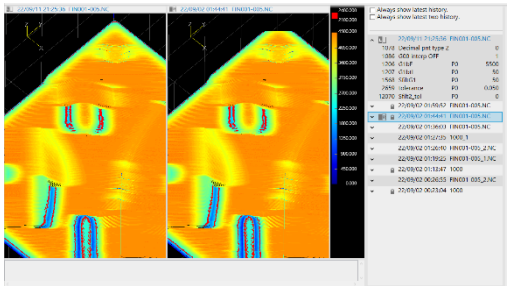
→ The clicked history is selected.



The selection state is not changed even if ⌄ and ⌃ are clicked.
The selection of the history can be moved up and down by pressing the [↑] and [↓] keys on the keyboard.
The top or bottom history is selected by pressing the [↑] or [↓] key while holding the [Ctrl] key.

(7) Click the [Two disp Right] command. (*1)

After the progress window appears, the machining shape of the simulation history selected in the step (6) is displayed in the simulation area (right).
On both screens, the state where [Zoom], [Rotate] and [Move] are executed in the step (5) is retained.
[Always show latest two history.] is unchecked.



The operation is the same in the following cases.
Press the [Enter] key while the [Right side of two disp] menu is selected.
Press the application key on the keyboard.

(*1) When the mouse is right-clicked and the [Right side of two disp] menu is selected, the behavior is the same.

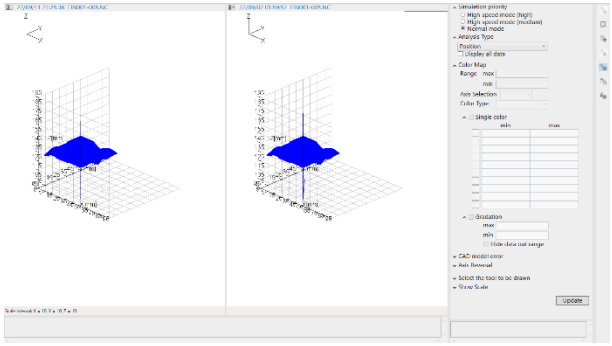
(*) If the simulation history displayed last time cannot be displayed because another machining program is selected, the amount of history data is limited, etc., the latest displayable simulation history is displayed.

9.2.2 Switch the analysis type

The analysis type of the machining shapes displayed to compare can be switched. On both screens, the machining shapes are displayed with the switched analysis type.
Analysis type: Position/Feed Map/Acceleration Map/Posit. Diff Map/Feed Diff Map/Axis Reversal

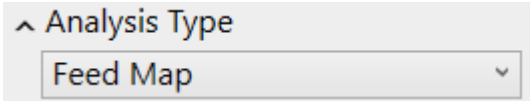
(1) Select [Analysis simulation settings] from the sub-display switch buttons.

The "analysis simulation setting" is displayed in the sub-display.



(2) Select an analysis type from [Analysis Type] in the sub-display "Analysis simulation settings".

The selected type is displayed in [Analysis Type].
This is an example when [Feed Map] is selected.



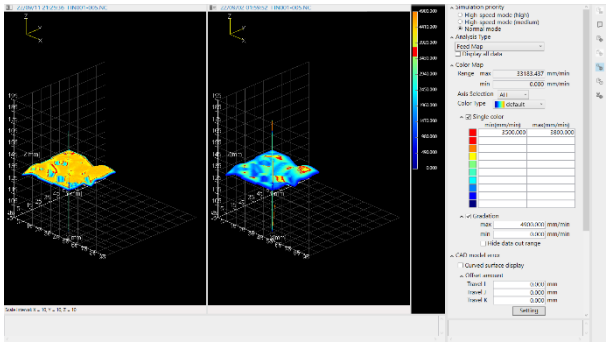
The range of the color map is as follows.

- min/max of [Range]: Minimum/maximum values of [Feed Map] for the history displayed in the simulation area
- min/max of [Single color], [Gradation]: Values entered last time

[Axis Selection] of [Axis Reversal] is not selectable.

(3) Click [Update].

After the progress window appears, the screen update is completed.
Both screens turn to the feed map display.

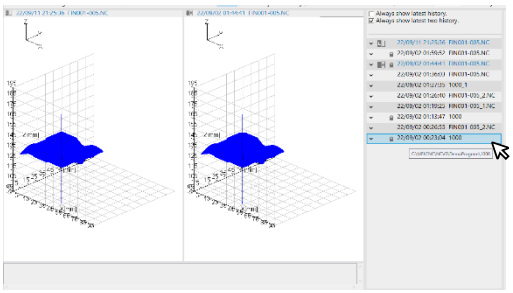


9.2.3 Switch to simulation result of a different machining program

The simulation result of a different machining program from the program for the currently displayed simulation result is displayed for comparison.
If the machining program file name displayed in the simulation history list is the same but the storage location path is different, the machining program is regarded as a different one.

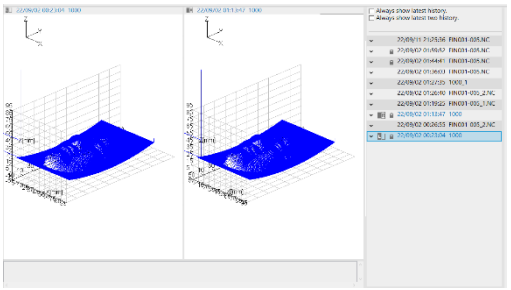
(1) Position the mouse pointer on a title in the simulation history list in the sub-display "Simulation History".

→ The full path of the machining program where the mouse pointer is positioned in the simulation history is displayed in a tooltip.



(2) Select a history of the simulation executed in a different machining program from the currently displayed simulation history.
Click the right-click menu [Left side of two disp].

→ After the progress window appears, the screen update is completed. The machining shapes are displayed in the simulation area on two screens. (*1)
Simulation area (left): Machining shape of the selected simulation history
Simulation area (right): Machining shape of the latest simulation history in the same machining program as the selected simulation history. (*2)



(*1) On both screens, the state before the step (2) is retained when [Zoom], [Rotate] and [Move] are executed. Adjust the machining shape view as needed.

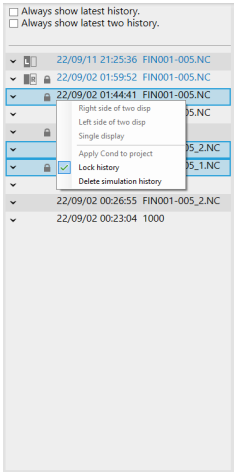
(*2) When there is no simulation history in a machining program with the same full path, the machining shape is displayed on one screen.

9.2.4 Retain simulation history

When the number of saved simulation histories exceeds the maximum number, the oldest history is automatically deleted.
An automatically deleted simulation history cannot be restored. Setting [Lock History] to a history to be retained will keep the history from being deleted and retain it.

(1) Select a history to be retained in the simulation history list, and click the [Lock History] command. (*1)

The selected history is retained but not automatically deleted.
The lock mark is displayed.



(3) Click the [Unlock History] command. (*1)

The lock mark is hidden, and the history is the target of automatic deletion.

(*1) When the mouse is right-clicked and the [Lock history] menu is selected, the history can be locked and unlocked.

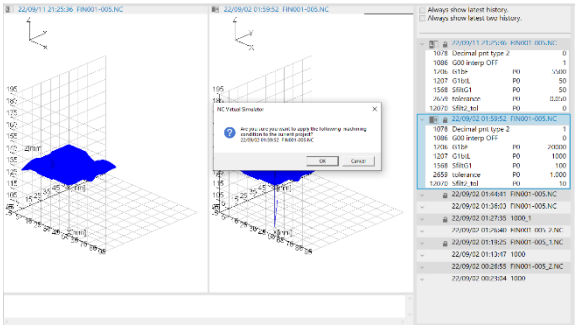
(Note1) When no simulation history is selected or 10 histories are selected, the [Lock History] is disabled.
(Note2) When two or more histories are selected, the top history without the lock mark is the target of history lock.

9.2.5 Apply machining conditions of simulation history to current project

Machining conditions (NC parameters) of the simulation history are applied to machining conditions of the currently selected project.
With this function, past machining conditions (NC parameters) can be easily applied to the current machining conditions.

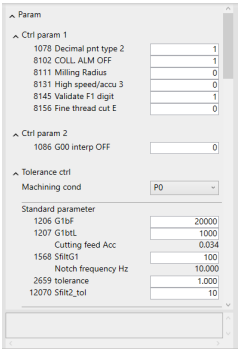
(1) Select a history to apply the machining conditions in the simulation history list.
Click the right-click menu [Apply Cond to project] or the [Apply Cond] command.

→ The confirmation dialog appears.
This menu is not selectable for histories other than those displayed in the simulation area.
This menu is not selectable while simulation is paused.



(2) Click [OK].

→ The parameters for the machining conditions of the history selected in (1) are applied to the project.
The applied parameters can be checked in the sub-display "NC Data List".

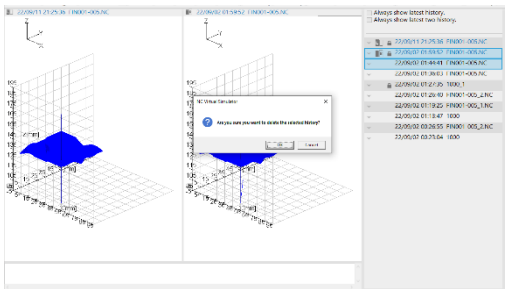


9.2.6 Delete simulation history

Any unnecessary simulation history can be manually deleted.

(1) Select a history to be deleted in the simulation history list.
Click the right-click menu [Delete simulation history] or the [Delete history] command.

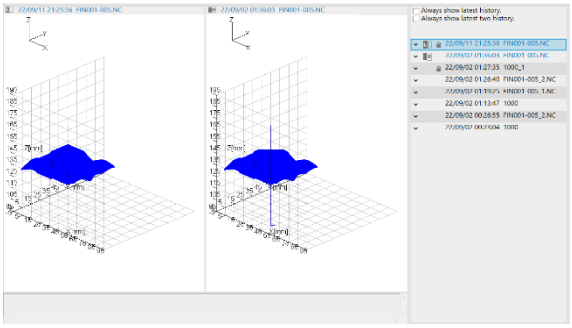
→ The confirmation dialog appears.



(3) Click [OK].

→ The history selected in (1) is deleted from the simulation history list.

- When the simulation history displayed in the simulation area is deleted, the latest simulation history is displayed.
- When all the histories are deleted, the simulation area is displayed on one screen and no machining shape is displayed in the graphic area.

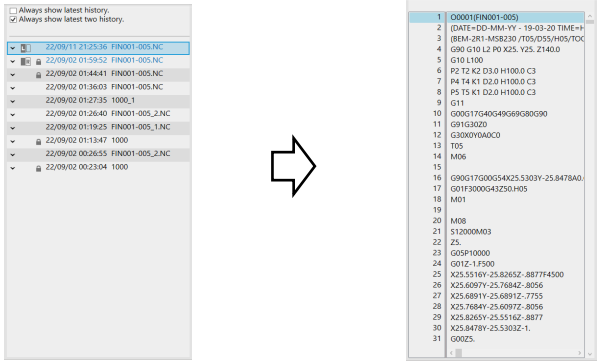


9.2.7 Enable program link

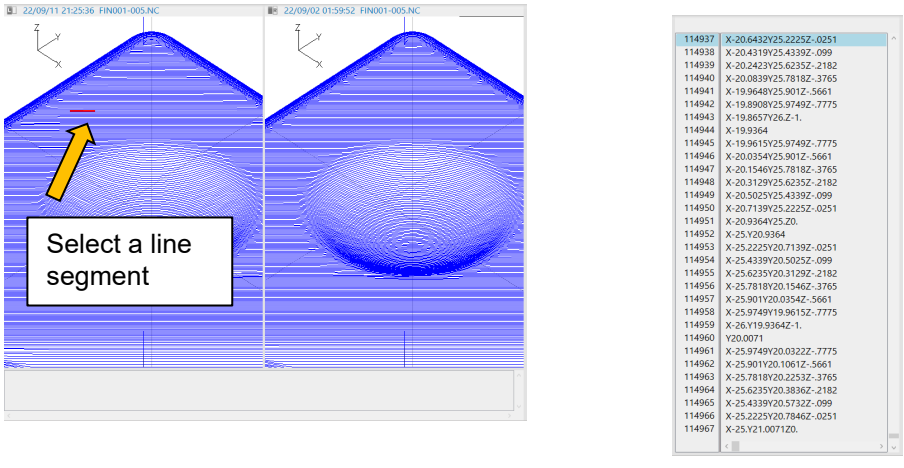
If you click a line segment in the machining shape, the line segment is highlighted in red, and the corresponding block in the machining program is highlighted.
Conversely, if you select a block in the machining program, the line segments in the machining shape corresponding to the block is highlighted.

■ Select a line segment in the machining shape

(1) Select the [Program Link.] command. ➡ The [Program Link.] command is highlighted.
The sub-display switch button [Display program] is selected, and the machining program appears in the sub-display.
If the machining program is not displayed in the sub-display, the display is automatically switched to the machining program.



(2) Select a line segment in the machining shape. ➡ The line segment you selected is highlighted, and the corresponding block in the machining program is highlighted.
The line segment for one block in the machining program is highlighted.



(*) Even if you switch the sub-display to another sub-display during [Program Link.], [Program Link.] remains enabled.

(*) When the selected line segment in the machining shape corresponds to a block in the main program, the main program is displayed in the program display sub-display. When the selected line segment in the machining shape corresponds to a block in the subprogram, the subprogram is displayed. You cannot switch between the main program and the subprogram through operation in the program display sub-display.

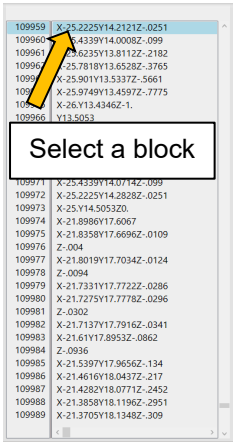
(*) Scrolling using the vertical scroll bar or by the cursor movement is only available within the displayed program.

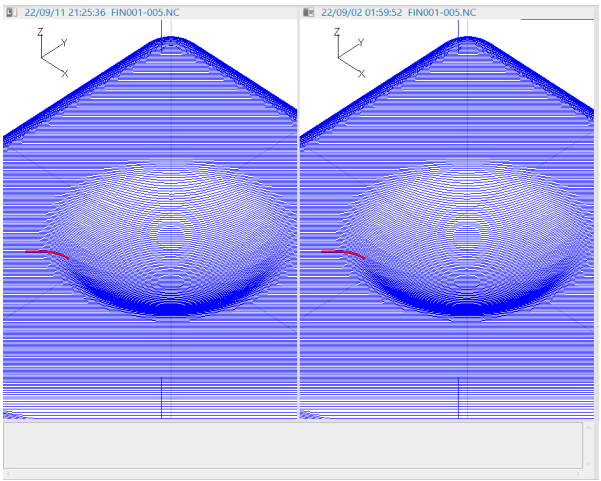
(*) When two screens are displayed, the line segment in the machining shape is highlighted only on the screen where the line segment is selected.

■ Select a block in the machining program

(1) Select the [Program Link.] command. → The [Program Link.] command is highlighted. The sub-display switch button [Display program] is selected, and the machining program appears in the sub-display.

(2) Select a block in the machining program. → The line segments in the machining shape that correspond to the selected block are all highlighted on both screens.





(*) If the block you selected in [Program Link.] is executed repeatedly by means of GOTO, etc., the line segments corresponding to the block executed first are highlighted.

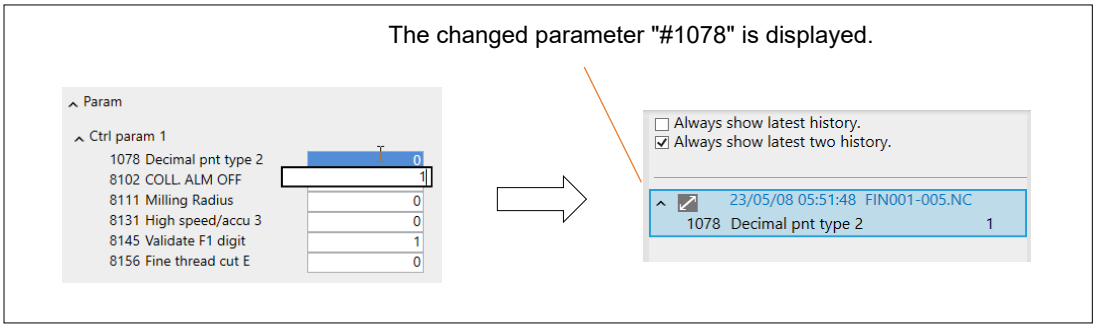
9.3 Caution

(1) Only parameters changed in the sub-display "NC Data List" are displayed in the machining conditions of the history displayed in the sub-display "Simulation History".
Parameters changed by G10 command are applied to the sub-display "NC Data List", but are not applied to the machining conditions of each history displayed in the sub-display "Simulation History".

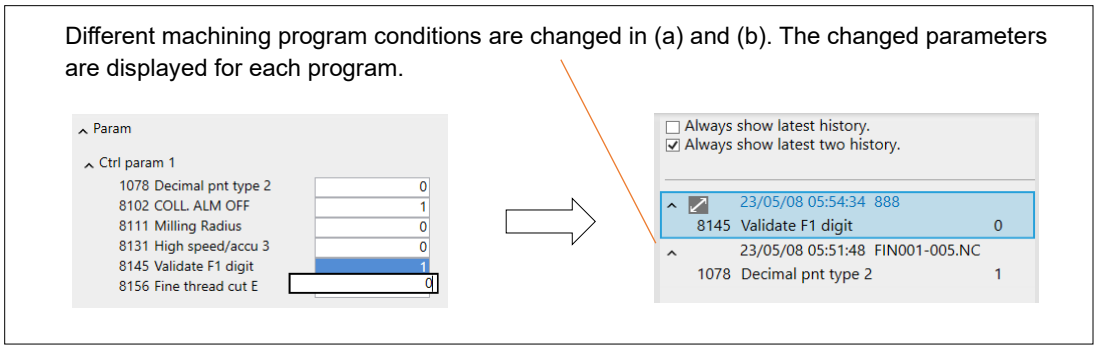
(2) Simulation histories that can be displayed on the right and left screens are those where both file path and file name of the machining program used for simulation are the same.
When contents such as the line number in the machining program and G code are different but both file path and file name of the machining program are the same, the simulation history is to be displayed on two screens. Therefore, the machining program displayed in the sub-display "Display program" may be inconsistent.

(3) In simulation history, machining conditions executed in a machining program with the same full path are displayed comparatively by going back to the past.
Differences of machining conditions simulated in different machining programs are not displayed.
When machining programs have the same file name but different paths, they are treated as different machining programs.
The following is a display example of machining conditions.

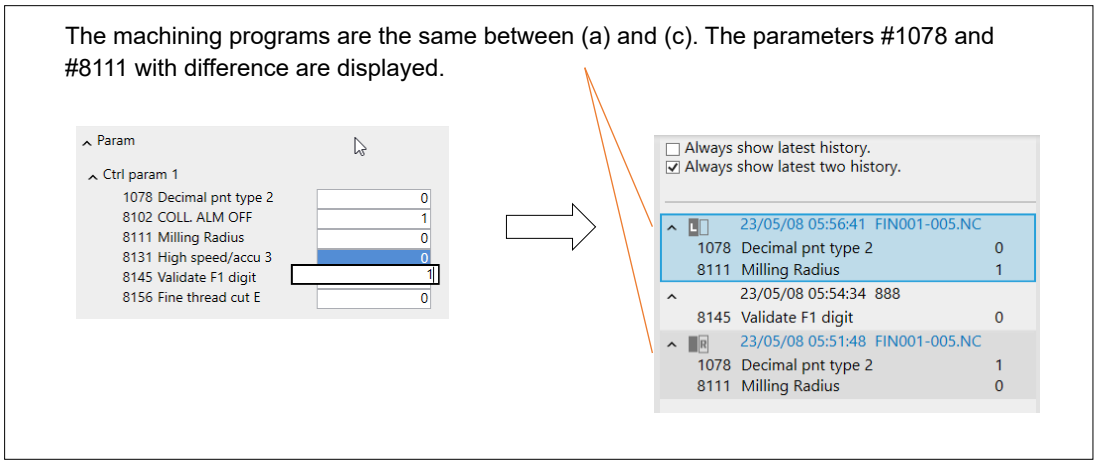
(a) Change the setting value of "#1078 Decimal pnt type 2" in the machining program "FIN001-005.NC" and run the simulation.



(b) Change the setting value of "#8145 Validate F1 digit" in the machining program "888" and run the simulation.



(c) Change the setting value of "#8111 Milling Radius" in the machining program "FIN001-005.NC" and run the simulation.



(4) In simulation history, CAD model error analysis is performed based on the product CAD model currently selected in machining simulation or analysis simulation and its display position.
To perform CAD model error analysis for a product CAD model different from the currently selected one and its display position, select another product CAD model file with the [Select CAD] command in machining simulation or analysis simulation.

(5) Simulation history settings and [Lock History] and [Unlock History] settings are retained even when a project is not saved.

(6) Simulation histories of machining programs with matching full paths can be compared on two screens. The full path of a machining program can be checked by positioning the mouse pointer on the title of the simulation history list to display it as a tooltip.

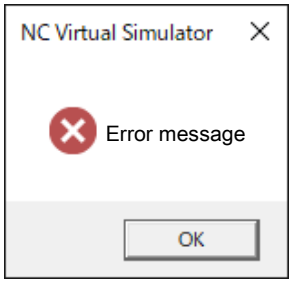
(7) When five or more digits are displayed after the decimal point according to the setting of the parameter "#1003 iunit", the setting value of the parameter in the machining conditions may not be completely displayed in the sub-display "Simulation History". In this case, extend the sub-display area to check the setting value.

(8) To use a simulation history created with NC Virtual Simulator version A4 or earlier in a version A5 or later, launch the simulation again using the same conditions. If it is used without launching the simulation, operations during program link may not function correctly.

10. Messages

10.1 Dialog messages

The message displayed in a dialog message is preceded by a symbol that indicates the type of message (error, warning, selection, information).



	Type	Description
	Error	Indicate that a problem has already occurred. Take the corrective action described in the table below.
	Warning	Indicate that a problem may occur.
	Selection	Indicate that there is more than one option.
	Information	Provide relevant information.

Error message	Description	Corrective action
No folder with the data was found.	The folder with the data that was selected during installation cannot be referenced.	Install NC Virtual Simulator again.
Enter a project name.	No project name is entered.	Correct the project name.
The project name you have entered is too long. Ensure that the project name is not longer than 80 characters (A double-byte character is counted as two characters).	The entered project name is too long.	
The project name that you have entered already exists. Specify a different project name.	A project with the same name as the one you entered already exists.	
A project name cannot consist of spaces only.	The project name contains only spaces.	
The project name that you have entered contains an invalid character (\ / : * ? < > | ").	The entered project name contains an invalid character.	
The project name that you have entered is not allowed. Specify a different project name.	The project name contains a reserved word of Windows.	

Error message	Description	Corrective action
The project name must not begin or end with a space or a period. Correct the project name.	The project name begins or ends with a space or a period (.).	
Enter a name.	No name is entered.	Correct the project name or folder name.
The name you have entered is too long. Ensure that the name is not longer than 80 characters (A double-byte character is counted as two characters).	The entered name is too long.	
The project name or folder name that you have entered already exists. Specify a different name.	A project or folder with the same name as the one you entered already exists.	
A name cannot consist of spaces only.	The name contains only spaces.	
The name that you have entered contains an invalid character (\ / : * ? < > | ").	The entered name contains an invalid character.	
The name that you have entered is not allowed. Specify a different name.	The name contains a reserved word of Windows.	
The name must not begin or end with a space or a period. Correct the name.	The name begins or ends with a space or a period (.).	
The folder that you have selected is either not the backup data of a machine tool or for an unsupported model. Ensure that the folder contains the correct data.	The folder that you have selected is either not machine tool's backup data or for an unsupported model.	Specify the folder where machine tool's backup data is saved.
Communication failed. Could not connect to the NC for one of the following reasons. * The NC does not exist. * The NC is not powered on. * Communication failure.	NC Virtual Simulator cannot connect to the NC.	Correct the IP address of the machine tool to be connected.
The project cannot be renamed because it is open. Close the project.	The project or folder you are trying to rename contains the currently open project.	Close the project.
The project cannot be deleted because it is open. Close the project.	The project or folder you are trying to delete contains the currently open project.	
Failed to start the Internet browser.	There may not be an Internet browser installed.	Establish the Internet access environment to access Mitsubishi Electric FA Global Website.

Error message	Description	Corrective action
Failed to launch the application. Failed to save the application settings. Failed to create a project. Failed to create a folder. Failed to rename the project or folder. Failed to delete the project or folder. Failed to save project.	The requested process failed due to the following reasons: ・ Corrupted execution file ・ Insufficient hard disk space ・ Insufficient memory ・ The destination project file (*1) is write-protected (*1) The destination project file is the file stored in the "project name" folder under the project storage folder (Default "C:\MELCNC\NCVS\Project)".	・ Close the other applications, and restart. ・ Reboot the PC. ・ Install NC Virtual Simulator again. ・ Check the free space on the hard disk. ・ Check the hardware specifications of the PC used. ・ Check whether the destination project file is write-protected.
Cannot open the project because it is not valid.	The files in the project folder are corrupted.	If you have the backup data of that project, open the project using it. Create a project again.
Failed to read the file.	The shared folder is not connected.	Connect to the shared folder from Windows Explorer.
Illegal machine model.	There are no machine configuration files in the selected folder.	Contact the machine tool builder.
The machine model name is too long. Ensure that the machine model name is not longer than 20 characters.	The machine model name is too long.	Contact the machine tool builder.
Tool number not found.	The specified tool number is not set in the tool data.	Set the tool number.
Are you sure you want to delete the tool data?	The tool data is about to be deleted.	Confirm that it is OK to delete it.
Tool data is already set. Do you want to overwrite it?	There is a tool set for the tool data.	Confirm that it is OK to overwrite it.
No tool data exists in the project you have selected.	No tool data has been set in the selected project.	Close the project and open it again to check the tool data. Select another project.
Tool data is already set at bottom line. Do you want to remove it?	Tool data is set in the line that is to be deleted when you insert a line.	Confirm that it is OK to delete it.
Tool Number is missing.	No tool number has been entered in the tool number input dialog.	Enter a tool number.
Tool Number is duplicate.	The tool data of the same tool number as the one to be added by Line paste exists.	Set another tool number.
Tool Number is out of range.	The tool number of the tool data to be added by Line paste is not within the range.	Set a tool number that is within the range.
Failed to register the license file. Please check: whether the selected license file was deleted whether the selected license file is being used by another process.	The selected license file could not be registered.	・ Check whether the selected license file was deleted. ・ Check whether the selected license file is being used by another process.

Error message	Description	Corrective action
The license files for other features cannot be registered because the license for NC Virtual Simulator is not valid.	License files other than NC Virtual Simulator cannot be registered.	Register the license for NC Virtual Simulator.
The license file could not be registered because the expiration date is past.	The selected license file has expired.	Purchase a new license.
The license file could not be registered because it contains an unsupported feature.	The selected license file contains an unsupported feature.	Upgrade NC Virtual Simulator.
The license for NC Virtual Simulator has expired.	The license for NC Virtual Simulator has expired.	To continue to use NC Virtual Simulator, purchase a new license.
The following license has expired. <license type>	The license for the displayed feature has expired.	To continue to use the feature, purchase a new license.
Cannot register a license file that will expire before the expiration date of the registered license.	A license file that will expire before the expiration date of the registered license cannot be registered.	Use a license file that will not expire before the expiration date of the registered license.
There is a problem with the registered license file.	・ The license file has been corrupted. ・ A license file issued for another PC is used.	Use the correct license file issued for the PC you are using.
	・ The Ethernet port or wireless LAN of the PC have been disabled. ・ The connection state of the Ethernet adapter has changed from that of when the license was applied.	・ Enable the Ethernet port or wireless LAN of the PC that was disabled. ・ Set the connection state of the Ethernet adapter the same as when the license was applied.
NC Virtual Simulator cannot be used because the Windows time has been changed.	NC Virtual Simulator cannot be used because it is recognized that Windows time has been changed improperly.	Contact the service center.
CAD model import failed.	The selected product CAD model cannot be displayed due to the following reasons: - Corrupted product CAD model file - Insufficient memory	Check the output contents of the product CAD model.
Invalid file path.	The selected file path exceeds 256 characters. ・ Program file selected by [Select Program] ・ Product CAD model file selected by [Select CAD] ・ Workpiece shape file selected by [Workpiece shape file] of the sub-display "Machining simulation settings" ・ Workpiece shape file exported by [Export Work Form] of the sub-display "Machining simulation settings"	Change the location to store the following files. ・ Machining program ・ Product CAD model file ・ Workpiece shape file

Error message	Description	Corrective action
Invalid file name.	The file name exceeds 32 characters. ・ Program file selected by [Select Program] ・ Product CAD model file selected by [Select CAD] ・ Workpiece shape file selected by [Workpiece shape file] of the sub-display "Machining simulation settings" ・ Workpiece shape file exported by [Export Work Form] of the sub-display "Machining simulation settings"	Change the following file names. ・ Machining program ・ Product CAD model file ・ Workpiece shape file
Invalid directory name.	A directory name that exceeds 48 characters exists in the selected file path. ・ Program file selected by [Select Program] ・ Product CAD model file selected by [Select CAD] ・ Workpiece shape file selected by [Workpiece shape file] of the sub-display "Machining simulation settings" ・ Workpiece shape file exported by [Export Work Form] of the sub-display "Machining simulation settings"	Change the name of the directory where the following files are stored. ・ Machining program ・ Product CAD model file ・ Workpiece shape file
Failed to select the machining program.	・ The selected file is in a format that cannot be used as a machining program. ・ The virtual folder (C:\MELCNC\NCVS\HD) is an actual folder. ・ A machining program is selected through the network.	・ Check the contents of the selected machining program file. ・ Delete the HD of the actual folder (C:\MELCNC\NCVS\HD) or change the name. ・ A machining program cannot be selected through the network. Select a machining program on the local disk.
After saving the project, the project will be reloaded. Are you sure you want to continue?	After saving the project setting, the project is reloaded.	Confirm that it is OK to save the project setting and reload the project.
Are you sure that you want to overwrite the machining time prediction setting?	The machining time prediction setting value is already set, but it is tried to be overwritten.	Conform that it is OK to overwrite it.
Path (*1) Path does not exist. Check the path and try again.	The path specified in the dialog does not exist. ・ Program selection dialog ・ Product model selection dialog ・ Workpiece shape selection dialog ・ Workpiece shape export dialog	Check the folder name and existence of the folder, and then try again.

Error message	Description	Corrective action
File name (*1) does not exist. Verify that the file name is correct.	The file specified in the program selection dialog does not exist.	Check the machining program file name and existence of the file, and then try again.
File name (*1) File not found. Check the file name and try again.	The file specified in the dialog does not exist. ・ Product model selection dialog ・ Jig model selection dialog ・ Workpiece shape selection dialog ・ Workpiece shape export dialog	Check the file name and existence of the file, and then try again.
No CAD model file is selected.	A product CAD model is tried to be displayed while no product CAD model file is selected.	Select a product CAD model.
No workpiece shape file is selected.	The workpiece is tried to be initialized when "Arbitrary shape" is selected in "Work Form" and no workpiece shape file is selected.	Select a workpiece shape file.
Failed to load jig model.	The selected jig model cannot be displayed due to a corrupted jig model file, insufficient memory, etc.	Check the output contents of the jig model.
Failed to load workpiece shape.	The selected workpiece shape could not be loaded due to the workpiece shape file being corrupted, etc.	Check the workpiece shape file.
Export failed because the size of Work Form is out of range.	The size of the workpiece shape is out of the range.	Change the width setting of the workpiece.
A CAD model file that is not in STEP format is selected.	A product CAD model file that is not in STEP format is selected.	Select a product CAD model file in STEP format.
Canceled loading CAD model.	The [Cancel] button is clicked while the product model file is being read.	Select a product mode again as needed.
Are you sure you want to apply the following machining condition to the current project? YY/MM/DD HH:MM:SS EIA NAME (*2)	The parameter in the machining condition of the selected simulation history is set to the parameter in the sub-display "NC Data List".	Confirm that it is OK to set it.
Are you sure you want to delete the selected history?	The selected simulation history is deleted.	Confirm that it is OK to delete it.

(*1) The content input in "Fine name" of the program selection dialog or product model selection dialog is displayed. When it includes "\", it is determined as a path, and when it does not include, it is determined as a file name.

(*2) "YY/MM/DD HH:MM:SS EIA NAME" indicates the title of the history selected from the simulation history list.

10.2 Simulation messages

The following lists the simulation messages that are displayed during simulation.

Message	Description
Simulation stopped	Simulation has been stopped. The process for one block has completed after the click of the [Step] command, but the processing of the machining program has been suspended.
Interference occurred during simulation. ([1]-[2]) Any of the following have entered [1] and [2]. Machine model, Tool, Work, Chuck	A machine, tool, workpiece, or jig interference occurred with the [Interference check valid] checkbox of sub-display "machining simulation settings" checked.
Invalid machine configuration.	The machine configuration file of the imported project is invalid.
No machine configuration file exists.	There are no machine configuration files in the imported project.
In alignment mode	The product CAD model is during alignment.
Out of range.	A value not within the range was set in an input field in the sub-display "machining simulation settings".
This value cannot be set.	An invalid value was set in an input field for sub-display "analysis simulation settings". A value that exceeds the range of the color map display cannot be set.
Analysis is in progress in a mode different from the current setting.	The operation mode of Simulation priority is different from the one at the start of simulation. When two screens are displayed, it is different from the one at the start of simulation on both screens.
Analysis is in progress in a mode other than the current setting on the left screen.	When two screens are displayed, the operation mode of Simulation priority is different from the one at the start of simulation on the left screen.
Analysis is in progress in a mode other than the current setting on the right screen.	When two screens are displayed, the operation mode of Simulation priority is different from the one at the start of simulation on the right screen.
Simulation is running.	NC data setting, machining time prediction setting or redraw cannot be performed because simulation is running.
Redrawing is in progress in a mode different from the current setting.	The setting of [Simulation priority] of when [Redraw] is clicked is different from the one at the start of simulation.
Cannot redraw due to missing data	Redraw cannot be performed because simulation has not been executed since a new project was created.
Unable to restore the partial enlargement display.	The pinpoint zoom display of the saved point of view was not restored because the workpiece size, the shift amount of the reference point of the workpiece, the simulation result, etc. were changed.
No CAD model file is selected.	The process stops because the CAD model error analysis is tried to be performed while no product CAD model file is selected.
A CAD model file that is not in STEP format is selected.	The process stops because the CAD model error analysis is tried to be performed while a product CAD model file that is not in STEP format is selected.

(*) For detailed information about the alarms related to automatic operation, refer to "Alarm/Parameter Manual" of the NC.

10.3 NC alarm messages

Consult the "Alarm/Parameter Manual" for your model.
If multiple NC alarms occur simultaneously, up to ten alarms at a time are displayed.
If an NC alarm message is displayed along with a simulation message, the simulation message is displayed at the top of the message display area.

11. Caution

11.1 General

Please keep in mind the following considerations related to NC Virtual Simulator.

- You cannot open more than one NC Virtual Simulator at the same time.
- You cannot run NC Virtual Simulator and another user support tool (e.g., NC Trainer, NC Trainer2 plus, NC Explorer) simultaneously.
- The result of a floating-point operation may differ between NC Virtual Simulator and the NC unit within a range of rounding error. (The floating-point operation of NC Virtual Simulator is the same as that of the CPU of the PC.)
- If a symbol other than "." is specified as the decimal separator on Control Panel, you may not be able to open or save projects successfully.
  To change the decimal separator, select [Start] > [All apps] > [Windows System Tool] > [Control Panel] > [Clock and Region] > [Region], and click [Additional Settings] button to display "Customize Format" dialog.
- In a project created from [All backup] while tool shape radius designation is set to radius (#8968 = 1), when tools are loaded by [Read NC data] on the Tool Settings window, the tool radius value registered in NC is displayed as the tool diameter value.
- When the PC is under a heavy load, such as when virus scan is run with an anti-virus software, operation of NC Virtual Simulator cannot be guaranteed. Use NC Virtual Simulator while virus scan, etc. is not run.
- When using backup data that is an analog spindle (when the "#3024 sout" setting is 2 to 10) to create a new project, more time than usual may be required to create the project.
- When "#1278 ext14/bit1" (Change miscellaneous command completion method) is "1 (high-speed method)", the machining time prediction may be longer than the actual machining time.
- During the spindle mode of the spindle position control (spindle/C axis control), the machine position of the target axis for spindle position control does not change. The C axis position may differ between NC Virtual Simulator and the NC unit when the spindle mode of the target axis for spindle position control is switched to the C axis mode.
- During the spindle mode of the spindle-mode rotary axis control, the machine position of the target axis for spindle-mode rotary axis control does not change. The servo axis position may differ between NC Virtual Simulator and the NC unit when the spindle mode of the target axis for spindle-mode rotary axis control is switched to the servo axis mode.

11.2 Caution related to machining programs

Please keep in mind the following considerations related to the machining programs to simulate on NC Virtual Simulator.

- Subprograms that can be called by M98 command are those stored in the same folder as the main program, in NC memory and in NC memory 2. As for NC memory and NC memory 2, the subprograms can be called only when NC memory and NC memory 2 are set in the parameter (#8880 or later) for the subprogram storage location.
  When calling a subprogram with the M98 command without specifying ",D" (device number for subprogram), the subprograms stored in the same folder as the main program will be prioritized and searched, regardless of the subprogram search order settings (#8890 and after). When calling a subprogram stored in NC memory or NC memory 2 without specifying ",D", do not store a subprogram with the same name in the same folder as the main program.
- Programs on NC memory such as subprograms called by G65 command and M code macros cannot be called unless they are contained in machine tool's backup data imported to the project.
- G54.4 Workpiece installation error compensation function is not supported. When it is commanded, the program error (P34) occurs.
- Machining condition selection cannot be used in a sample project.
- The skip commands (G31, G31.1, G31.2, G31.3) operate as follows depending on the setting of the parameter "#1258 set30/bit0" (Skip I/F switch).
  When the parameter is set to "0" (A contact: Skip is carried out at rising edge of signal), the movement is performed to the end point of the skip command.
  When the parameter is set to "1" (B contact: Skip is carried out at falling edge of signal), the skip is performed at the same time with the skip command block.
- When subprogram call and return are performed continuously, the edge shape of the corner part in the subprogram call or return may be more similar to the machining program commanded shape than the actual machining shape, depending on the execution timing.
- For a subprogram stored in the same folder as the main program, if the machining is performed between the subprogram and the call source program consecutively (which means, cutting is performed across subprogram calls, execution, and returns to the source), the simulation results near subprogram call or return may be different from the actual machine operation.
- This software does not support the vibration cutting control function (G8.5). If it is commanded, the program error (P39) occurs.

11.3 Caution related to re-display of machining simulation results

Please keep in mind the following considerations related to re-display of the machining simulation results.

- When a project is opened, the last displayed machining simulation result is displayed on the machining simulation window. Therefore, when a project is saved after any of the following operations and the project is opened next time, a machining simulation result different from the project simulation result is displayed.
  - [Apply Cond to project] is performed from the sub-display "Simulation history" on the difference analysis window.
  - Simulation is run on the analysis simulation window.
  - The workpiece is initialized on the machining simulation window after simulation was run.

  To re-display the project simulation result on the machining simulation window, open the sub-display "Machining simulation settings" and click the [Redraw] button.

11.4 Caution related to NC Trainer2 plus link

Please keep in mind the following considerations related to the link between NC Trainer2 plus and NC Virtual Simulator.

- The [Analysis simulation] and [Analyze Difference] tabs cannot be selected during the NC Trainer2 plus link.
- The [Open]/[Save]/[Close] commands of the [File] tab cannot be used during the NC Trainer2 plus link. This means that the project cannot be saved as an NC Virtual Simulator project. In the second and subsequent links, the machining simulation result is not re-displayed.
- The [Start]/[Step]/[Reset]/[Block Skip]/[Ignore M01]/[Side-by-side]/[Select CAD]/[View CAD]/[Pinpoint Zoom] commands of the [Machining simulation] tab cannot be used during the NC Trainer2 plus link. High-speed mode (high)/(medium) of "Simulation priority" and the [Redraw] button in the sub-display "Machining simulation settings" cannot be used.
- The path and file name of the machining program in the memory and memory 2 of NC Trainer2 plus cannot be displayed.
- The machining program in the memory and memory 2 of NC Trainer2 plus is not displayed in the sub-display "Display program".
- The execution time set in the sub-display "Machining time prediction setting" of NC Virtual Simulator is added to the predicted machining time displayed in [MachTime] of the sub-display "Display processing status"; however, it is not added to the machining time displayed on the NC Trainer2 plus screen.
- During the NC Tainer2 plus link, when the [Tool Settings] tab > [Read another proj.] command is clicked, only the linked projects are displayed in the "Read another proj." dialog. When the [Read NC data] command is clicked, the tool data of the linked NC trainer2 plus is read.
- During the NC Trainer2 plus link, when the [Read from another project] button in the sub-display "Machining time prediction setting" is clicked, only the linked projects are displayed in the "Read another proj." dialog.
- During the NC Trainer2 plus link, the display update of the NC Trainer2 plus is prioritized. Depending on the load of the PC, the display update of program processing status, machine position counter, etc. may be delayed on NC Virtual Simulator.
- During the NC Trainer2 plus link, when the tabs of NC Virtual Simulator are changed while cycle start is executed (including the stop status by single block stop or by feed hold stop), response of NC Virtual Simulator may be slow. The operation will be available after a while.
- During NC Trainer2 plus link and during cycle start, do not use the sub-display "Display program" of NC Virtual Simulator. If it is used, the simulation may not be completed.
- During NC Trainer2 plus link, when the simulation stops due to an NC alarm, etc., the machining time on the NC Trainer2 plus screen is updated even during the stop; however, the machining time in the sub-display "Display processing status" of NC Virtual Simulator is not updated.
- When "Machining time prediction setting" is changed in the state of block stop during the NC Trainer2 plus link, the change contents are not reflected immediately. The state is returned to the reset state by completing the simulation, then they are reflected at the time of next or subsequent cycle start.
- When a manual operation such as handle feed is performed in NC Trainer2 plus during the NC Trainer2 plus link, [MachTime] of the sub-display "Display processing status" of NC Virtual Simulator does not include the time set in "Machining time prediction setting".

11.5 Caution related to tool display

(1) When a tool change command is issued, if the setting value of the tool data corresponding to the T number is out of the effective range, the tool is displayed based on the default values.
The values out of the effective range and the default values of the tool data are as follows.

Tool type	Item	Value out of the effective range (mm)	Default value
Ball endmill Flat endmill	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Drill	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Nose angle: C	Less than 2.000, more than 179.999	118 (°)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Radius endmill	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Corner R: C	Less than 0.030, more than 9999.999	1 (mm)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Chamfer	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Tip angle: C	Less than 1.000, more than 179.999	atan(2.0/([Diameter: B] - [Tip dia.: D]))
	Tip dia.: D	Less than 2.000, more than 9999.999	5 (mm)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)

Tool type	Item	Value out of the effective range (mm)	Default value
Tap	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Pitch: C	Less than 2.000, more than 9999.999	5 (mm)
	Root dia.: D	Less than 2.000, more than 9999.999	0.5 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Face mill	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Cutter L: C	Less than 2.000, more than 9999.999	50 (mm)
	Shank D: D	Less than 2.000, more than 9999.999	10 (mm)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	15 (mm)
Tap - UN No. Tap - UN Frac Tap - Pipe Rc Tap - Pipe G Tap - Pipe Rp	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 2.000, more than 9999.999	20 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Boring Back boring	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)
Taper endmill	Length: A	Less than 2.000, more than 9999.999	150 (mm)
	Diameter: B	Less than 1.000, more than 9999.999	20 (mm)
	Corner R: C	Less than 0.000, more than 9999.999	1 (mm)
	Taper: D	Less than 0.100, more than 44.999	10 (°)
	Cut. Edge L: E	Less than 2.000, more than 9999.999	50 (mm)
	Holder H: G	Less than 0.000, more than 9999.999	50 (mm)

Tool type	Item	Value out of the effective range (mm)	Default value
	Holder D: H	Less than 0.000, more than 9999.999	30 (mm)

(2) When a tool change command is issued, the shape data of the tool corresponding to the T number is invalid, the following values are changed and displayed.

Tool type	Item	Condition to be invalid	Value to be changed
Ball endmill Flat endmill	Length: A	Less than [Cut. Edge L: E] + [Holder H: G]	[Holder H: G] = 0
	Cut. Edge L: E	More than [Length: A]	Equal to [Length: A]
	Holder D: H	0	[Holder H: G] = 0
Drill	Length: A	Less than ([Tool radius] x tan(90.0° - ([Nose angle: C] / 2.0))) +[Holder H: G]	(Only the holder is displayed.)
		Less than [Cut. Edge L: E] + [Holder H: G]	[Holder H: G] = 0
	Cut. Edge L: E	More than [Length: A]	Equal to [Length: A]
	Holder D: H	0	[Holder H: G] = 0
Radius endmill	Length: A	Less than [Corner R: C] + [Holder H: G]	(Only the holder is displayed.)
		Less than [Cut. Edge L: E] + [Holder H: G]	[Holder H: G] = 0
	Corner R: C	More than [Tool radius]	(Only the holder is displayed.)
	Cut. Edge L: E	More than [Length: A]	Equal to [Length: A]
	Holder D: H	0	[Holder H: G] = 0
Chamfer	Length: A	Less than (([Tool radius] - [Tip radius]) x tan([Tip angle: C])) + [Holder H: G]	(Only the holder is displayed.)
		Less than [Cut. Edge L: E] + [Holder H: G]	[Holder H: G] = 0
	Tip angle: C	89.9 or more	(Only the holder is displayed.)
	Tip dia.: D	[Diameter: B] or more	Half of [Diameter: B]
	Cut. Edge L: E	More than [Length: A]	Equal to [Length: A]
	Holder D: H	0	[Holder H: G] = 0
Tap	Pitch: C	More than [Length: A]	(Only the holder is displayed.)
	Holder H: G	More than [Length: A]	[Holder H: G] = 0
	Holder D: H	0	[Holder H: G] = 0
Face mill	Length: A	Less than [Cutter L: C] + [Holder H: G]	[Holder H: G] = 0
	Cutter L: C	More than [Length: A], [Tool diameter]/tan(0.1°) or more, [Tool diameter]/tan(89°) or less	(Only the holder is displayed.)
	Cut. Edge L: E	More than [Cutter L: C], [Cut. Edge L: E] = 0	Equal to [Cutter L: C]
	Holder D: H	0	[Holder H: G] = 0
Tap - UN No. Tap - UN Frac Tap - Pipe Rc Tap - Pipe G Tap - Pipe Rp	Holder H: G	More than [Length: A]	[Holder H: G] = 0
	Holder D: H	0	[Holder H: G] = 0
Boring Back boring	Length: A	Less than ([Tool radius]/2.0) + [Holder H: G]	(Only the holder is displayed.)
	Holder D: H	0	[Holder H: G] = 0

Tool type	Item	Condition to be invalid	Value to be changed
Taper endmill	Length: A	Less than [Corner R: C] + [Holder H: G]	(Only the holder is displayed.)
		Less than [Cut. Edge L: E] + [Holder H: G]	[Holder H: G] = 0
	Corner R: C	More than [Tool radius]	(Only the holder is displayed.)
	Cut. Edge L: E	More than [Length: A]	Equal to [Length: A]
	Holder D: H	0	[Holder H: G] = 0

(3) When a tool change command is issued, if the tool cannot be displayed because of inconsistency in tool data, only the holder is displayed. The tool data described in "6.2.1 Registering tool data" is used.

12. Troubleshooting

The following describes common problems related to NC Virtual Simulator and how to troubleshoot them.

12.1 Machine model is not displayed

If the machine model is not displayed when you run machining simulation, try the following:

(1) Open the xml file in the "machine" folder under the project folder (*1), and verify the file name of the geo tag (geo="○○○. stl").

(*1) The project folder is the "project name" folder under the project storage folder (default "C:\MELCNC\NCVS\Project").

(2) Verify that all stl files checked in (1) are in the machine folder.

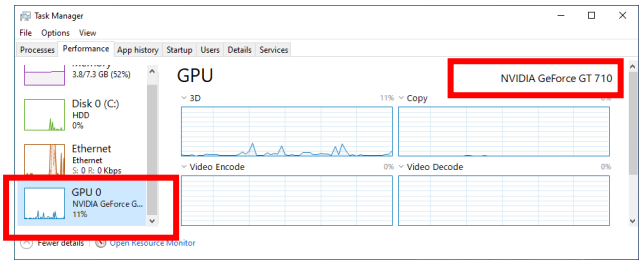
(3) If any file is missing, contact the machine tool builder and obtain the machine model again.

12.2 Simulation takes time depending on PC

If the graphic board of the graphics processor (GPU) mounted on the PC is not correctly recognized, or if the version of the driver is old, simulation may take time. This may be improved by the following steps.

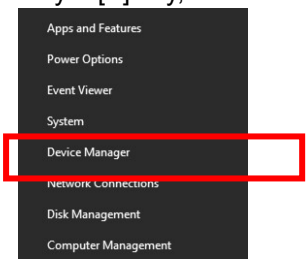
(1) Press [Ctrl] + [Alt] + [Del] keys at the same time and click [Task Manager] to open Task Manager.

(2) Select the [Performance] tab, and confirm that the graphic board type shown in [GPU] is the same as that of your PC.



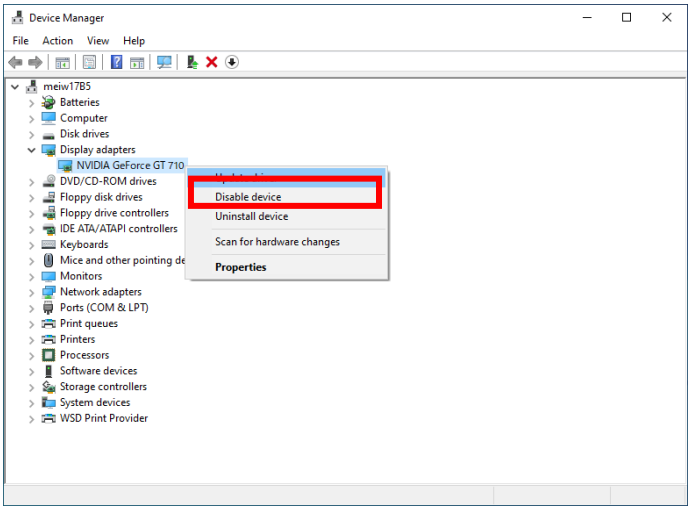
(3) If the graphic board type is not the same, or if [GPU] is not displayed in the Task Manager, obtain and install a driver from the GPU manufacturer's website.
If the graphic board type is the same, update the driver from Device Manager by the following steps.

(a) Press the Windows key + [X] key, and select [Device Manger] in the advanced menu.



(b) Click "Display adapters" to show devices. Select and right-click the device to update.

(c) Click "Update driver".



(d) When "Search automatically for updated driver software" is selected on the "How do you want to search for drivers?" window, driver search starts. When the driver is found, follow the instructions to update.

12.3 Machining condition selection in [NC Data List] cannot be used

Machining conditions in the sub-display "NC Data List" may not be able to be set.

- Even if a value other than "P0" is selected in [Machining cond] of [Tolerance ctrl], the value returns to "P0".
- If values are entered in the parameters "#1206 G1bF" to "#12070 Sfilt2_tol" of [Machining cond P1-Q1] to [Machining cond P3-Q3] and the [Enter] key is pressed, the sub-message "This value cannot be set." appears and the values cannot be set.

These occur for a project created from a machine tool where the machine condition selection is disabled.
Create a project after executing [Execute init.] on the machining condition setting screen of the machine tool.

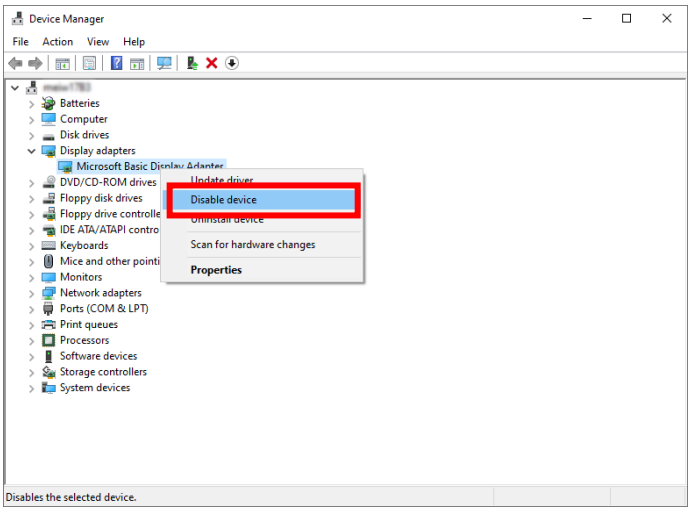
12.4 Macro alarm message appears during simulation and simulation stops

If macro call of miscellaneous command (macro call of M, S, T, B codes) is performed while a machining program is running, "P422 macro alarm message" may appear and simulation may stop.
In this case, check [Disable macro call] of [Simulation priority] in the sub-display "Machining simulation settings" or "Analysis simulation settings", and then run the machining program again.

12.5 Application does not work correctly when Remote Desktop Connection is used

When NC Virial Simulator is operated with Remote Desktop Connection, there are some cases where an application does not work correctly, for example, the application ends incorrectly, the view of machining simulation or analysis simulation is not updated, and so on. This is considered because the graphics driver of the graphics processor (GPU) mounted on the PC is old or the version of OpenGL supported by the graphics driver is 4.4 or earlier.

In this case, update the graphics driver of the graphics processor (GPU).
To update the graphics driver of the graphics processor (GPU), see "12.2 Simulation takes time depending on PC". To check the OpenGL version, you can use a commercially available free software.
If the problem is not solved by updating the graphics driver (the version of OpenGL cannot be updated to 4.5 or later), select [Disable device] in [Display adapters] of [Device Manager].



12.6 Saving or opening a project takes time depending on the project

When a project is saved or ended in NC Virtual Simulator, the last executed simulation result is saved so that the shape can be checked soon when the project is opened next time. Therefore, if a large machining program or a complicated shape machining program where a simulation takes a long time is simulated and the project is ended, it might take time to save the project, end the application and open the project.
To solve this problem, check the [Geometry saving invalid] checkbox of the sub-display "Machining simulation settings".
By doing so, the last executed simulation result is not saved and re-displayed, and it takes less time to save the project, end the application and open the project.

12.7 Application does not work correctly when a product model is selected or displayed by a curved surface in analysis simulation

When the graphics driver of the graphics processor (GPU) mounted on the PC is "AMD Radeon(TM) Graphics", there are some cases where the application ends incorrectly or the curved surface display is invalid in the following cases, for example: the product model is selected on the analysis simulation window and the difference analysis window; the displayed product model is changed to the curved surface display, then another operation is performed.

(Example 1) When the version of "AMD Radeon(TM) Graphics" is "23.19.12", the application ends incorrectly by clicking [Select CAD].

(Example 2) When the version of "AMD Radeon(TM) Graphics" is "21.30.25.01", the product model of analysis simulation is displayed; however the curved surface display is invalid.

If this problem occurs, it might be solved by any of the following methods.

■ **Change the version of the graphics driver.**

Obtain "Radeon Software Adrenalin 21.10.1" from the AMD's official website, confirm the compatibility with your graphics processor (GPU), and then install it.

■ **Disable the graphics driver.**

To disable it, see "12.5 Application does not work correctly when Remote Desktop Connection is used".

If the invalid curved surface display is not still solved, use the wire frame display by unchecking [Curved surface display] in the sub-display "Analysis simulation settings". This does not affect the behavior of the color map display of CAD model error.

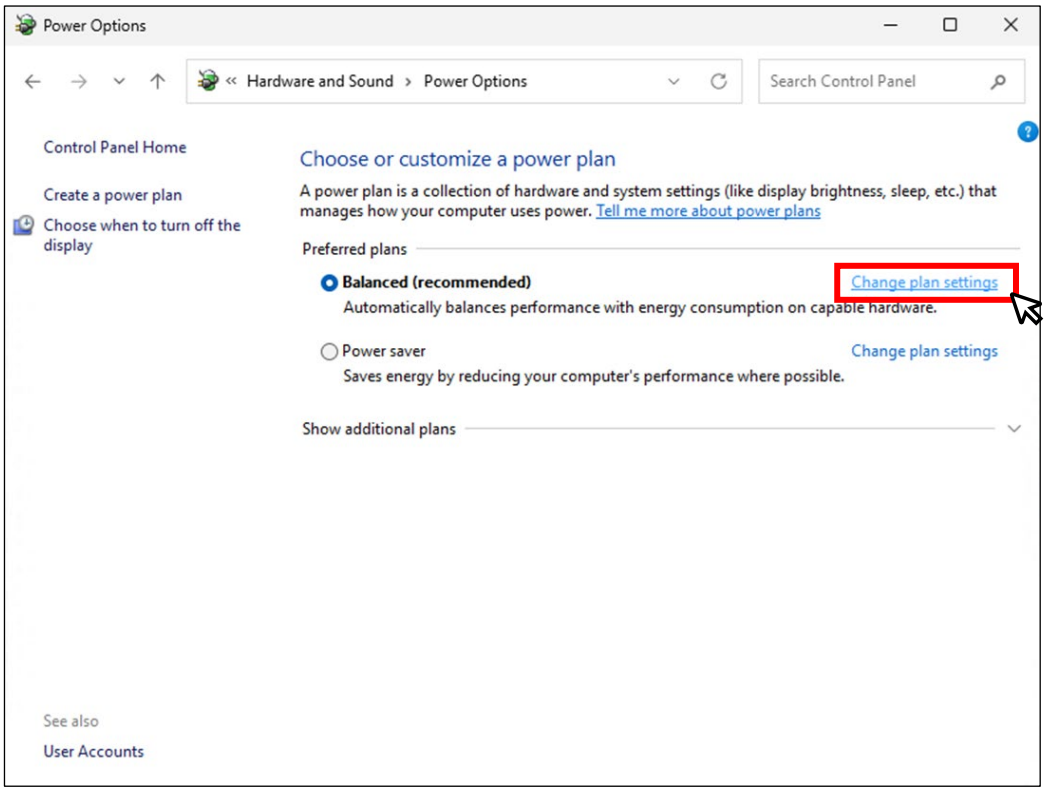
12.8 "There is a problem with the registered license file." is displayed when NC Virtual Simulator is started

The wireless LAN adapter may be disabled unintentionally depending on the personal computer setting. When the wireless LAN adapter is disabled or a new network adapter is added, the network adapter configuration differs from that at license registration. When NC Virtual Simulator is started in this state, the message "There is a problem with the registered license file." is displayed.
When this message is displayed, register the issued license again by referring to "3.2 License registration".

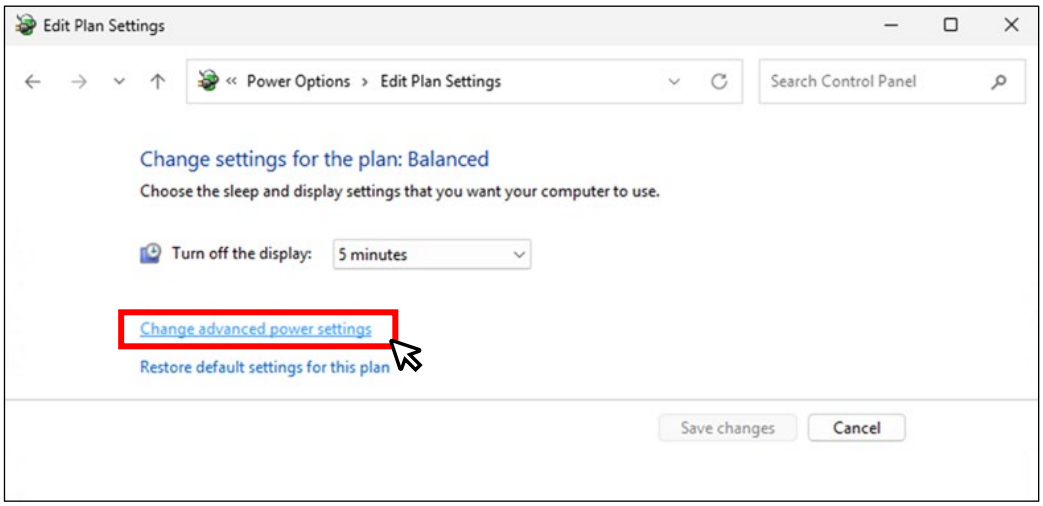
Change the personal computer setting by any of the following (a) to (c). The problem may be solved when NC Virtual Simulator is started next and subsequent times.

(a) Setting of power options of personal computer

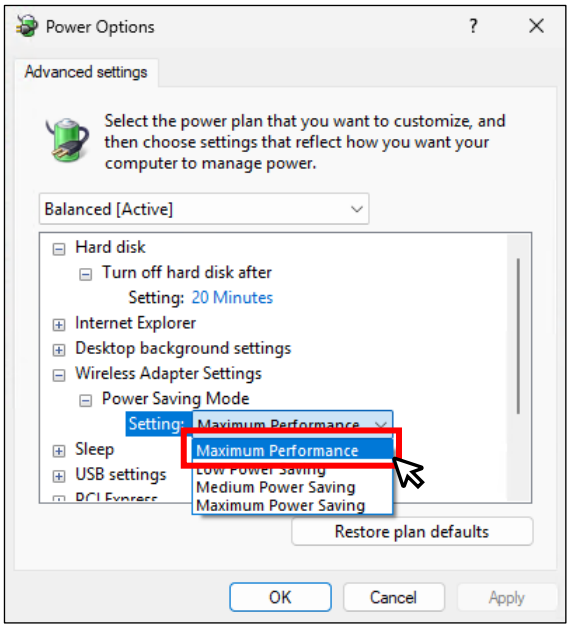
(1) Open "Control Panel".
(2) Click "Hardware and Sound".
(3) Click "Power Options".
(4) Click "Change plan settings" of the selected plan.



(5) Click "Change advanced power settings".



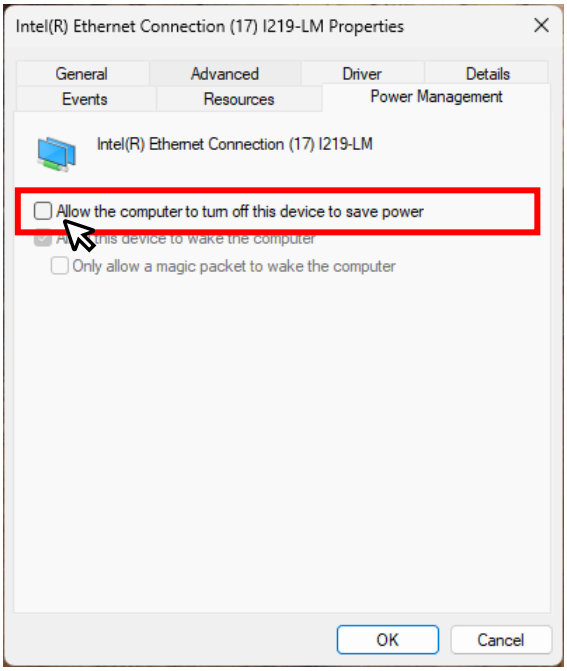
(6) Click "+" of "Wireless Adapter Settings".
(7) Click "+" of "Power Saving Mode".
(8) Select "Maximum Performance" if it is not selected.
When "On battery" and "Plugged in" are displayed, select "Maximum performance" for both items.



(9) Click the [OK] button.

(b) Setting of wireless LAN adapter

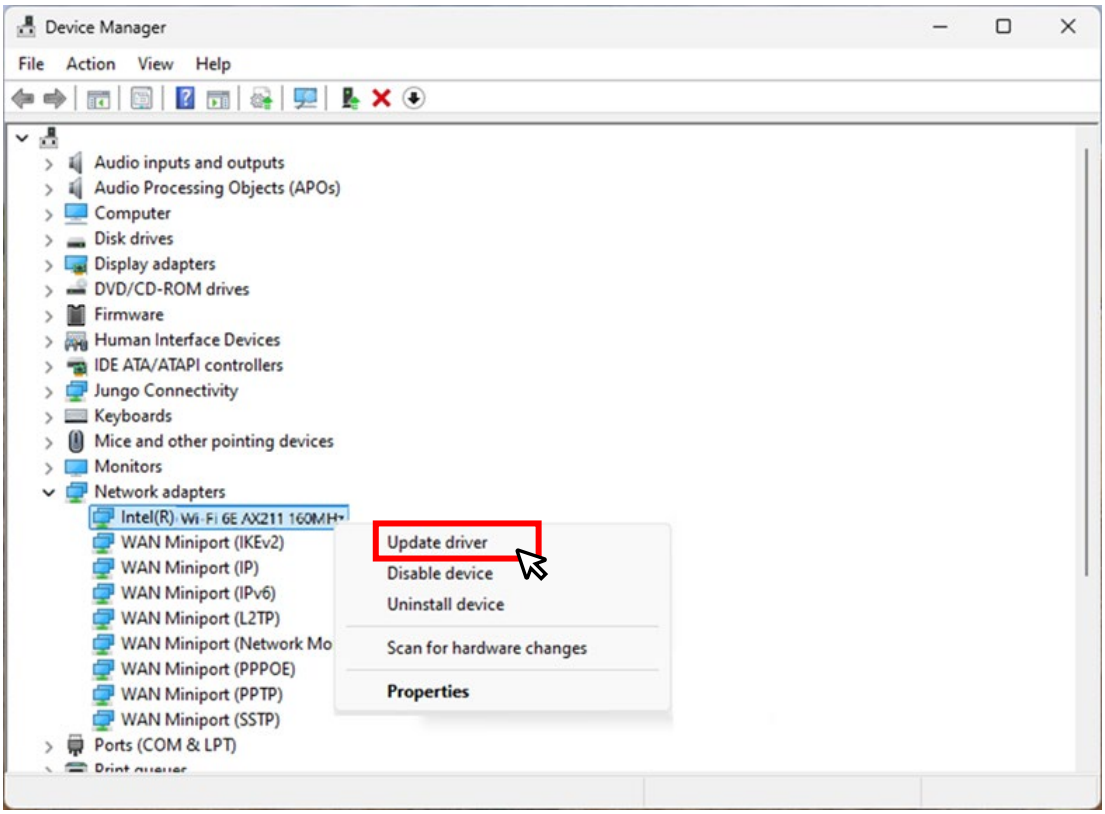
(1) Open "Device Manager".
(2) Click "Network adapters" to show the devices.
(3) Select the wireless LAN adapter, right-click it, and click "Properties".
(4) Select the "Power Management" tab.
(5) Uncheck "Allow the computer to turn off the device to save power".



(6) Click the [OK] button.

(c) Update of wireless LAN adapter

(1) Open "Device Manager".
(2) Click "Network adapters" to show the devices.
(3) Select the wireless LAN adapter, right-click it, and click "Update driver".



(4) When "Search automatically for updated driver software" is selected on the "How do you want to search for drivers?" window, driver search starts. When the driver is found, follow the instructions to update.

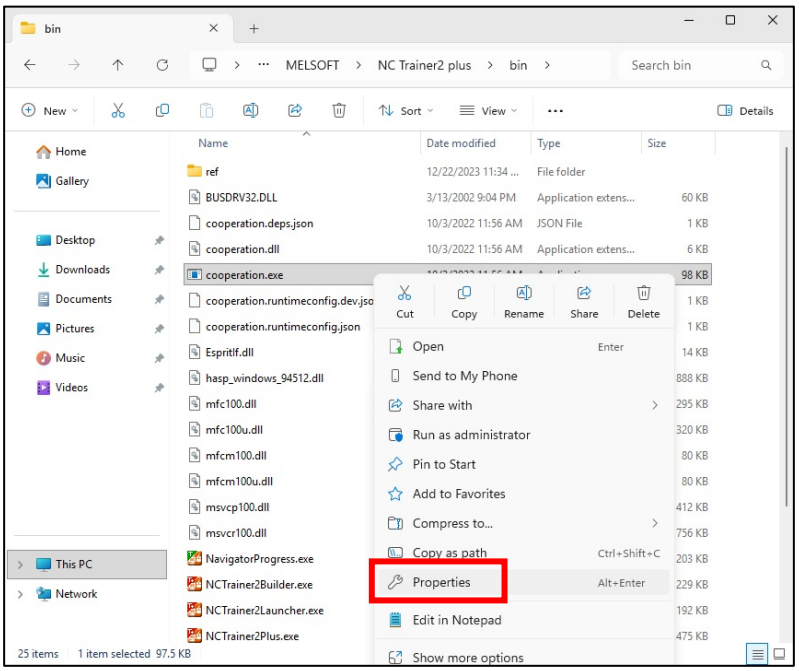
12.9 NC Trainer2 plus link may fail when the update program is applied after upgrading Windows 11 to version 23H2 or later

When the update program is applied to Windows 11 (version 23H2 or later), the error "Failed to activate the NC." is displayed at the start of the NC Trainer2 plus link, and the NC Trainer2 plus link may not be available.

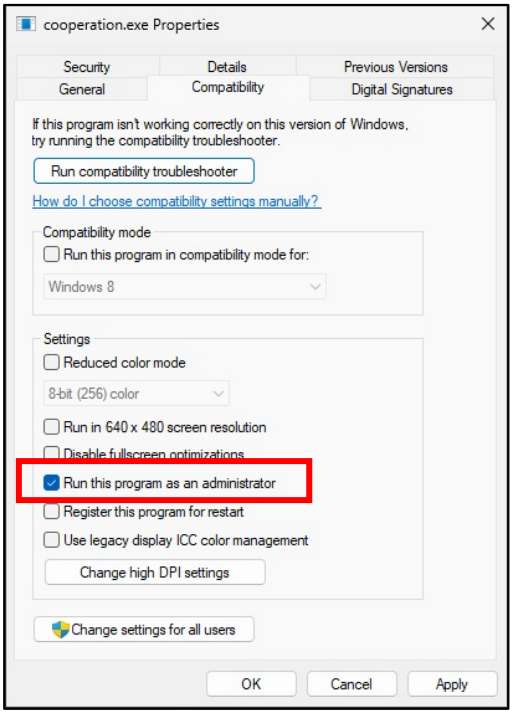
When this error is displayed, take the following measure 1 or 2. The problem may be resolved at the next or subsequent NC Trainer2 plus link.

■ Measure 1: Execute a specific file as an Administrator.

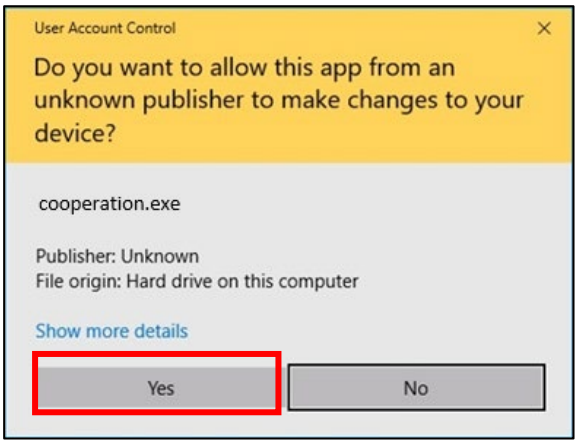
(1) Right-click the file "cooperation.exe" in the "bin" folder under the folder where NC Trainer2 plus is installed ("C:\Program Files (x86)\MELSOFT\NC Trainer2 plus" by default), and open the property.



(2) Open the compatibility tab of the property window, check "Run this program as an administrator" and click [OK].



(3) The confirmation window of the user account control (UAC) appears at the start of the NC Trainer2 plus link.
Select [Yes].



(*) When the personal computer is not used as an Administrator, the administrator user name and password are required to input.

■ **Measure 2: Use an older version of Windows 11 or uninstall the update program.**

When the personal computer is used stand-alone (it is not connected to network and security is less concerned), the problem may be resolved by downgrading the version (*1) as follows.

・ Use a version 22H2 or earlier.

・ To use a version 23H2 or later, uninstall the applied update program.

(*1) The security of the personal computer is not updated. Take necessary protection measures with sufficient consideration for security to use the personal computer.

13. Applicable NC version

NC Virtual Simulator is applicable to the following CNC and versions.

<table>
<tr><th rowspan="3">NC Virtual Simulator</th><th colspan="8">M800V/M80V</th></tr>
<tr><th colspan="4">Machining center system</th><th colspan="4">Lathe system</th></tr>
<tr><th>M800VW</th><th>M800VS</th><th>M80VW</th><th>M80V</th><th>M800VW</th><th>M800VS</th><th>M80VW</th><th>M80V</th></tr>
<tr><td>A1</td><td rowspan="3">A4</td><td rowspan="3">A4</td><td rowspan="4">-</td><td rowspan="3">A4</td><td rowspan="5">-</td><td rowspan="5">-</td><td rowspan="5">-</td><td rowspan="5">-</td></tr>
<tr><td>A2</td></tr>
<tr><td>A3</td></tr>
<tr><td>A4</td><td rowspan="2">A9</td><td rowspan="2">A9</td><td rowspan="2">A9</td></tr>
<tr><td>A5</td><td>A9</td></tr>
</table>

<table>
<tr><th rowspan="3">NC Virtual Simulator</th><th colspan="8">M800/M80</th></tr>
<tr><th colspan="4">Machining center system</th><th colspan="4">Lathe system</th></tr>
<tr><th>M800W</th><th>M800S</th><th>M80W</th><th>M80</th><th>M800W</th><th>M800S</th><th>M80W</th><th>M80</th></tr>
<tr><td>A1</td><td rowspan="3">F8</td><td rowspan="3">F8</td><td rowspan="4">-</td><td rowspan="3">F8</td><td rowspan="5">-</td><td rowspan="5">-</td><td rowspan="5">-</td><td rowspan="5">-</td></tr>
<tr><td>A2</td></tr>
<tr><td>A3</td></tr>
<tr><td>A4</td><td rowspan="2">FC</td><td rowspan="2">FC</td><td rowspan="2">FC</td></tr>
<tr><td>A5</td><td>FC</td></tr>
</table>

Revision History

Revision date	Manual No.	Changes
Nov. 2022	IB(NA)1501743-A	The first edition was created.
Jun. 2023	IB(NA)1501743-B	Contents were revised to correspond to NC Virtual Simulator version A2. The following chapters were added. ・ 4.3.2.7 Machining time prediction setting ・ 5.5 NC Trainer2 plus link ・ 6.3.12 Measure distance on workpiece shape ・ 6.3.14 Save/restore the point of view ・ 6.3.16 Read machining time prediction setting from another project ・ 7.3.5 Display CAD model error ・ 11.3 Caution related to re-display of machining simulation results ・ 11.4 Caution related to NC Trainer2 plus link ・ 12.6 Application does not work correctly when Remote Desktop Connection is used ・ 12.7 Saving or opening a project takes time depending on the project The following chapters were changed. ・ 1. Overview ・ 2.1 System requirements ・ 2.2 Installation ・ 3.1 Types of licenses ・ 4.1 Title bar ・ 4.2 Function menu ・ 4.3.2 Sub-display switch buttons ・ 6.3.10 Superimpose product CAD model on workpiece shape ・ 6.3.11 Adjust display position of product CAD model ・ 6.4 Caution ・ 7.3.3 Turn to line segment path (position) ・ 7.3.4 Turn to color map display ・ 7.4 Caution ・ 9.3 Caution ・ 10.1 Dialog messages ・ 10.2 Simulation messages ・ 11. Caution ・ 13. Applicable NC version The title and contents of the following chapters were changed. ・ 4.3 Sub-display area ・ 4.4 Display area ・ 6.1.2 Machining simulation commands ・ 6.2.2 Setting workpiece data ・ 6.3.6 Change the point of view on workpiece ・ 7.1.2 Analysis simulation commands ・ 9.1.2　Simulation difference analysis commands The structure of the following chapters were changed. ・ 6. Machining simulatio ・ 7. Analysis simulation Other mistakes were fully corrected.

Revision date	Manual No.	Changes
Oct. 2023	IB(NA)1501743-C	Contents were revised to correspond to NC Virtual Simulator version A3. The following chapters were changed. ・ 1. Overview ・ 6.1.2 Machining simulation commands ・ 6.3.9 Switch between machine simulation and workpiece simulation ・ 6.4 Caution ・ 11.1 General
Jun. 2024	IB(NA)1501743-D	Contents were revised to correspond to NC Virtual Simulator version A4. ・ The following chapters were added. ・ 6.3.2 Register jig model ・ 6.3.4 Change the simulation priority ・ 6.3.15 Display cross section of workpiece shape ・ 11.5 Caution related to tool display ・ 12.8 Application does not work correctly when a product model is selected or displayed by a curved surface in analysis simulation The following chapters were changed. ・ 1. Overview ・ 2.1 System requirements ・ 3.1 Types of licenses ・ 4.1.1 Settings ・ 4.3.2.4 Machining simulation settings ・ 4.3.2.5 Analysis simulation settings ・ 5.4 Importing machine model to NC Virtual Simulator ・ 6.1.2 Machining simulation commands ・ 6.2.2 Setting workpiece data ・ 6.3.3 Check interference ・ 6.3.10 Partially enlarge workpiece shape ・ 6.3.12 Superimpose product CAD model on workpiece shape ・ 6.3.14 Measure distance on workpiece shape ・ 6.3.15 Display cross section of workpiece shape ・ 6.3.17 Save/restore the point of view ・ 6.4 Caution ・ 7.1.2 Analysis simulation commands ・ 7.3.5 Display CAD model error ・ 7.4 Caution ・ 8.1 Screen configuration and display items ・ 9.3 Caution ・ 10.1 Dialog messages ・ 10.2 Simulation messages ・ 11.4 Caution related to NC Trainer2 plus link ・ 13. Applicable NC version Other mistakes were corrected.

Revision date	Manual No.	Changes
Sep. 2025	IB(NA)1501743-E	Contents were revised to correspond to NC Virtual Simulator version A5. The following chapters were added. ・ 6.3.20 Specifying machining programs to be skipped ・ 6.3.21 Exporting/importing a workpiece shape ・ 7.3.7 Selecting a tool for drawing ・ 12.8 "There is a problem with the registered license file." is displayed when NC Virtual Simulator is started The following chapters were changed. ・ Introduction ・ 1. Overview ・ 2.2 Installation ・ 2.3 Upgrade ・ 2.4 Uninstallation ・ 4.3.2 Sub-display switch buttons ・ 6.1.2 Machining simulation commands ・ 6.2 Preparation ・ 6.3.2 Register jig model ・ 6.3.3 Check interference ・ 6.3.4 Change the simulation priority ・ 6.3.10 Partially enlarge workpiece shape ・ 6.4 Caution ・ 7. Analysis simulation ・ 7.1 Analysis simulation window ・ 7.3.3 Turn to line segment path (position) ・ 7.3.4 Turn to color map display ・ 7.3.5 Display CAD model error ・ 7.3.6 Turn to plot point highlighting (axis reversal) ・ 7.4 Caution ・ 8.1 Screen configuration and display items ・ 8.3.1 Set tool data ・ 8.3.2 Search tool data ・ 8.3.3 Sort tool numbers ・ 8.3.6 Read NC tool management data ・ 8.3.7 Import tool data of another project ・ 8.4 Caution ・ 9. Simulation difference analysis ・ 9.1.1 Screen configuration and display items ・ 9.2.2 Switch the analysis type ・ 9.2.3 Switch to simulation result of a different machining program ・ 9.2.4 Retain simulation history ・ 9.3 Caution ・ 10.1 Dialog messages ・ 10.2 Simulation messages ・ 11. Caution ・ 11.1 General ・ 11.2 Caution related to machining program ・ 13. Applicable NC version The following chapter was deleted. ・ 12.3 Memory usage reaches upper limit during simulation and simulation stops Other mistakes were corrected.

Revision date	Manual No.	Changes
Feb. 2026	IB(NA)1501743-F	Contents were revised to correspond to NC Virtual Simulator version A5. The following chapter was added. ・ 12.9 NC Trainer2 plus link may fail when the update program is applied after upgrading Windows 11 to version 23H2 or later The following chapters were changed. ・ 2.3 Upgrade ・ 4.3.2.1 Processing status display ・ 4.3.2.3 NC data list ・ 6.3.12 Superimpose product CAD model on workpiece shape ・ 6.4 Caution ・ 7.3.5 Display CAD model error ・ 7.4 Caution ・ 9.3 Caution ・ 11.2 Caution related to machining programs ・ 11.4 Caution related to NC Trainer2 plus link Other mistakes were corrected.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)
HQ and Central Region Service Center (Chicago)
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
Minneapolis, MN Service Satellite
Detroit, MI Service Satellite
Grand Rapids, MI Service Satellite
Cleveland, OH Service Satellite
St. Louis, MO Service Satellite
Indianapolis, IN Service Saltellite

South/East Region Service Center (Georgia)
1845 SATELLITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL +1-678-258-4529 / FAX +1-678-258-4519
Charleston, SC Service Satellite
Charlotte, NC Service Satellite
Raleigh, NC Service Satellite
Dallas, TX Service Satellite
Houston, TX Service Satellite
Hartford, CT Service Satellite
Knoxville, TN Service Satellite
Nashville, TN Service Satellite
Huntsville, AL Satellite
Pittsburg, PA Service Satellite
Tampa, FL Service Satellite
Miami, FL Satelite
Lafayette, LA Service Satellite
Allentown, PA Satelite

Western Region Service Center (California)
5900-B KATELLA AVE. - 5900-A KATELLA AVE. CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650
San Francisco, CA Satelite
Seattle, WA Service Satellite
Denver, CO Service Satellite

Canada Region Service Center (Toronto)
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Queretaro)
Parque Tecnológico Innovación Querétaro, Lateral Carretera Estatal 431, Km 2+200, Lote 91 Modulos 1 y 2
Hacienda la Machorra, CP 76246, El Marqués, Querétaro, México
TEL: +52-442-153-6050
Monterrey, NL Service Satellite
Mexico City, DF Service Satellite

BRAZIL

MITSUBISHI ELECTRIC DO BRASIL COMÉRCIO E SERVIÇOS LTDA.
Votorantim Office
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3023-9000
Blumenau, Santa Catarina Office

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
European Service Headquarters (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-5000 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)
SCHELMENWASENSTRASSE 16-20, 70567 STUTTGART, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center (Paris)
2 RUE DE L'UNION, 92565 RUEIL-MALMAISON CEDEX, FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)
240, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center (Milan)
VIA ENERGY PARK 14, VIMERCATE 20871 (MB) ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy Service Satellite (Padova)
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-288-780 / FAX: +44-1707-278-695

Spain Service Center
CTRA. RUBI, 76-80 8174 SAINT CUGAT DEL VALLES, BARCELONA, SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Hungary Service Center
MITSUBISHI ELECTRIC EUROPE B.V. HUNGARIAN BRANCH
BUDAÖRS OFFICE PARK, SZABADSÁG ÚT 117., 2040 BUDAÖRS, HUNGARY
TEL: +36-70-433-2263

Turkey Service Center
MITSUBISHI ELECTRIC TURKEY ELEKTRİK ÜRÜNLERİ A.Ş
SERIFALI MAHALLESI KALE SOKAK. NO.41 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-969-2500 / FAX: +90-216-661-44-47

Czech Republic Service Center
MITSUBISHI ELECTRIC EUROPE B.V. PRAGUE OFFICE
Pekařská 621/7, 155 00 PRAHA 5, CZECH REPUBLIC
TEL: +420-734-402-587

Sweden Service Center
HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251200 / FAX: +46-8-6251014

Bulgaria Service Center
AKHNATON Ltd. (Service Partner)
4 ANDREJ LJAPCHEV BLVD. POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine Service Center (Kiev)
CSC Automation Ltd. (Service Partner)
4 B, YEVHENA SVERSTYUKA STR., 02002 KIEV, UKRAINE
TEL: +380-44-494-3344 / FAX: +380-44-494-3366

South Africa Service Center
Adroit Technologies (Service Partner)
20 WATERFORD OFFICE PARK, WATERFORD DRIVE, CNR OF WITKOPPEN ROAD,
FOURWAYS JOHANNESBURG SOUTH AFRICA
TEL: +27-11-658-8100 / FAX: +27-11-658-8101

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

PHILIPPINES

MELCO FACTORY AUTOMATION PHILIPPINES INC.
Head Office
128 LOPEZ RIZAL STREET, BRGY., HIGHWAY HILLS, MANDALUYONG CITY , MM PHILIPPINES 1550
TEL: +63-2-8256-8042 / FAX: +632-8637-2294

Philippines Service Center
KM.23 WEST SERVICE ROAD SSH, CUPANG ,MUNTINLUPA CITY, PHILIPPINES
TEL: +63-2-8807-0420 / FAX: +63-2-8842-5202

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO., LTD.
Vietnam Ho Chi Minh Service Center
11TH & 12TH FLOOR, VIETTEL TOWER B, 285 CACH MANG THANG 8 STREET, WARD 12, DISTRICT 10,
HO CHI MINH CITY, VIETNAM
TEL: +84-28-3910-5945 / FAX: +84-28-3910-5947

Vietnam Hanoi Service Center
14TH FLOOR, CAPITAL TOWER, 109 TRAN HUNG DAO STREET, CUA NAM WARD,
HOAN KIEM DISTRICT, HA NOI CITY, VIETNAM
TEL: +84-24-3937-8075 / FAX: +84-24-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA
Indonesia Service Center (Cikarang)
JL. KENARI RAYA BLOK G2-07A, DELTA SILICON 5, LIPPO CIKARANG - BEKASI 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

MALAYSIA

MITSUBISHI ELECTRIC SALES MALAYSIA SDN. BHD.
Malaysia Service Center (Kuala Lumpur Service Center)
LOT 11, JALAN 219, P.O BOX 1036, 46860 PETALING JAYA, SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-7626-5032

Johor Bahru Service Center
9, Jalan Perjiranan 4/6, Bandar Dato Onn, 81100 Johor.
TEL: 07-3642250 / 2256

Pulau Pinang Service Center
25-G, Pusat Perniagaan Perdana Jaya, Jalan Permatang Rawa, 14000 Bukit Mertajam, Pulau Pinang
TEL: +60-4-510-1838 / Fax: +60-4-510-1835

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO., LTD.
Thailand Service Center (Bangkok)
101, TRUE DIGITAL PARK OFFICE, 5TH FLOOR, SUKHUMVIT ROAD, BANGCHAK, PHRA KHANONG,
BANGKOK, 10260 THAILAND
TEL: +66-2-092-8600 / FAX: +66-2-043-1231-33

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
Service Head office and South India Service Center (Bangalore)
CNC eXPerience Park, PLOT NO. 57/A, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
PLOT 517, GROUND FLOOR, UDYOG VIHAR PHASE-III, GURUGRAM 122008, HARYANA, INDIA
TEL : +91-124-463-0300
Ludhiana Service Satellite
Panthnagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite
Manesar Service Satellite

West India Service Center (Pune)
ICC-Devi GAURAV TECHNOLOGY PARK, UNIT NO.402, FOURTH FLOOR, NORTH WING,
SURVEY NUMBER 191-192 (P), NEXT to INDIAN CARD CLOTHING COMPANY Ltd,
OPP. VALLABH NAGAR, PIMPRI, PUNE- 411 018, MAHARASHTRA, INDIA
TEL : +91-20-6819-2274
Kolhapur Service Satellite
Aurangabad Service Satellite
Mumbai Service Satellite

West India Service Center (Ahmedabad)
204-209, 2ND FLOOR, 31FIVE, CORPORATE ROAD PRAHLADNAGAR,
AHMEDABAD -380015, GUJARAT, INDIA
TEL : + 91-79-6777-7888
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
CNC Call Center
TEL: +86-400-921-5130

Shanghai Service Center
NO.1386 HONG QIAO ROAD, CHANG NING QU, SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
Qingdao Service Center
Suzhou Service Center
Wuhan Service Center
Ningbo Service Center
Hefei Service Center
Beijing Service Center
Tianjin Service Center
Xian Service Center
Dalian Service Center
Chengdu Service Center

Shenzhen Service Center
LEVEL8, GALAXY WORLD TOWER B, 1 YABAO ROAD, LONGGANG DISTRICT,
SHENZHEN 518129, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8229-3686
Dongguan Service Center
Xiamen Service Center

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Seoul Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9631 / FAX: +82-2-3664-8668

Korea Daegu Service Center
2F KT BLDG. 8, HOGUK-RO, BUK-GU, DAEGU 41518, KOREA
TEL: +82-53-382-7401 / FAX: +82-53-382-7412

Korea Changwon Technical Center
#205, 888, CHANGWON-DAERO, SEONGSAN-GU, CHANGWON-SI,
GYEONGSANGNAM-DO, 51532, KOREA
TEL: +82-55-261-5559 / FAX: +82-55-261-5558

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO. 8-1, GONGYEQU 16TH RD., XITUN DIST., TAICHUNG CITY 40768 , TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
11F, NO.88, SEC.6, ZHONGSHAN N. RD., SHILIN DIST., TAIPEI CITY 11155, TAIWAN
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan Tainan Service Center
11F-1, NO.30, ZHONGZHENG S. RD., YONGKANG DIST., TAINAN CITY 71067, TAIWAN
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA PTY. LTD.
Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269/ FAX: +61-2-9684-7245

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible. Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

COPYRIGHT 2022-2026 MITSUBISHI ELECTRIC CORPORATION ALL RIGHTS RESERVED

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	NC Virtual Simulator
MODEL CODE	100-873
Manual No.	IB-1501743

IB(NA)1501743(ENG)-F(2602)MEE
Made in JAPAN

Specifications are subject to change without notice.