

NCVIEW 2022

NCVIEW Neo 2022

NCVIEW MC3 2022

NCVIEW / NCVIEW Neo / NCVIEW MC3 2022 Release Notes

A) New Feature Overview

B) Main Improvements and Modifications

(Improvements and modifications after release of NCVIEW / NCVIEW Neo / NCVIEW MC3 are listed)

C) List of Support Reception Numbers

D) Operating Environment

The numbers listed in parentheses () in the descriptions below are Support Reception Numbers

See the Previously reported Support responses.

Marks indicate supported modules:

Neo: NCVIEW Neo, **MC3**: NCVIEW MC3, **SOLID**: NCVIEW SOLIDwatch,

MULTAX: NCVIEW MULTAXwatch, **TURN**: NCVIEW TURNwatch, **TOOL**: NCVIEW TOOLwatch

A) New Features Overview

1. Improved processing speed for cutting simulation and tool interference calculations

(approximately 10 times faster than NCVIEW2020) **Neo** **MC3**

Cutting simulation time reductions of **50% - 90%**.

NCVIEW 2022 has achieved further speedups through improvements to the solid modeler and multi-threaded processing.

The processing speed of tool collision calculations has also been improved, reducing simulation time with and without collision checks.

Simulation time can be reduced by **50% - 90%** compared to NCVIEW2021.

Compared to NCVIEW2021, simulation time can be reduced by **90%** or more for many machining data.

(The % in the simulation time brackets below are values compared to NCVIEW2020.)

■ 3 axis die machining A

Processing time : 122 hour

Simulation times:

NCVIEW2020 : 6 hour 19 min

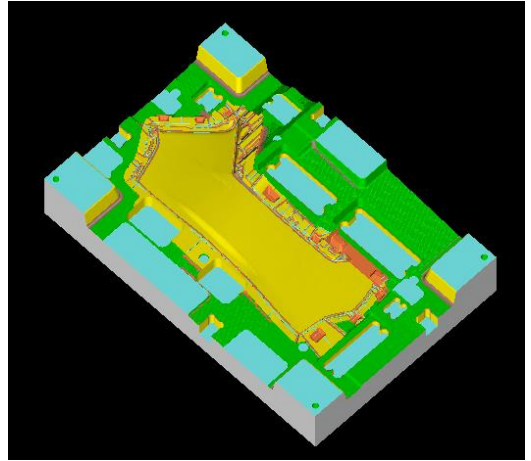
NCVIEW2021 : 1 hour 32 min (80% decrease)

NCVIEW2022 : 45 min (**90% decrease**)

Dimensions : 1350 x 900 x 280

NC data : 13 million lines

Collision check: OFF



■ 3 axis die machining B

Simulation times:

NCVIEW2020 : 4 min 20 sec

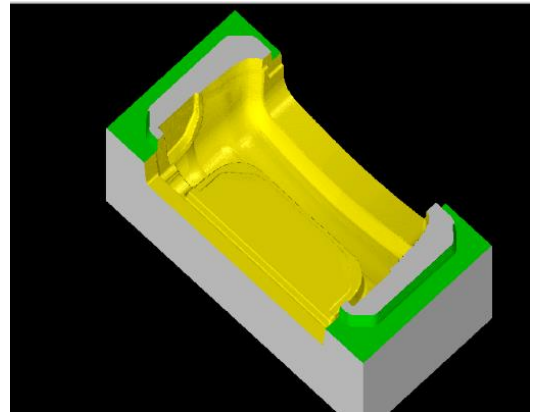
NCVIEW2021 : 1 min 30 sec (65% decrease)

NCVIEW2022 : 23 sec (**90% decrease**)

Dimensions : 2000 x 1000 x 850

NC data : 450,000 lines

Collision check : OFF



■ 4 axis machining

Processing time : 54 hour

Simulation times

NCVIEW2020 : 1 hour 26 min

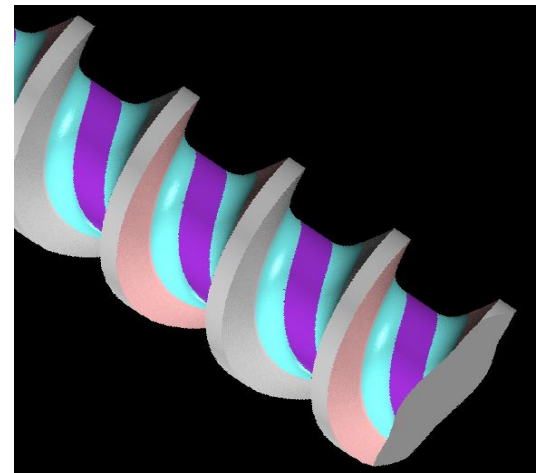
NCVIEW2021 : 52min (40% decrease)

NCVIEW2022 : 5min (**95% decrease**)

Dimensions : 75.2 ϕ x 1250

NC data : 440,000 lines

Collision check: **ON**



■ 5 axis blade machining

Simulation times

NCVIEW2020 : 1 min 41 sec

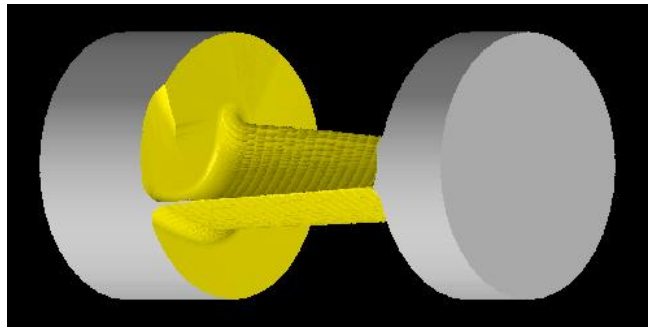
NCVIEW2021 : 19 sec (80% decrease)

NCVIEW2022 : 9 sec (**90% decrease**)

Dimension : 100 ϕ x 200

NC data : 5000 lines

Collision check : **ON**



■ 3 axis die machining C

Processing time : 17.5 hour

Simulation times

NCVIEW2020 : 4 分 25 秒

NCVIEW2021 : 2 min 14 sec (50% decrease)

NCVIEW2022 : 38 sec (**85% decrease**)

Dimension : 670 x 220 x 180

NC data : 300,000 lines

Collision check : **ON**

■ 5 axis impeller machining

Processing time : 13 hour

Simulation times

NCVIEW2020 : 22.5 min

NCVIEW2021 : 8 min (65% decrease)

NCVIEW2022 : 2 min (**90% decrease**)

Dimension : 160 ϕ x 50

NC data : 560,000 lines

Collision check : **ON**

■ 5 axis indexing A

Processing time : 60 min

Simulation times

NCVIEW2020 : 1 min 56 sec

NCVIEW2021 : 15 sec (80% decrease)

NCVIEW2022 : 4 sec (**95% decrease**)

Dimension : 20 ϕ x 200

NC data : 3000 lines

Collision check : OFF

■ 5 axis indexing B

Processing time : 123 min

Simulation times

NCVIEW2020 : 14 min

NCVIEW2021 : 3.5 min (75% decrease)

NCVIEW2022 : 1 min (**93% decrease**)

Dimension : 60 x 180 x 60

NC data : 140,000 lines

Collision check : **ON**

(note) •Simulation reduction time varies depending on processing data and PC performance
•The PC used for measurements differs from the NCVIEW2021 Release Notes

2. Detailed check of drilling cycle operation

A detailed hole drilling cycle check function has been added to the hole drilling report.

By clicking the [Details] button, the following information can be checked for each drilling process.

- Detailed tool drilling operation



Reproduction of drilling tool operation can be confirmed by pressing the playback button.

It is also possible to playback as if you were watching a frame-by-frame movie.

The screenshot displays a software interface for checking drilling operations. On the left is a 'Details' panel with a graph and coordinate tables. On the right is a 3D 'View: 1' window showing a blue drilling tool over a yellow workpiece with a grid of holes.

Details Panel:

Machine coordinate system

	X	Y	Z
I	-15.0000	21.0000	50.0000
R	-15.0000	21.0000	3.0000
Z	-15.0000	21.0000	-9.5000
P1	0.0000	0.0000	0.0000
P2	0.0000	0.0000	0.0000
P3	0.0000	0.0000	0.0000

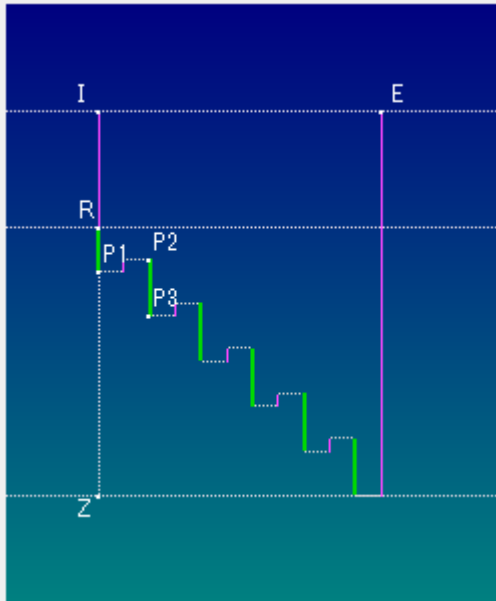
Q [R - P1]	0.0000
B [P1 - P2]	0.0000
Q2 [P2 - P3]	0.0000

Buttons: Report output, Close

- Position of initial point, return point, bottom of hole, etc., depth of cut, amount of clearance

Displays a 2D preview of the drilling cycle movement.

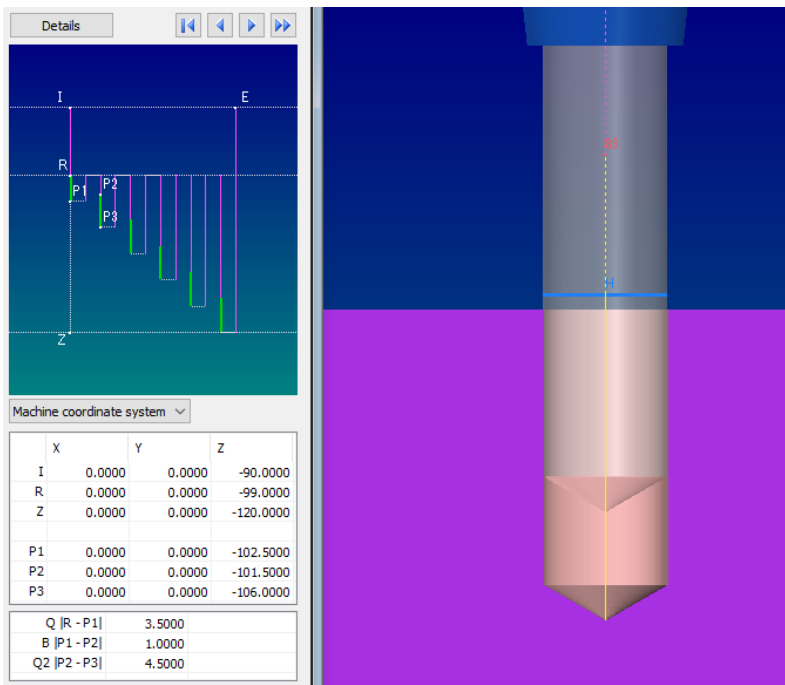
Also displays detailed information such as point locations and depth of cut.



	X	Y	Z
I	0.0000	0.0000	9.0000
R	0.0000	0.0000	0.0000
Z	0.0000	0.0000	-21.0000
P1	0.0000	0.0000	-3.5000
P2	0.0000	0.0000	-2.5000
P3	0.0000	0.0000	-7.0000
Q [R - P1]		3.5000	
B [P1 - P2]		1.0000	
Q2 [P2 - P3]		4.5000	

- Intermittent feed operation

Detailed operation of intermittent feed can be visually confirmed.



(note) Simulation cannot be restarted after detailed display

Supported G codes are as follows

- Machining G code
G73, G74, G76, G81, G82, G83, G84, G85, G86, G87, G88, G89
- Turning G code
G83, G84, G85, G87, G88, G89
- OSP combined processing machine G code
G178, G179, G181, G182, G184, G189

The following operations are not supported

- Shift operation for fine boring and back boring
- IJ command depth of cut in deep drilling cycle
- Movement for back boring relief parameter min.
- OSP lathe-based deep drilling cycle
- Designation of return point by OSP fixed cycle NC command

3. Axis stroke check using machine commands

Axis stroke checks can now be performed with the following machine commands. Please refer to Help for more information about the commands.

AXIS_LIMIT_MINMAX Axis number Minimum value Maximum value

Control to stop at the axis stroke limit in positioning motion is now supported. The minimum and maximum axis stroke values must be set. Please contact us for details. (201406004)

4. Cutting color setting by feed rate

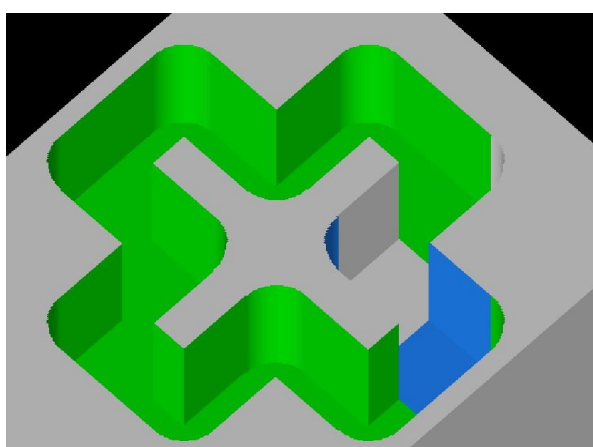
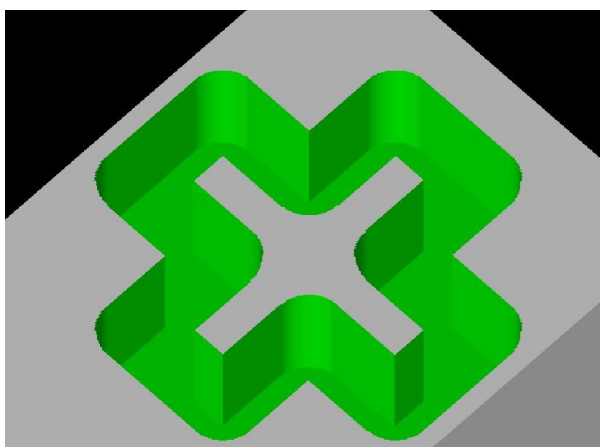
The workpiece cutting color can now be color-coded according to the feedrate command value from [Simulation Setting] - [View] - [Cutting Color Setting by F]. The workpiece cutting color is determined by the color set in the [Cutting Color Setting by F] dialog box, not by the tool color set in the tool registration panel. In the figure below, the workpiece is colored blue if F is less than 3.0 and 1.0 or more is commanded, green if F is less than 1.0, and gray otherwise. This function is enabled only for milling tools.

Enable on Color Settings by Feedrate Out of Range: ▼

<input checked="" type="checkbox"/> <input type="text" value="3.0000"/> Below <input type="text" value="3"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="12"/> ▼
<input checked="" type="checkbox"/> <input type="text" value="1.0000"/> Below <input type="text" value="2"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="13"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="4"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="14"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="5"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="15"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="6"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="16"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="7"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="17"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="8"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="18"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="9"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="19"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="10"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="20"/> ▼
<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="11"/> ▼	<input type="checkbox"/> <input type="text" value=""/> Below <input type="text" value="21"/> ▼

Simulation can be used to check cutting points at unintended feedrates.

Example: Result of cutting with the same NC data while changing a part of the feed rate (only one tool is used). The left figure shows cutting with normal tool color, and the right figure shows cutting with color-coded feed speeds. The right figure shows the results of cutting with different feedrates in the areas colored blue and gray.



B) Main Improvements and Modifications

- Improve the processing time of rapid move
- Improve the processing time of deep drilling cycle (FANUC G73/G83)
- Modify the tool tip control on inclined rotation axis (IJK type)
- Modify the INTEGREGX operation
- Modify the tool registration operation (2002105002)
- Modify the behavior of during coordinate conversion
- Fixed behavior of compound fixed cycles (202103001)
- Fixed mirror operation on OSP (G62) (202106005)
- Fixed rotary table settings
- Fixed HEIDENHAIN sub-program behavior
- Added the machine command: MC_CYCLE83_RETURN_VAL
- Fixed the system variable of OSP, VSAP
- Modify the simulation settings
- Fixed the Cutting History Report behavior (202107001)
- Fixed the Batch-log behavior (202107001)
- Fixed the Hole report behavior
- Fixed a problem with simultaneous circular interpolation and rotation axis commands (202107003)
- Fixed the error check on SIEMENS
- Fixed the turning cycle on FANUC, G74 (202201007)
- Fixed the OSP sub-program behavior
- Fixed the turning cycle on OSP
- Fixed the workpiece cross-section display

C) List of Support Reception Numbers

202202006, 202202005, 202202001, 202201001, 202201001, 202112009, 202112008, 202112004, 202112007, 202112002, 202111006, 202111005, 202111003, 202108003, 202108002, 202107004, 202107001, 202106005, 202106003, 202106002, 202105004, 202105003, 202105002, 202105001, 202104014, 202104012, 202104011

D) Operating Environment

OS	< NCVIEW / NCVIEW Neo 32-bit Version > Windows 10 / Windows 10 64bit ※Also operates on 32-bit if installed on a 64-bit OS < NCVIEW Neo 64-bit Version/ NCVIEW MC3> Windows 10 64bit
Memory	16 GB or higher (64bit) / 2 GB or higher (32bit) recommended
CPU	Intel Core i7 recommended
Graphics	NVIDIA recommended

Copyright (c) 2022 Cimple Technology Inc. All rights reserved.