NCVIEW 2023
NCVIEW Neo 2023
NCVIEW MC3 2023

NCVIEW 2023/ NCVIEW Neo 2023/ NCVIEW MC3 2023 Release Note

A) New Feature Overview

B) Functions planned for future development

C) Main Improvements and Modifications

D) List of Support Reception Numbers

E) Notice of Discontinued Supported Products

F) Operating Environment

A) New Feature Overview

---

1. **Attachment interference check is now faster** （NCVIEW Neo / MC3）

   Performance has been improved to enable high-speed interference checks even when machining while the attachment is in close proximity to the workpiece.
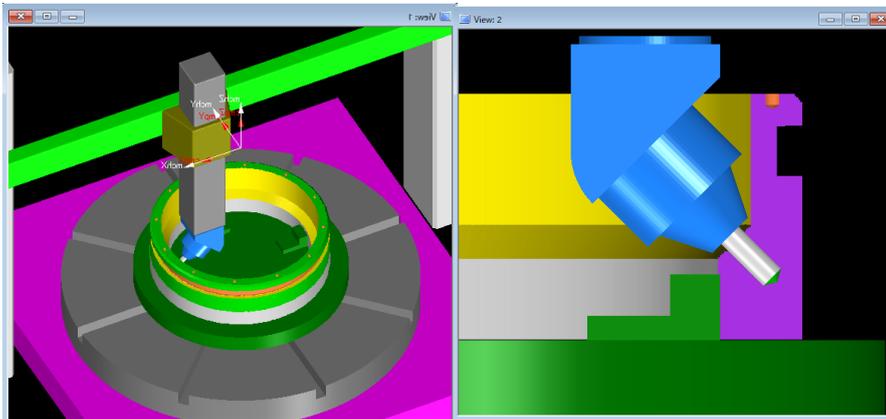
2. **Extended support for sloped surface machining instructions**

   When machining a hole on an inclined surface of a workpiece,

   It is easier to create NC programs if the machining position can be

   commanded in the coordinate system on the inclined plane.

   For NC programs are easier to create.

   Feature coordinate system setting command G68.2

   and tool axis direction control command G53.1

   The range of support for machining of inclined surfaces has been extended.

1
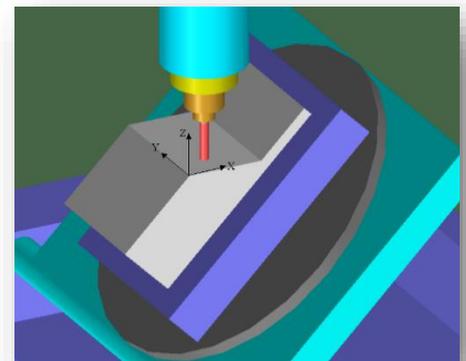
3. **Tool life management systems specific to machine tool manufacturers are now supported**

The function has been extended to flexibly handle cases where T, H, and D commands are commanded by tool life management system numbers.

4. **Support for Windows 11**

All NCVIEW modules are support for Windows 11

5. **Network license is support for Windows Server 2022**

Network license use with Windows Server 2022 may require a new license key. Please contact us for details.

B) Functions planned for future development

We are planning to expand the functionality of the "Macro Debugger," a function that allows macro programs to be analyzed in NCVIEW. As soon as new functions and implemented versions are determined, we will announce them on our website, etc.

C) Main Improvements and Modifications

- Fixed a problem with PDI input
- Fixed positioning behavior during fixed cycle
- Fixed arc command behavior during position offset
- Corrected rotary axis behavior for tool axis direction control
- Fixed model comparison behavior
- Fixed OSP simulation behavior
- Fixed error checking for arc commands in OSP
- The number of tool correction numbers that can be defined has been expanded
- Fixed tap processing simulation behavior
- OSP multitasking machine fixed cycle behavior corrected
- Corrected behavior of automatic correction (length correction)
- Fixed fast-forwarding behavior on additive axes
- Fixed simulation of drilling in OSP
- Tool tip endpoint control (vector command) behavior has been corrected.
- Fixed lathe finishing cycle operation
- Fixed a bug in the drilling detail operation
- Fixed a problem with attachment interference
- Fixed batch simulation behavior

D) List of Support Reception Numbers

202101003 202111003 202212002 202211001 202210001 202210008 202208002 202206006 202206002 202206003 202205001 202202002

E) Notice of Discontinued Supported Products

With the spread of 64-bit OS, support for 32-bit OS versions of the following products has been terminated.

- NCVIEW Neo
- NCVIEW TOOLwatch

Customers currently using the above products on a 32-bit OS are requested to use the 64-bit OS version starting from NCVIEW 2023. We apologize for any inconvenience this may cause and appreciate your understanding.

F) Operating Environment

OS	〈 NCVIEW TURNwatch / SOLIDwatch / MULTAXwatch〉 Windows 10 / Windows 10 64bit / Windows11 ※Also operates on 32-bit if installed on a 64-bit OS 〈 NCVIEW Neo / NCVIEW MC3 / NCVIEW TOOLwatch 〉 Windows 10 64bit / Windows11
Memory	16 GB or higher (64bit) / 2 GB or higher (32bit) recommended
CPU	Intel Core i7 recommended
Graphics	NVIDIA recommended