# NCVIEW 2021
# NCVIEW Neo 2021
# NCVIEW MC3 2021

NCVIEW / NCVIEW Neo / NCVIEW MC3 2021 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、

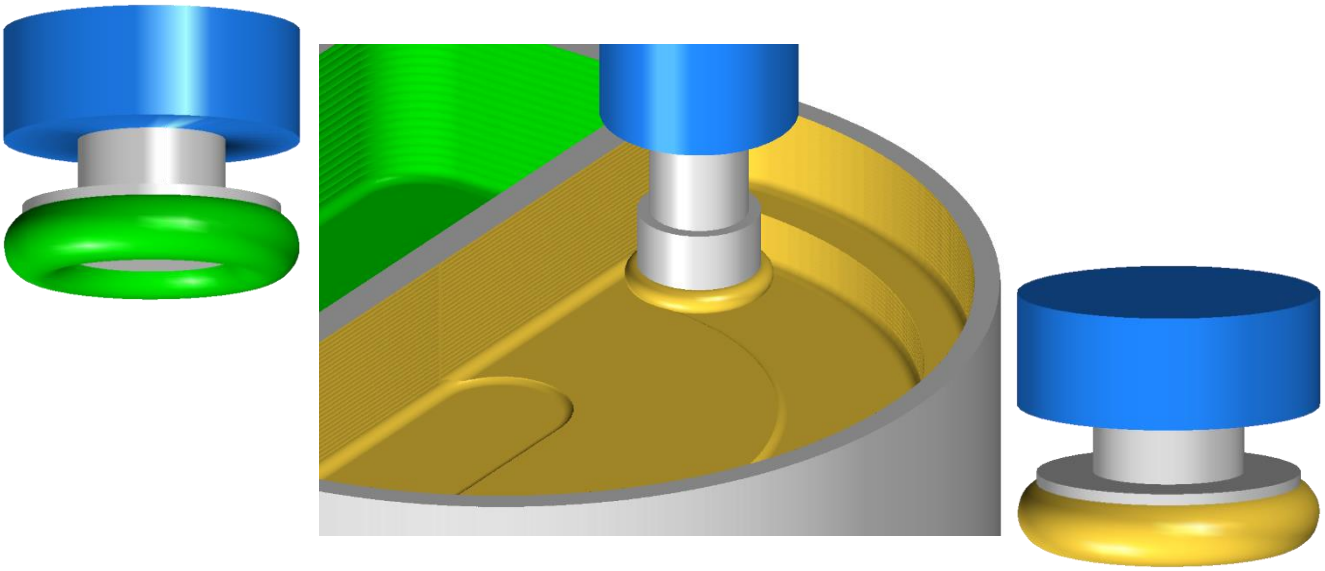
MULTAX: NCVIEW MULTAXwatch、 TURN: NCVIEW TURNwatch、 TOOL: NCVIEW TOOLwatch

## A) New Features Overview

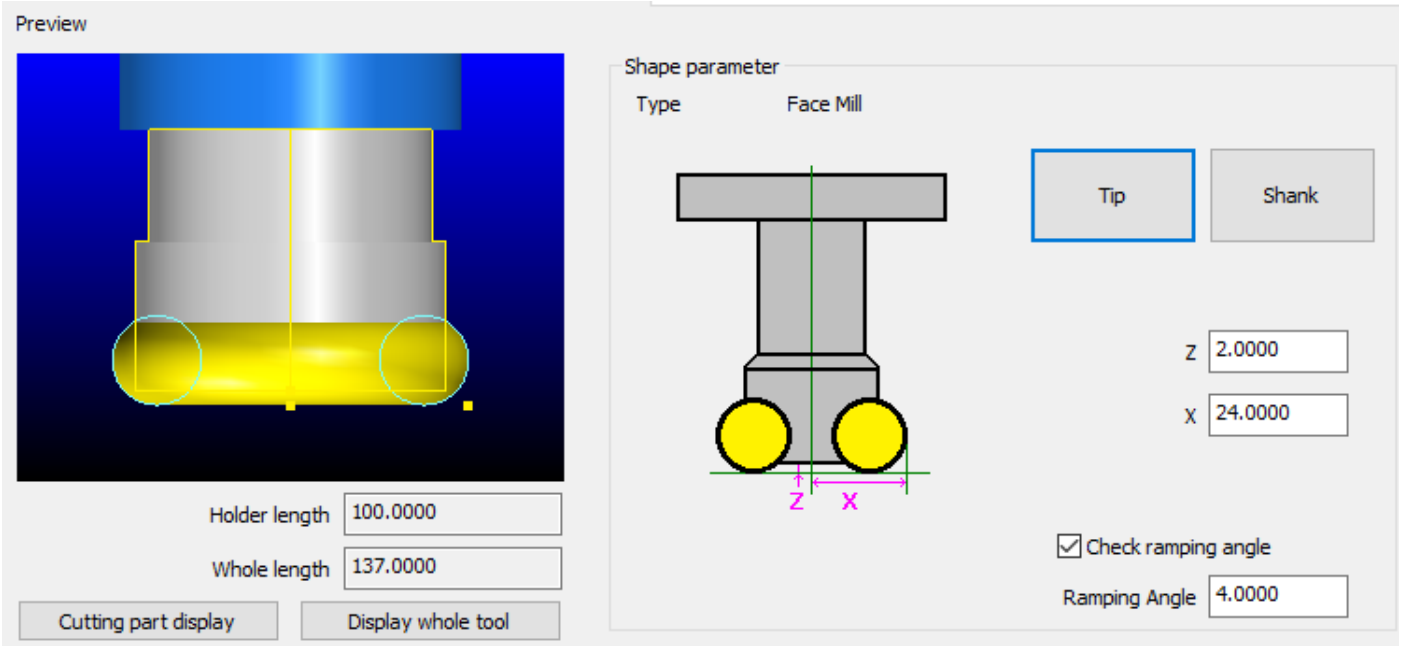
### 1. Added to support Face Mill Neo MC3 TOOL

It possible to simulate Face Mill(round top tip) tools.

The cutting shape is accurately reproduced, and **collision of the tool bottom**, which was difficult to check before, can be verified in advance by simulation. It can also be set **the maximum ramping angle** for each tool and check if the moving angle during machining is within the set angle. Prevents damage to tools and workpieces and enables **safe machining**.



■ **Face Mill Tool settings Dialog**　　Tools can be registered from the following dialog box



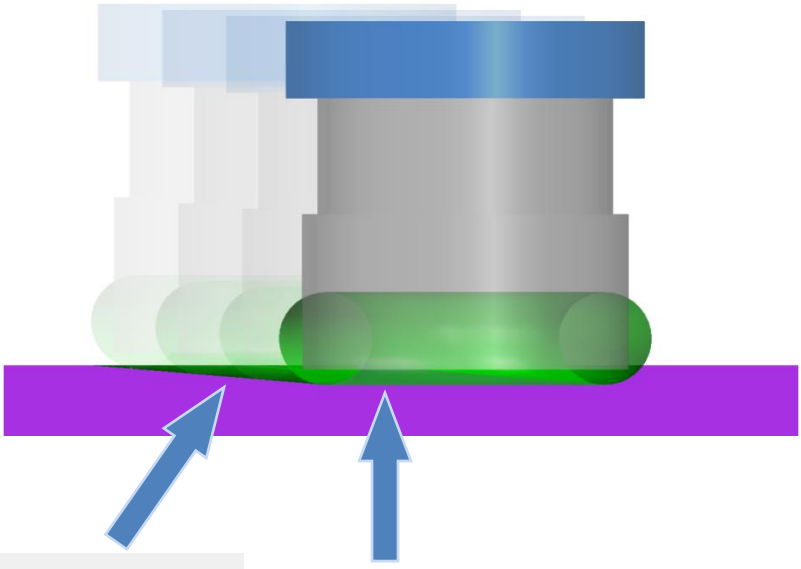

■ Check the ramping process

Detect collision at the bottom of the tool(the part inside the tip)

Check if the direction of movement exceeds the set ramping angle.

Also in case of herical processing, detect collision at the bottom of the tool(the part inside the tip)

※ Face Mill collision check/Ramping angle check are function of NCVIEW Neo / NCVIEW MC3




**Tool bottom collision check**

2. Improved the processing speed of cutting simulation Neo MC3

Cutting simulation is decreased **50%–90%** compared to NCVIEW2020.

■ 3-axis mold machining A

Total processing time：122 hours

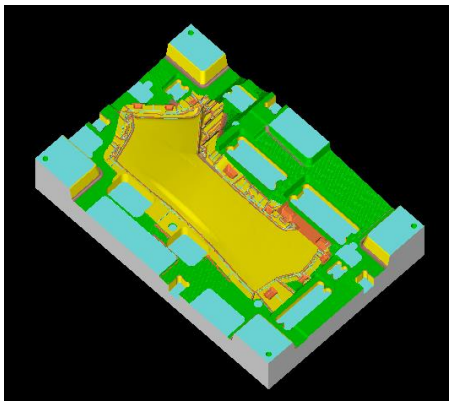
Simulation time

NCVIEW2020：630 minutes

NCVIEW2021：127minutes（**80% decreased**）

Work size：1350 x 900 x 280

NC data：13,000,000 blocks



■ 3-axis mold machining B

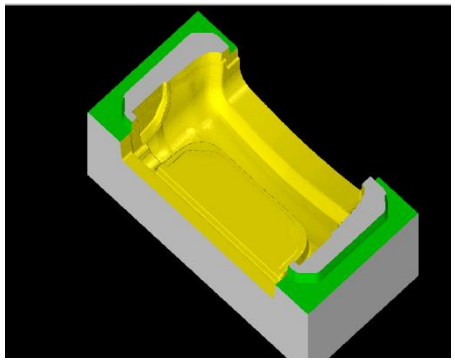
Simulation time

NCVIEW2020：260 seconds

NCVIEW2021：107 seconds（**60% decreased**）

Work size：2000 x 1000 x 850

NC data：450,000blocks



■ Simultaneous 5-axis machining

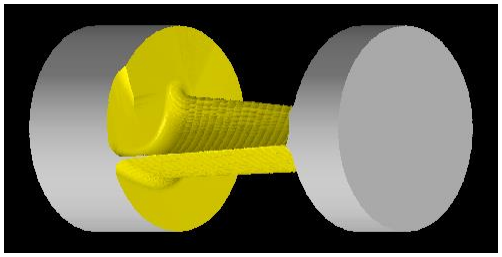
Simulation time

NCVIEW2020：103 seconds

NCVIEW2021：19 seconds（**80% decreased**）

Work size：100$\phi$ x 200

NC data：5,000 blocks



■ Indexing 5-axis machining

Total processing time：60 minutes

Simulation time

NCVIEW2020：176 seconds

NCVIEW2021：18 seconds（**90% decreased**）

Work size：20$\phi$ x 200

NC data：3,000 blocks


(Note) ・Simulation decreased rate varies depending on processing data and PC performance.

・Processing speedup is only for NCVIEW Neo / MC3.

・Total processing time refers to NCVIEW processing report.

3. Expanded OSP controller
   - ■ Added support for OSP program call by variable.

4. Expanded TOSNUC controller
   - ■ Enable to allow “Symbols” to be specified for TOSNUC sub-program arguments.

5. Expanded the fixed cycles related function.
   - ■ Not to change on cycle operation when “Q0” is set in a cycle command such as G83.

6. Expanded Batch command
   - ■ Added Batch commands for setting the 2nd, 3rdand 4th return points.

7. Improved usability
   - ■ Enabled to set animation ON/OFF when NCVIEW starting.

## B) Main Improvements and Modifications

- Fixed the processing of modal group 30.
- Fixed the processing of macro conversion output when the program number is a variable. (20200207)
- Fixed the processing when the BC axis is commanded by an address other than ABC on the universal head.
- Fixed the EXOFS processing of the additional axis.
- Fixed the batch report output.
- Improved the VM display of the fixed attachment (FIX_STL). (202001002)
- Fixed sub-pro call by file name. (202001006)
- Improved rapid traverse path line in DXF output. (201912010)
- Fixed the processing in which the coordinate system does not rotate with the table in the tool tip point control of the tilted rotation axis.
- Fixed the processing of the tool template screen.
- Fixed the incorrect curve output in DXF output of the tool path.
- Improved the coordinate value display of the model comparison result.
- Fixed incorrect output processing in DXF output of the tool path. (202004003)
- Improved the alarm message display of OSP coordinate conversion (G137).
- Fixed the processing of OSP coordinate conversion COPY command. (202004004)
- Improved the contact coordinate value when there is no contact in OSP SKIP function.
- Fixed MULTIAXIS processing for inclined surface machining.

- Fixed the composite fixed cycle tolerance for arc shapes processing.
- Fixed the processing of nose R compensation.
- Fixed the C-axis processing of the TOSNUC PX100 fixture offset.
- Improved the view display of the horizontal slant
- Fixed the handling of inequivalence operator for SIEMENS.
- Fixed the handling in case that the end point of the composite fixed cycle is below the start point.
- Fixed the processing when reversing the mounting direction of the turning tool. (202003001)
- Improved the pick processing in the tool path filter.
- Improved the error check for tap hole processing. (202009001)
- Fixed the hole drilling report output display. (202008002)
- Fixed M code processing in CALLBYGM. (201612005)
- Fixed the processing of tool tip point control (when the rotation axis of the work coordinate system is specified). (202011003)
- Fixed the processing of OSP system variables (work coordinate system values).
- Fixed the processing of non-commanded axis operation in G53 block. (202010001)
- Improved the status display when canceling OSP cycle operation (G180). (202004002)
- Improved the work coordinate system preview display in the work jig setting dialog box.
- Fixed the project loading process at batch startup. (202011007)
- Fixed the processing of the initial value of the rotate axis in the tool tip point control IJK command.
- Fixed the collision detection process while turning the attachment. (202009007)
- Fixed the B-axis correction process in G43 compensation of turning tools. (202101002)
- Fixed VM collision processing when the distance is more than the tolerance.
- Added the batch command CUT_COMP for tool correction ON / OFF to HELP. (202009006)
- Corrected the description of the machine command in HELP. (202009004)
- Fixed the turning process near the Z axis.

## C) List of Support Reception Numbers

201612005、201912004、201912010、202001002、202001006、202002007、202003001、202003005、202003006、202004002、202004003、202004004、202006004、202008002、202009001、202009004、202009006、202009007、202010001、202011003、202011007、202101002

## D) Operating Environment

| | |
|---|---|
| OS | < NCVIEW / NCVIEW Neo 32-bit Version ><br>Windows 10 / Windows 10 64bit<br>Windows 8.1 / Windows 8.1 64bit<br>※Also operates on 32-bit if installed on a 64-bit OS<br><br>< NCVIEW Neo 64-bit Version/ NCVIEW MC3><br>Windows 10 64bit<br>Windows 8.1 64bit |
| Memory | 16 GB or higher (64bit) / 2 GB or higher (32bit) recommended |
| CPU | Intel Core i7 recommended |
| Graphics | NVIDIA recommended |