Multidisciplinary Integrated Analysis Solution for Design Optimization

midas NFX **2026 R2 Feature Overview**

midas NFX New Features 2026

Key Updates (Release Version : 2026.11.06)

Structural Analysis

- CAD Interface update (latest CAD import versions, part names of pattern parts in specific CAD programs) → p.3
- Added Newmark-beta time integration options → p.5
- Added Mohr-Coulomb material model → p.7
- Added frequency response-based fatigue analysis and improved fatigue analysis version compatibility → p.8
- Improved and expanded Nastran format export → p.10
- Adopted RLM-based license system → p.14

Computational Fluid Dynamics (CFD)

- Added vector display control for CFD results (Hide Below, Max. Vectors) → p.15 (How to use p.16)
- Added warning message for gas compression conditions in VOF analysis → p.17
- Improved streamline display range → p.18
- Improved detailed initial condition definition (restart file) → p.19 (How to use p.20)

Common Features

- Improved automatic report generation → p.21 (How to use p.22–24)

midas NFX provides a fully integrated simulation environment for structural, thermal, fluid, and optimization analysis using a single model within a unified workspace.
With its Windows-based GUI and MIDAS customer support ecosystem, the software enables engineers to perform simulation efficiently while providing a structured environment for technical training and support.

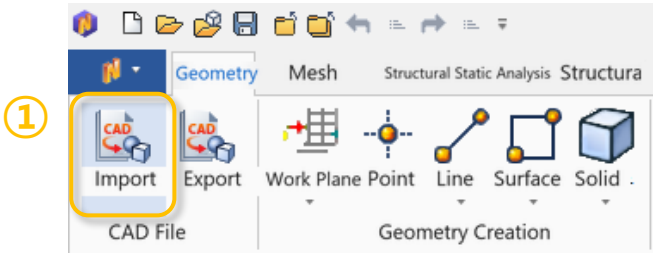
1. CAD Interface Update and Improvements

CAD Interface Version Update

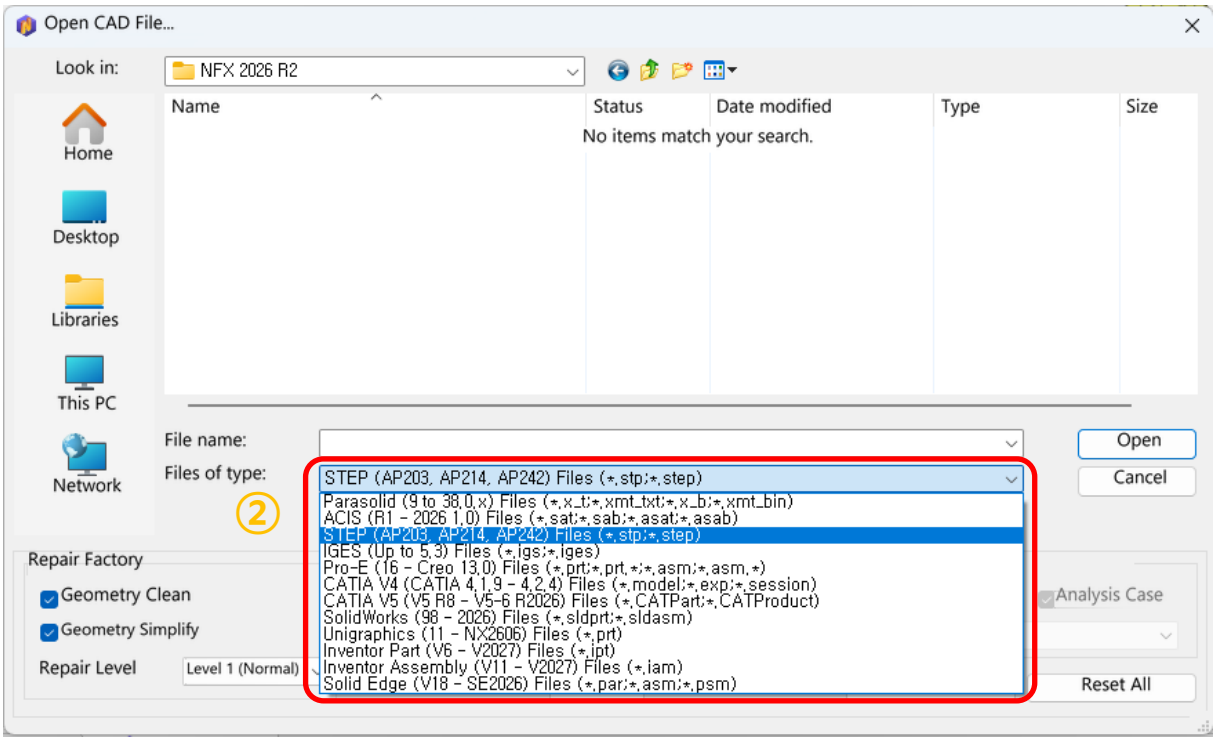
The IOP library has been updated to ensure compatibility with the latest CAD formats. With this update, part and assembly files in the updated formats can be imported, and geometry translation compatibility is maintained for previously supported CAD files.

Support for the latest CAD versions may be delayed depending on the update schedules of CAD vendors. If the latest version is not yet supported, we recommend converting the file to Parasolid format before importing. We will continue to work to support the latest CAD versions as quickly as possible.

Category	File Extension	Applicable Version
Parasolid	x_t, xmt_txt, x_b, xmt_bin	9.0 ~ 38.1.x
ACIS	sat, sab, asat, asab	R1 ~ 2026.1.0
STEP	stp, step	AP203, AP214, AP242
IGES	igs, iges	Up to 5.3
Pro-E / Creo	prt, prt.*, asm, asm.*	16 ~ Creo 13.0
SolidWorks	sldprt, sldasm, slddrw	98 ~ 2026
CATIA V4	model, exp, session	4.1.9 ~ 4.2.4
CATIA V5	CATPart, CATProduct	V5 R8 ~ V5-6R2026
Unigraphics(NX)	prt	11 ~ NX2606
Inventor Part	ipt	V6 ~ V2027
Inventor Assembly	iam	V11 ~ V2027
SolidEdge	par, asm, psm	V18 ~ SE2026



1. Geometry – Select (CAD) Import



2. Select the CAD file format to import

Part Names of Pattern Parts in Specific CAD Programs

When objects were created with the pattern feature in a specific CAD program (SolidWorks), the assigned part names were not correctly reflected in the NFX model tree.

1. Improvement

   When importing CAD files that use the pattern feature, the part names assigned in the original CAD are now correctly reflected in the model tree. This keeps object names consistent between the CAD model and the analysis model and makes it easier to identify target geometry when assigning materials and boundary conditions by part.

2. Expected Benefits

   • Consistent object names between CAD and analysis models

   • Easier identification when assigning materials and boundary conditions by part

   • No need for a separate conversion to Parasolid (*.x_t)

3. Usage Note

   • Place the assembly and part files exported from the CAD program in the same folder before use.

2. Added Newmark-beta Time Integration Options

Newmark-beta options have been added so that users can directly set the time integration conditions in transient response analysis. Constant acceleration and linear acceleration methods can be selected, and the Gamma and Beta coefficients can be entered directly through the user-defined option. In addition, the HHT-α option is provided to control the numerical damping of high-frequency responses that occur during time integration.

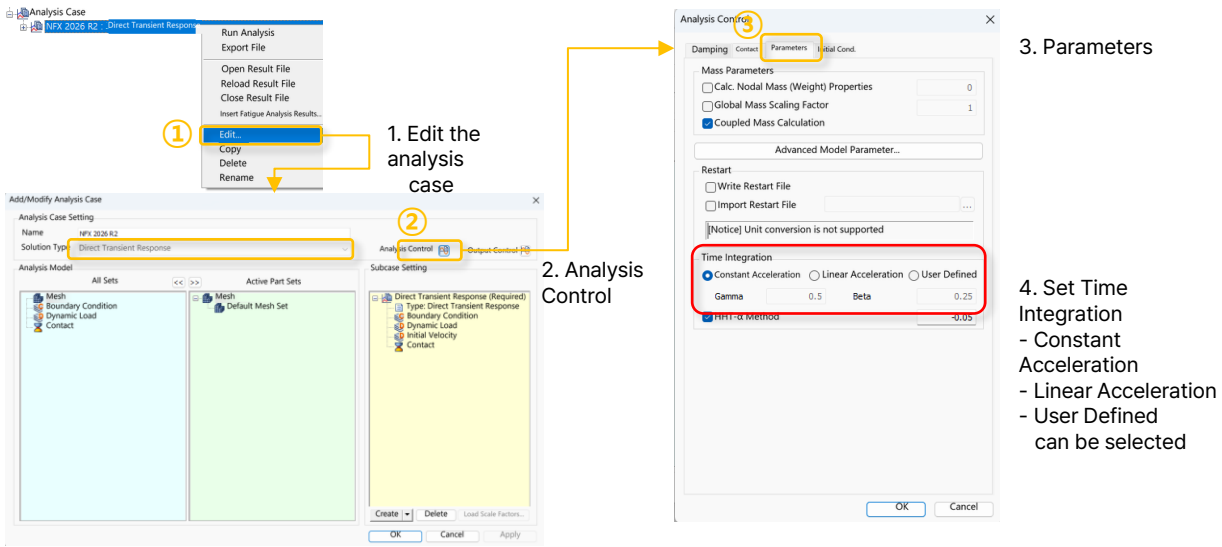
1. Technical Background and Necessity

   Transient response analysis uses time integration to calculate the dynamic behavior of a structure under time-varying loads. Since time integration conditions can affect the stability, accuracy, and numerical damping of high-frequency components, appropriate integration methods and coefficients need to be applied according to the analysis objective. Previously, NFX applied the solver defaults for the time integration coefficients and numerical damping, making it difficult for users to set these values according to the characteristics of the analysis model. In this version, Newmark-beta time integration options have been added to the analysis control of transient response analysis. The HHT-α option can also be used to control the numerical damping of unwanted high-frequency responses that may occur during the analysis.

2. Time Integration Method Setting Added

   A Time Integration Method setting has been added at the following location.

   • Analysis Control > Parameters > Time Integration



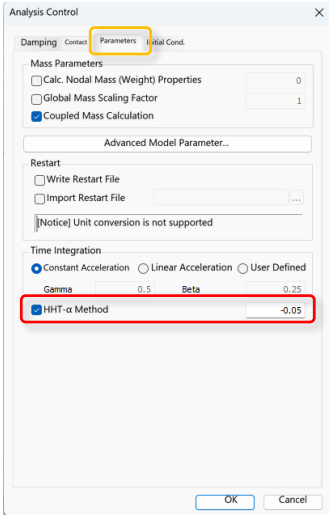
Users can select one of the following three Newmark time integration methods according to the analysis objective.

Option	Coefficients	Key Characteristics
Constant Acceleration	γ = 0.5, β = 0.25	Representative Newmark method with unconditionally stable characteristics. Provided as the default
Linear Acceleration	γ = 0.5, β = 1/6	Calculates the response considering a linear variation of acceleration within each time step
User Defined	Direct input of γ and β	Coefficients can be set to match the desired numerical stability and accuracy

3. HHT-α Option Added

The HHT-α (Hilber-Hughes-Taylor) numerical damping option is provided together with Newmark-beta time integration. Applying HHT-α effectively damps unwanted high-frequency numerical oscillations during the analysis while minimizing the effect on the main structural response.

- Option to enable and disable HHT-α
- User-defined Alpha (α) input
- Input value validation added
- HHT-α settings saved to and reloaded from .mec
- Default value identical to the previous solver default (Alpha = -0.05)
- Existing models keep the same time integration conditions when opened, without any setting changes



Analysis Control > Parameters > HHT-α Method
- Enabled: HHT-α value can be entered directly
- Disabled: pure Newmark-beta (HHT-α = 0)

4. Expected Benefits

- Improved analysis stability through reduced high-frequency numerical oscillation
- Applicability to a wide range of dynamic analysis conditions
- Easier saving and reuse of time integration settings
- Result continuity with existing models

5. Applicable Analysis Types

The Newmark-beta time integration options are provided for the following analysis types.

- Linear Direct Transient Analysis
- Nonlinear Implicit Transient Analysis
- Sequential Nonlinear Analysis
- 2-way FSI Analysis

3. Added Mohr-Coulomb Material Model

A Mohr-Coulomb material model has been added to simulate the nonlinear behavior of materials whose shear strength depends on pressure. By defining cohesion, internal friction angle, and dilation angle, users can analyze the yield and plastic behavior of pressure-dependent materials such as concrete and soil.

1. Background and Necessity

   Materials such as concrete and soil exhibit increasing shear strength as compressive confinement increases. Therefore, conventional metal plasticity models based on a constant yield stress cannot adequately represent the pressure-dependent yield behavior of these materials. The Mohr-Coulomb model defines shear strength through cohesion and internal friction angle, taking into account strength that varies with confining pressure. The dilation angle can also be used to define the volume change caused by plastic shear deformation after yielding.

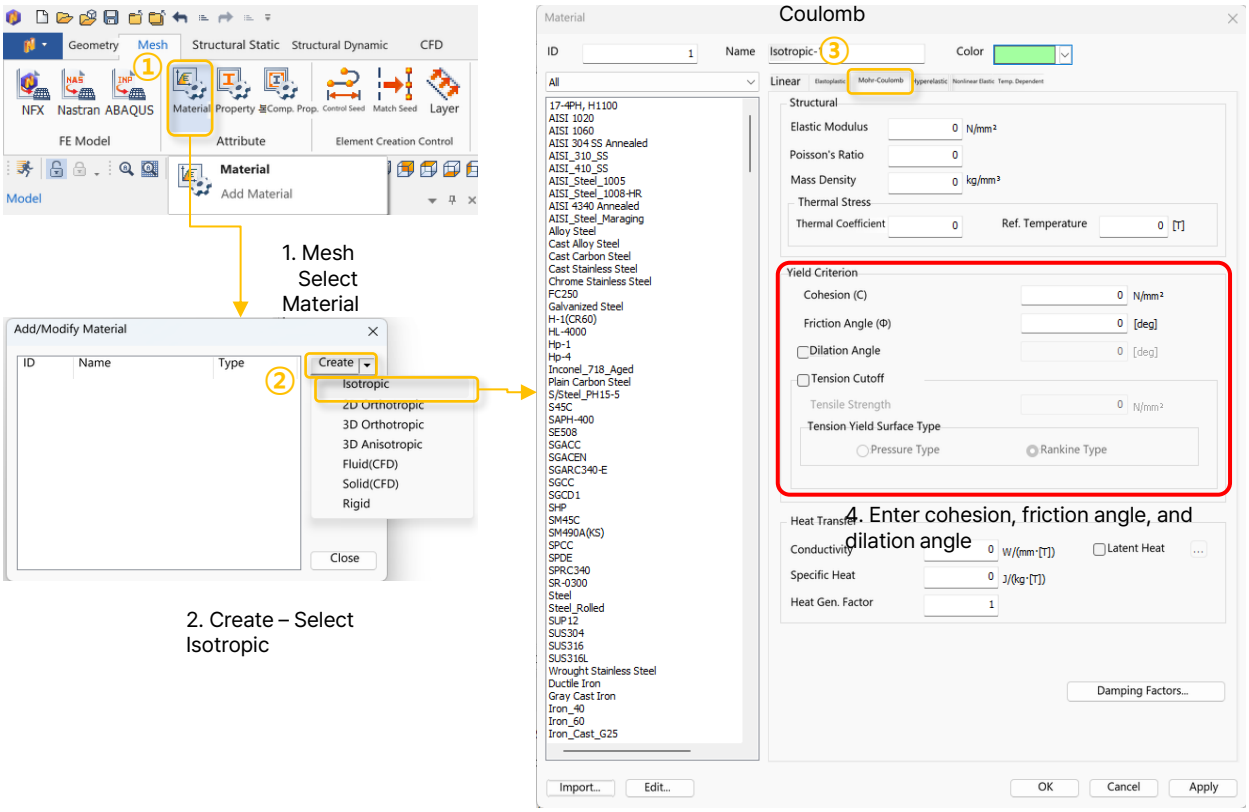
2. New Capability

   In 2026 R2, the Mohr-Coulomb model has been added to the nonlinear material models. Together with the existing elastic properties, users can enter cohesion, internal friction angle, and dilation angle to define the pressure-dependent yield and plastic behavior of the material.

   Cohesion represents the basic shear strength the material can resist with no normal stress, and the internal friction angle defines how much the shear strength increases as compressive confinement increases. The dilation angle determines the volume change that occurs as the material undergoes plastic shear deformation.

   This feature can be used to evaluate the nonlinear behavior of concrete structures under compressive confinement, as well as for nonlinear analysis of materials whose shear strength is affected by confining pressure, such as soil, rock, and granular materials. This allows shear failure zones and plastic deformation to be evaluated according to pressure conditions.

3. Location



4. Frequency Response-Based Fatigue and Version Compatibility

Frequency Response-Based Fatigue Analysis Added

A capability has been added to evaluate the fatigue life and damage of structures using the stress results at each frequency calculated by frequency response analysis. When the duration of each frequency component is entered, the number of stress cycles is calculated automatically and applied to the fatigue analysis. The stress-life and strain-life methods are supported, and various mean stress correction methods can be applied according to material and load characteristics.

1. Technical Background and Necessity

   Rotating machinery and vibrating structures are subjected to cyclic loads at various frequencies during operation. In particular, when increased stress near a resonance frequency is repeated over a long period, fatigue failure can occur even if static strength criteria are satisfied. Previously, stresses at each frequency could be reviewed through frequency response analysis, but it was difficult to directly evaluate the fatigue life and damage when those stresses were repeated over a period of time. Frequency response results therefore had to be organized separately and the number of cycles calculated before applying them to the fatigue evaluation. In 2026 R2, stresses calculated by frequency response analysis are linked directly to fatigue analysis, enabling fatigue evaluation that considers the stress magnitude and duration at each frequency.

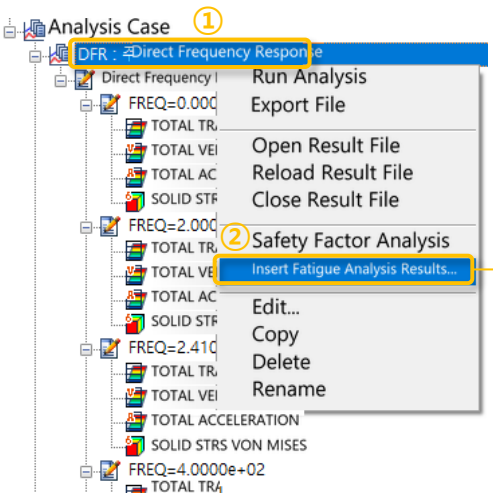
2. Key Updates

   When the preceding frequency response analysis result and the duration are set in a fatigue analysis case, the number of cycles at each frequency is calculated as the product of the frequency and the duration. For example, if a stress response at 500 Hz lasts for 400 seconds, 200,000 cycles are applied.

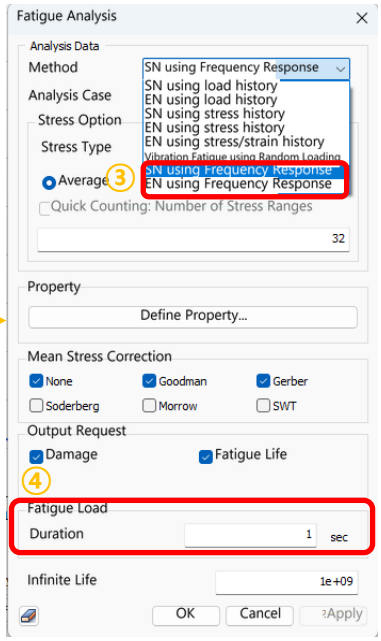
   The stress calculated at each frequency is processed as a full cycle, and fatigue life and damage are evaluated using the calculated number of cycles and the fatigue properties of the material. Frequency response-based fatigue analysis supports the stress-life (SN) and strain-life (EN) methods.

   The Goodman, Gerber, and Soderberg mean stress corrections are available for the stress-life method, and the Morrow and SWT methods are supported for the strain-life method. This enables a continuous workflow from frequency response analysis to fatigue evaluation for rotating machinery, support structures, and mechanical components subjected to repeated vibration.

3. Location

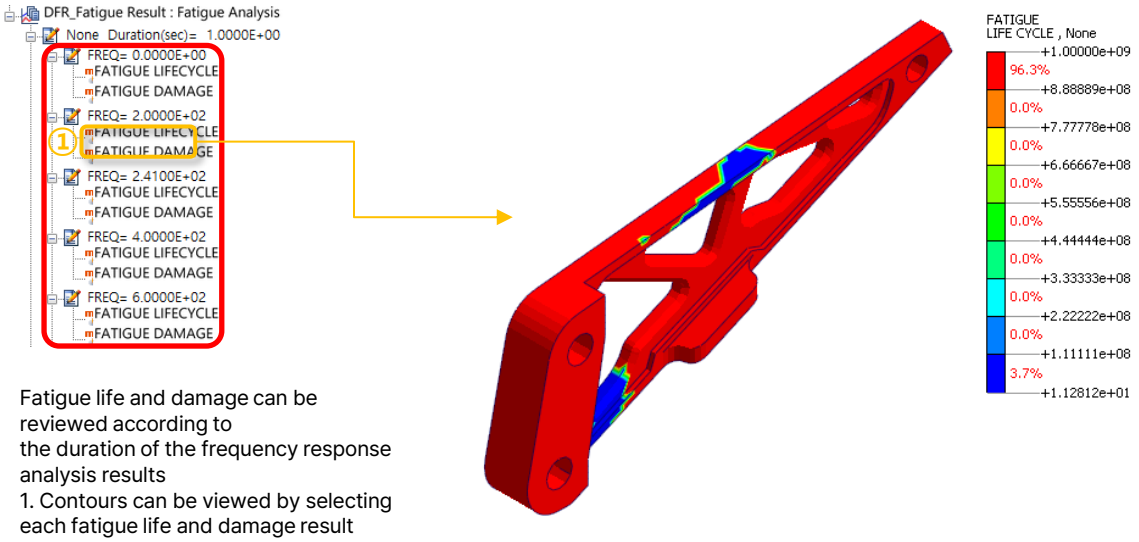


1. Right-click a completed frequency response analysis (direct or modal) in the Analysis & Results window
2. Select Add Fatigue Analysis



3. Analysis Method - Select SN & EN method using frequency response

4. Enter the fatigue load duration



Fatigue life and damage can be reviewed according to the duration of the frequency response analysis results

1. Contours can be viewed by selecting each fatigue life and damage result

Frequency response-based fatigue results represent the condition in which each frequency is sustained for the entered duration.

For example, the fatigue life result at 200 Hz for a duration of 10 sec corresponds to 10 (sec) * 200 (Hz = cycle/sec) = 2,000 cycles.

Improved Fatigue Analysis Version Compatibility

Compatibility of fatigue analysis models between versions has been improved. While applying the load cycle data structure extended in 2026 R2, data handling has been improved so that existing R1 models can be used reliably, minimizing inconsistencies in load cycle information that may occur when changing versions.

1. Key Improvements
   - Compatibility between R1 iCycle and R2 dCycle
   - Existing load cycle information automatically applied when R1 models are opened in R2
   - Load cycle information retained even when an R2 model is modified and re-saved in R1
   - Default load cycle values automatically applied when creating new fatigue and frequency response-based fatigue analyses
   - Input and display of real-valued load cycles (dCycle)
   - Improved model loading stability through enhanced load cycle validation

5. Improved and Expanded Nastran Format Export

Improved MSC Nastran Format Export of Analysis Input Models (Preprocessing)

NFX supports exporting analysis input models to MSC Nastran Bulk Data Format (.bdf). To improve integration with MSC Nastran-based analysis environments, the Nastran format export has been improved. The range of supported analysis data conversion has been improved and expanded to increase the usability of NFX preprocessing models.

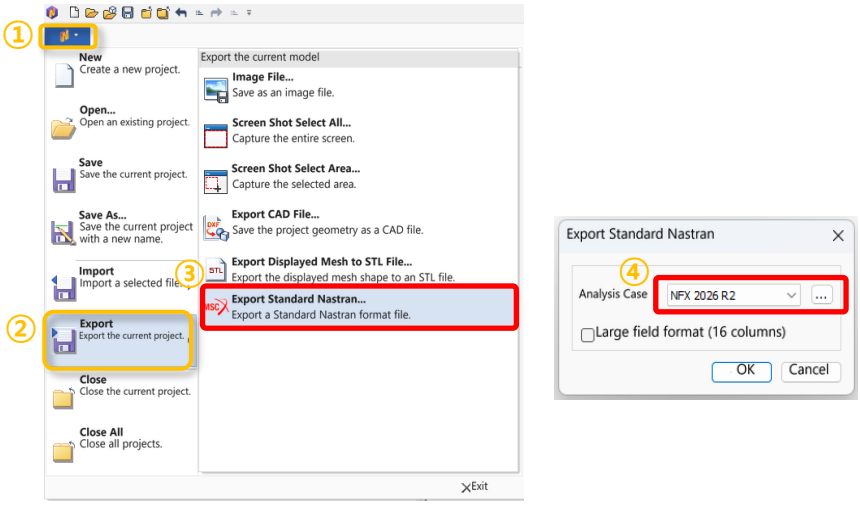
1. Background

   In some industries, modeling and preprocessing are performed in NFX and the analysis is then run in a dedicated MSC Nastran-based environment. The previous Nastran export supported basic analysis information, but conversion of nonlinear properties, contact conditions, and certain analysis cases was limited. The Nastran format export has therefore been improved, and a log of any information omitted during export can now be reviewed for user convenience.

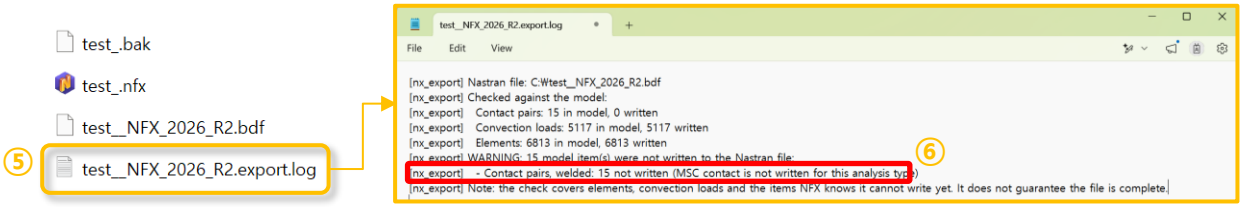
2. Key New Capabilities

   - Improved export of property cards and case information for nonlinear analysis
   - Improved export of contact condition information
   - Expanded Nastran analysis case support (thermal stress analysis and response spectrum analysis added)
   - Log output for unsupported and omitted information during Nastran format export
   - Fixed composite property conversion
   - Improved conversion accuracy and resolved data loss for loads, constraints, time functions, and case settings

3. Location



1. Click the menu at the top left of the NFX interface
2. Click Export
3. Select Export to Standard Nastran File
4. Select the analysis case and click OK



5. Open the .export.log file (in the .nfx folder)
6. Check information not exported to Nastran

4. Scope of Application

Category	Supported Items
Supported	Linear Static, Modal Analysis, Linear Buckling, Prestress Analysis, Direct and Modal Frequency Response Analysis, Response Spectrum Analysis, Nonlinear Static Analysis, Steady-State Heat Transfer Analysis, Transient Heat Transfer Analysis, Nonlinear Heat Transfer Analysis, Transient Response Analysis, Thermal Stress Analysis
Model	Grids and Coordinate Systems (GRID, CORD1x, CORD2x), Spring and Damper Elements (CELAS1, CELAS2, CDAMP1, CDAMP2), Mass Elements (CMASS1, CMASS2, CONM2), Bush and Gap Elements (CBUSH, CGAP), Rod/Bar/Beam Elements (CROD, CONROD, CTUBE, CBAR, CBEAM), Shell Elements (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR), Solid Elements (CTETRA, CPYRAM, CPENTA, CHEXA), Rigid and MPC Elements (RBAR, RBE2, RBE3, MPC, MPCADD)
Materials	MAT1, MAT2, MAT4, MAT5, MAT8, MAT9, MATS1, MATHP, MATHE, MATEP, MATORT, MATT1, MATT2, MATT4, MATT8, MATT9, MATTORT
Properties	PBAR, PBARL, PBEAM, PBEAML, PROD, PTUBE, PSHELL, PSOLID, PCOMP, PCOMPG, PCOMPLS, PBUSH, PBUSHT, PELAS, PDAMP, PMASS, PGAP, PWELD
Loads and Boundary Conditions	FORCE, FORCE1, FORCE2, MOMENT, MOMENT1, MOMENT2, GRAV, RFORCE, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, LOAD, DLOAD, DAREA, TLOAD1, TLOAD2(Partially supported), RLOAD1, RLOAD2, RANDPS, TIC, TSTEP, TSTEPNL, SPC, SPC1, SPCADD, SPCD, SUPORT(Response Spectrum), SUPORT1
Contact and Special Loading	MSC Contact, Follower Pressure, Rotational Force, BSURF, BCSURF, BCONECT, BCONPRG, BCONPRP, BCPARA, BCTABL1
Functions and Analysis Controls	TABLED1, TABLED2, TABLED3, TABLEM1, TABLEM2, TABLEM3, TABLES1, TABDMP1, TABRND1, EIGRL, FREQ, FREQ1, FREQ2, FREQ3, NLPARM, NLPCI, NLSTEP, STEP, SUBSTEP, ANALYSIS, STATSUB(PRELOAD), BCONTACT
Composite	Composite Shell (PCOMP, PCOMPG), Composite Layered Solid (PCOMPLS)
Other Export Information	PARAM, LSEQ, DTI, Unit System Information, Large-Field Format (16-column), Unsupported Item Report

Some cards may be converted to equivalent representations based on NFX's internal data structure.

Added Export of Analysis Results in Nastran Output File Format

A capability has been added to save the results of analyses performed in NFX in Output2 (.op2), the MSC Nastran result file format. This allows NFX analysis results to be used with external post-processing tools and improves data compatibility with Nastran-based analysis environments.

1. Background and Necessity

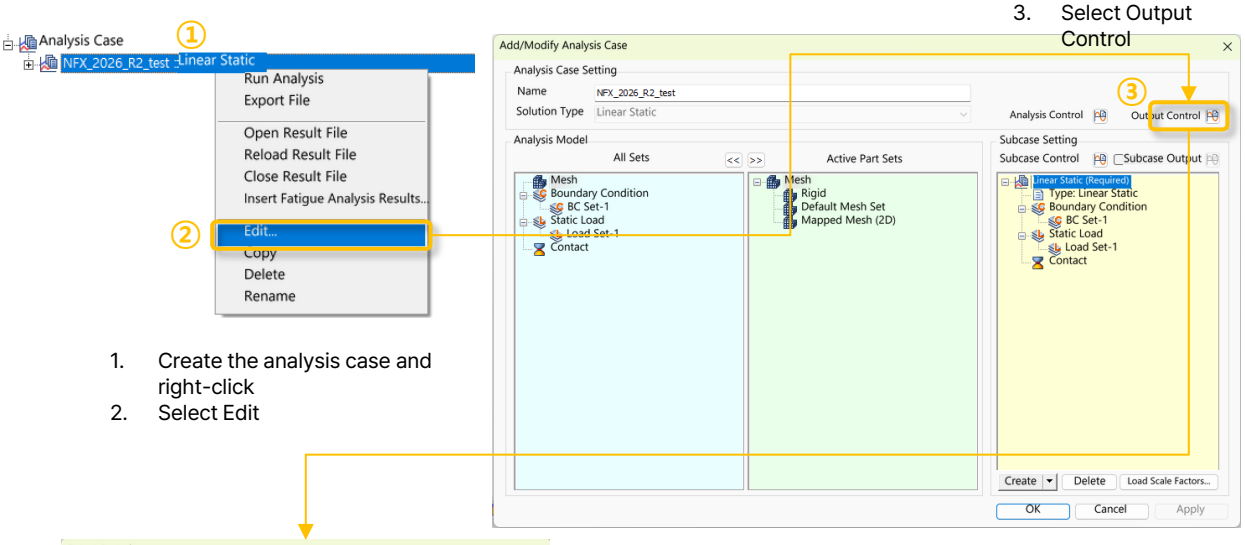
   Analysis result data is used for verification and analysis with various post-processing tools, and the OP2 format in particular is a standard result file format widely used in MSC Nastran-based analysis environments. Previously, a separate result conversion process or a dedicated post-processing environment was required to use NFX results in external post-processing tools.

2. Key Developments

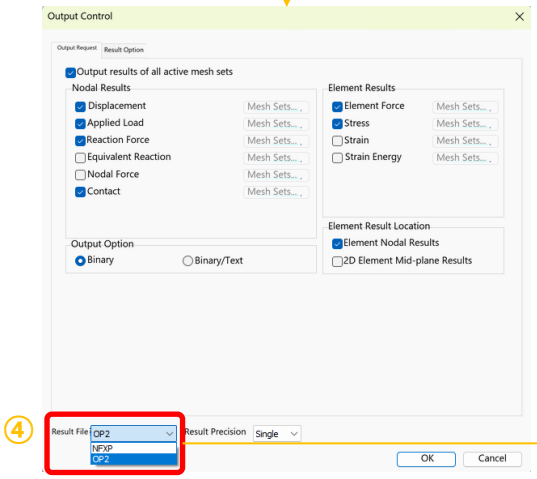
   After the analysis is complete, result data can be saved in OP2, the MSC Nastran binary result file format.

   - OP2 output of results from analyses performed in NFX
   - Includes key analysis results such as displacement, stress, applied force, and reaction force
   - Result data can be linked with MSC Nastran-based post-processing environments

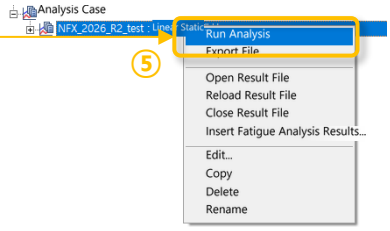
3. Location



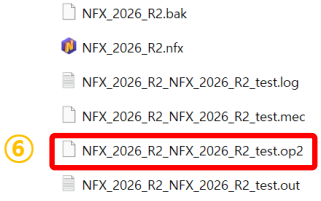
1. Create the analysis case and right-click
2. Select Edit
3. Select Output Control



4. Output Control > Result File: select OP2



5. Output Control > Result File: select OP2



5. Check the op2 file in the .nfx file folder

※ The op2 file is created as [*NFX file name_analysis case name*.op2].

4. Scope of Application

Category	Supported Items
Analysis Types	Linear static, Steady-state heat transfer, Normal modes, Linear buckling, Nonlinear static (including contact), Linear, nonlinear and modal transient, Explicit dynamics, Transient heat transfer, Response spectrum
Model	Grids and coordinate systems, CROD, CBAR, CBEND, CELAS2, CBUSH, CTRIA3, CQUAD4, CTRIA6, CQUAD8, CTRIAX6, CQUADX, CTETRA, CPENTA, CHEXA (linear and quadratic, Pyramid), CONM2, SPC, SPC1, MPC, RBE2, RBE3
Nodal Results	Displacements, Velocities, Accelerations, Mode shapes and buckling shapes, Eigen values, Reaction forces (SPC forces), MPC forces, Applied loads, Temperatures, Applied and reaction heat flows
Element Results	Stresses, Strains, Element forces, Composite ply stresses and strains, Composite failure indices, Strain energy, Nonlinear stresses and strains for shells, rods and solids, Heat flux and temperature gradients, Boundary heat flux
Response Spectrum	Combined displacements, reaction forces, stresses, strains and element forces

Some result components may be converted to equivalent representations based on NFX's internal data structure.

6. Adopted RLM-Based License System

The Flex LM-based license system previously used for product license authentication and management has been replaced with an RLM (Reprise License Manager)-based system. With this change, the same license management scheme is applied to NFX and related executable modules, and product authentication is supported in both Windows and Linux environments. From 2026 R2, users will use RLM-based license issuance and authentication procedures.

1. Background

   When NFX starts, it checks the available products and modules through the license server and authenticates the validity of the license. Previously, a Flex LM-based license system was used for this process. As products and execution environments diversified, a consistent license authentication scheme was needed not only for NFX but also for related products and standalone solvers. It was also necessary to integrate license issuance, server management, and product authentication to improve maintenance efficiency.

2. Key Changes

   In 2026 R2, the Flex LM-based license system has been replaced with an RLM-based system, and the new license authentication scheme can be used in NFX and related execution environments.

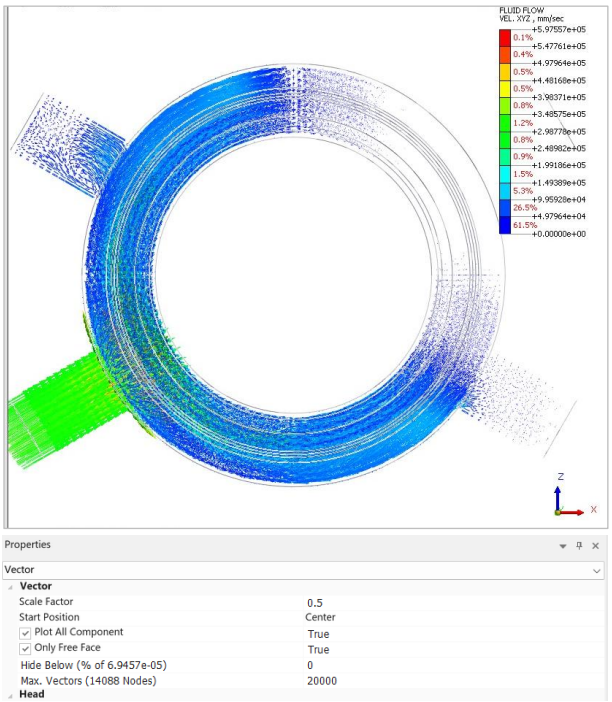
   - License authentication for NFX product execution and related products and executable modules
   - License check for MecSolver running on Linux
   - RLM Publisher environment for license issuance

   The change of license system does not affect how structural analysis models are created or the analysis capabilities of the solver.

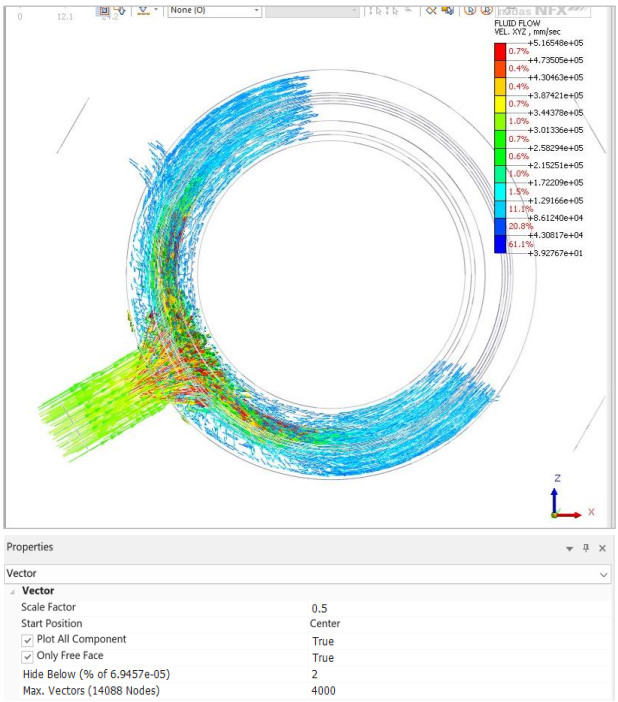
7. Added Vector Display Control for CFD Results

To solve the problem of too many vectors making flow structures difficult to read in CFD post-processing, two display options that control the vectors drawn on screen have been added. These are display-only features and do not affect the analysis results.

1. Background
   - When the cut plane mesh was dense, too many vectors were displayed, making it difficult to identify flow direction, recirculation, backflow, and vortex structures.
   - The existing Show Uniform function took a long time to process or did not display results correctly in some models.
2. Key Features
   - **Hide Below (% of Max)**: Hides small vectors below the entered percentage (%) of the largest vector magnitude. The reference maximum is the same as the legend maximum and is recalculated automatically when the result or step changes. (0 = nothing hidden)
   - **Max. Vectors**: Specifies the maximum number of vectors drawn on screen. Vectors are selected evenly across the entire model so that they are not concentrated in one area. (0 = all displayed)
   - **Order of Application**: Hide Below first determines the display targets, and Max. Vectors then limits the number among the remaining targets. Raising Hide Below concentrates the same number of vectors in high-velocity regions such as jets and wakes.
3. Location
   - Whole-model vectors: Post-processing Properties window > Vector group / Section vectors: Clip & Slice Plot dialog > Vector group



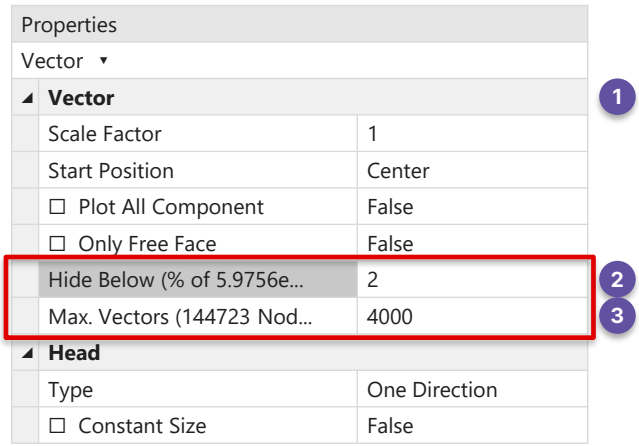
Previous version (all vectors displayed)



2026 R2 (Hide Below 2 %, Max. Vectors 4,000)

7-1. How to Use Vector Display Control

Both options are available for whole-model vectors and section vectors and are applied on screen immediately. Since result values are not changed, different values can be compared when creating figures for reports.



Properties window > Vector group (reconstructed from the R2 release build)

1. Setting Procedure

① **Location** Whole model: Post-processing Properties window > Vector group / Section: Clip & Slice Plot dialog > Vector group

② **Hide Below (% of Max)** Enter the hiding threshold (%). The current maximum is shown next to the option name.

③ **Max. Vectors** Enter the maximum number of vectors to draw. The total number of nodes (elements) is shown next to the option name.

④ **Check** The reference maximum is recalculated automatically when the result or step changes.

2. Input Values and Behavior

Option	When 0 is entered	Example	Behavior
Hide Below (% of Max)	Nothing hidden	5	Vectors smaller than 5% of the maximum are not drawn
Max. Vectors	All displayed	500	Even with 50,000 targets, only about 500 are drawn, selected evenly across the model

3. Usage by Purpose

Situation	Option to adjust	Effect
Vectors overlap and flow direction cannot be seen	Reduce Max. Vectors	Fewer vectors make direction, recirculation, and vortex structures easier to read
Main flow is buried among low-velocity vectors	Raise Hide Below	The same number of vectors is concentrated in high-velocity regions such as jets and wakes
Creating figures for reports	Use both options	e.g. Hide Below 2 %, Max. Vectors 4,000 (comparison figure on the previous page)

4. Note

- The order of application is Hide Below → Max. Vectors. Raising Hide Below does not reduce the number of vectors; it only moves the display locations to high-velocity regions.
- The arrow head and body are always processed together, so no truncated arrows occur, and the function works independently of the existing Show Uniform function.

8. Added Gas Compression Warning for VOF Analysis

When simulating the compression of gas in a closed vessel with VOF (Volume of Fluid) and pressure-based compressible analysis, a warning message is now displayed at the analysis file generation stage if the mesh material settings prevent the gas from being compressed.

1. Technical Background

   In VOF analysis, the fluid model assigned to the mesh (Material) determines the compressibility of the entire analysis domain. Even if the phase material is set to an ideal gas, the analysis domain does not become compressible.

   - If the mesh is created with a liquid (incompressible) material, gas in a closed space is not compressed even when a gas phase exists.
   - In this case, the interface between the two phases may form non-physically or the liquid may not flow in properly, which users could easily mistake for an analysis error.

2. Improvement

   A warning message is displayed in the Output window when all of the following conditions are met. Only a warning is shown, and the analysis continues.

   - VOF (free surface) multiphase flow analysis is used
   - A phase material is an Ideal Gas or an incompressible ideal gas
   - The fluid material of the analysis mesh is not an ideal gas (incompressible, incompressible ideal gas, weakly compressible, and barotropic)

   Species transport (mixture) analysis is not checked because the mixture model determines the fluid model.

```
> [Warning] Gas phase is detected in a VOF simulation, but the domain (mesh) material is not compressible (Ideal Gas).
> [Warning] In this configuration, trapped gas cannot be compressed, which may lead to non-physical results in closed-volume
  filling simulations. Set the fluid model of the mesh material to Ideal Gas.
```

Warning message in the Output window

3. How to Use (Gas Compression in a Closed Space)

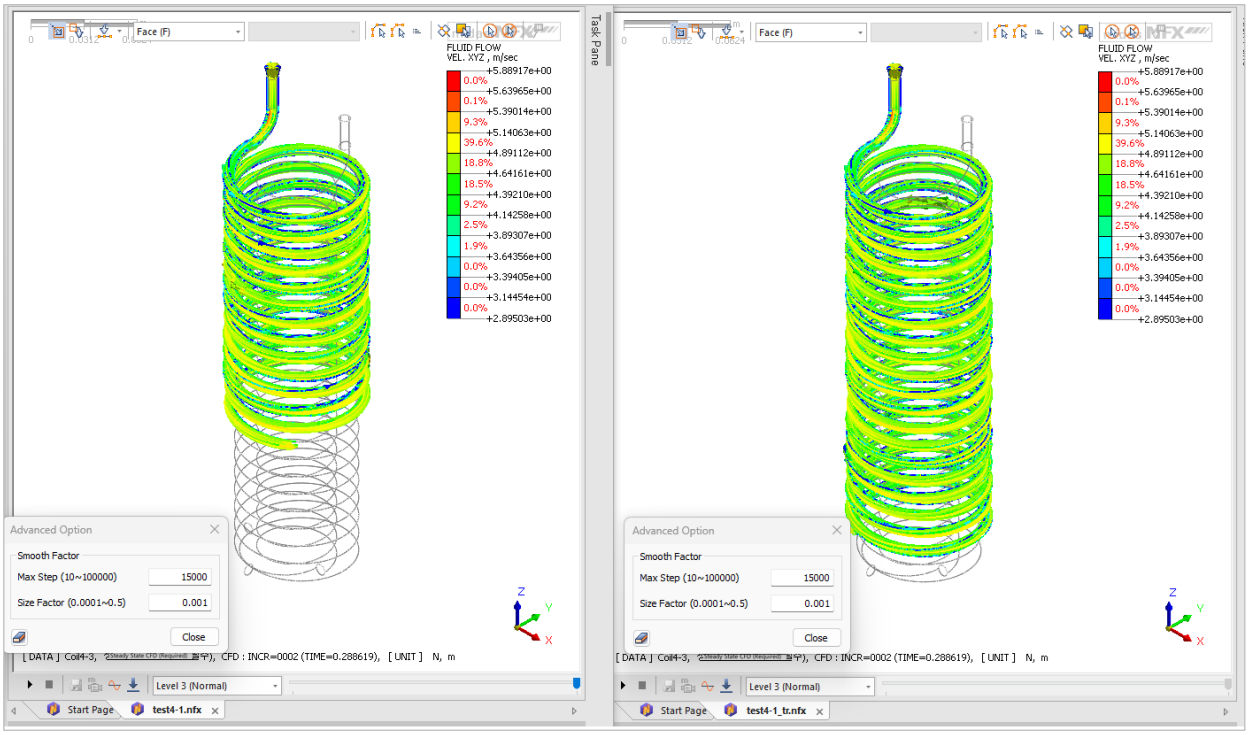
Step	Setting
① Material	Define the fluid model of the gas phase material and of the material assigned to the analysis mesh as Ideal Gas.
② Mesh	Assign the ideal gas material from ① to the analysis mesh. (The mesh material determines the compressibility of the entire domain)
③ Phase	Assign the materials of the gas phase and the liquid phase in the VOF multiphase flow.
④ Check	Check the Output window after generating the analysis file. If a warning appears, review the mesh material setting in ②.

4. Behavior by Mesh Material

Fluid model of mesh material	Gas compression in closed space	Warning message
Ideal Gas	Simulated	Not displayed
Incompressible / Incompressible ideal gas / Weakly compressible / Barotropic	Not simulated	Displayed in the Output window (analysis continues)

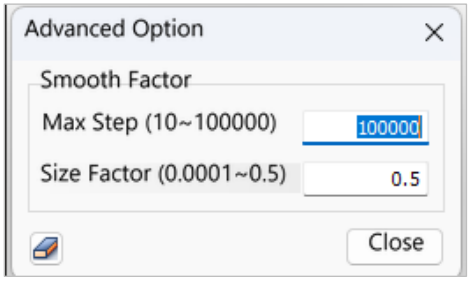
9. Improved Streamline Display Range

1. Improved Streamline Display Range
   - Issue: The streamline length was internally limited to "model size (bounding box diagonal) × 10", so streamlines were cut off midway in models with long flow paths such as coils. This limit could not be exceeded even by increasing the maximum step and step size in the advanced options.
   - Improvement: Streamlines now travel up to "maximum step × step size". The defaults (3000 / 0.001) are unchanged, and for models with long flow paths, streamlines can be displayed up to the outlet by increasing the maximum step in the advanced options.



Previous version (streamline cut off in the middle of the coil) 2026 R2 (streamline displayed up to the outlet)

2. How to Use (Streamlines in Models with Long Flow Paths)
   - **Location** Advanced options of the streamline settings
   - **Input range** Max step 10 ~ 100,000, step size 0.0001 ~ 0.5 (default 3,000 / 0.001)
   - **Streamline length** Travels up to max step × step size.
   - If streamlines are cut off before the outlet, increase the max step first. The example coil model is displayed up to the outlet at about 20,000 or more.



Streamline advanced options dialog

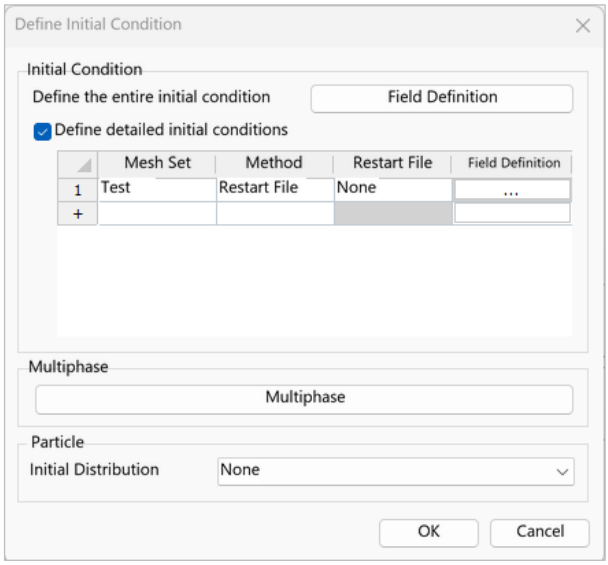
10. Improved Detailed Initial Condition Definition (Restart File)

In the "Define detailed initial conditions > Restart File" function, which uses previous analysis results as initial values for a new analysis, the problem of the analysis stopping for models with modified boundary conditions has been resolved.

1. Issue

   In a model where some wall conditions were removed and meshes were added after a flow analysis, running a heat transfer analysis with the previous flow result as the restart file output "ERROR [1041] : WRONG RESTART FILE FORMAT" and the analysis stopped.

2. Cause and Improvement
   - Cause: The solver read the inlet, outlet, and wall boundary information stored in the restart file by boundary set number, so adding or deleting boundary conditions shifted the numbers and the boundaries of the previous analysis could not be found.
   - Improvement: When used for initial values, mismatched boundary information is now ignored and the analysis proceeds.
   - Additional improvement: Nodes in the specified mesh set that do not exist in the previous result, which were previously initialized to 0, now use the global initial condition values.
3. How the Function Works (Reference)
   - Only result values (velocity, pressure, turbulence, temperature, etc.) are taken from the restart file, matched by node number, and applied only to nodes of the specified mesh set.
   - Boundary conditions are taken from the current analysis case, not from the restart file.
   - Results not in the restart file (e.g. temperature of a flow result without heat transfer) use the values from Define the entire initial condition.
   - Since a new analysis is started, time restarts from 0 seconds. To continue from the previous analysis time, use "Restart" in Analysis Control.



Analysis Control > Initial Condition > Define detailed initial conditions (Restart File)

10-1. How to Use Detailed Initial Condition Definition (Restart File)

This function uses previous analysis results as initial values for a new analysis. For example, a heat transfer analysis with modified boundary conditions or meshes can be started using flow analysis results as initial values.

1. Setting Procedure

Step	Setting
① Previous analysis	Complete the analysis to be used as initial values (e.g. flow analysis) to prepare the restart file.
② New analysis case	Set up the analysis case with new conditions, such as modified boundary conditions or added meshes.
③ Detailed initial condition	In Analysis Control > Initial Condition > Define detailed initial conditions, select "Restart File" as the method, and specify the restart file and the mesh sets to which the values are applied.
④ Global initial condition	Values not in the restart file are taken from Analysis Control > Initial Condition > Define the entire initial condition > Field Definition. e.g. For a heat transfer analysis from flow results, initial temperature = "Fluid Temperature" input value
⑤ Run analysis	The analysis time starts from 0 seconds.

2. Items Taken from Previous Results

Category	Values taken	How applied
Result values	Velocity, pressure, turbulence, temperature, etc.	Matched by node number and applied only to nodes of the specified mesh sets
Boundary conditions	Not taken	Boundary conditions of the current analysis case are used
Nodes/items without results	—	Global initial condition values used (from 2026 R2; previously 0)

3. Function Selection Guide

Situation	Function to use	Analysis time
Start an analysis with new conditions using previous results as initial values	Define detailed initial conditions > Restart File	Restarts from 0 seconds
Continue the previous analysis under the same conditions	Analysis Control > Restart	Continues from the previous analysis time

11. Improved Automatic Report Generation

The Report function, which automatically creates a Word analysis report from analysis results, can be used again. It applies to both structural and fluid/thermal analysis results.

1. Issue

   Running Results > Special Post > Report caused the program to terminate abnormally.

2. Improvement

   The cause of the abnormal termination has been fixed and the Report function has been re-enabled.

   Report generation has been verified with structural (linear static) and heat transfer (steady-state) examples.

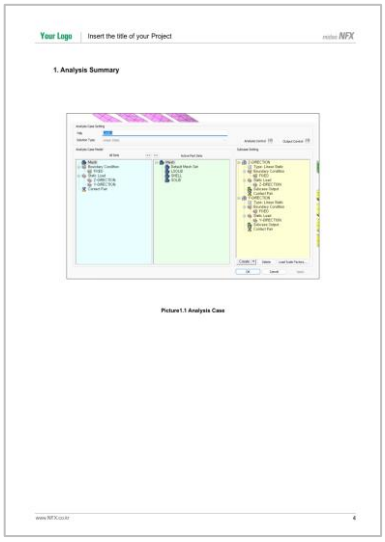
3. Contents of the Generated Report

   Cover, table of contents

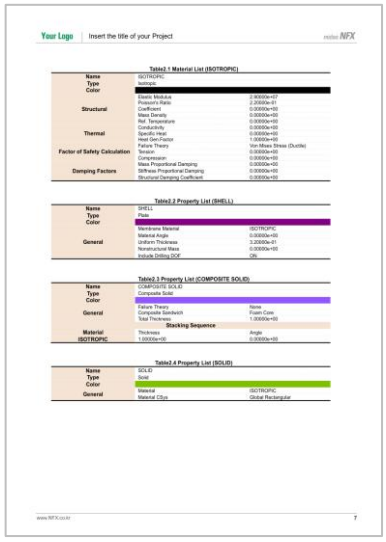
   1. Analysis Summary: analysis case settings

   2. Model: mesh shape, material/property colors, material/property list tables, boundary condition/load label figures

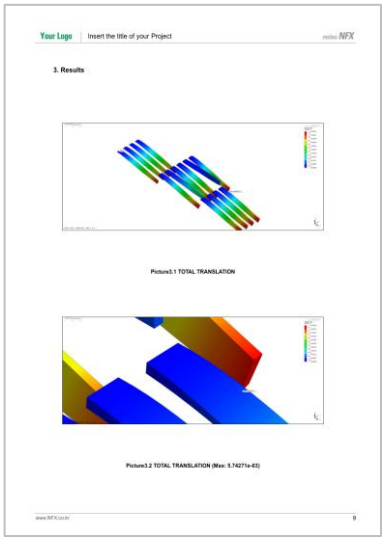
   3. Results: result contour figures and max/min result tables (e.g. displacement, von-Mises stress, and safety factor for structural analysis / temperature for heat transfer analysis)



1. Analysis Summary



2. Model (material/property lists)



3. Results (displacement contour)

Example of an automatically generated report (linear static example, CASE1_Report.doc)

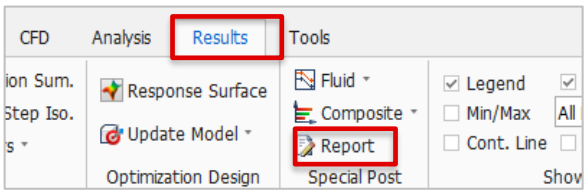
How to Use (details: p.22–24)

Run Results > Special Post > Report on a model with completed analysis to create the report automatically from the analysis results. The default file name is "AnalysisCaseName_Report.doc".

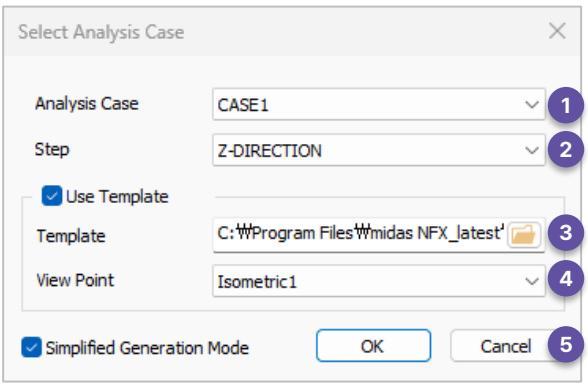
The cover title, subtitle, date, and project name in the header are created as placeholders and are edited directly in Word. The result items included in the report are predefined for each analysis type.

11-1. Report Function: Launch and Analysis Case Selection

Run the function on a model with completed analysis. A Word report (.doc) is created in the order Select Analysis Case → edit the layout in the Report Generation window → Generate.



Results > Special Post > Report



Select Analysis Case dialog

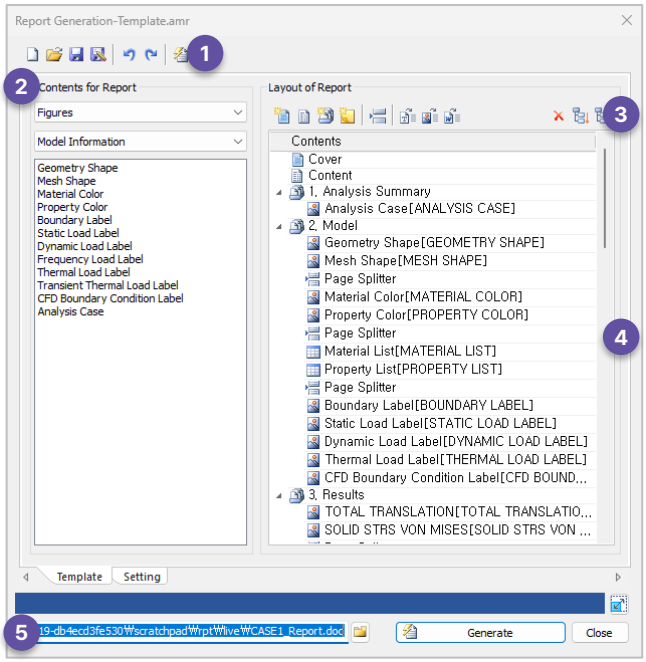
1. Select Analysis Case Dialog Items

Item	Description
① Analysis Case	Select the analysis case for the report.
② Step	Select the step (subcase) from which results are taken. e.g. subcases by load condition: Z-DIRECTION, Y-DIRECTION
③ Use Template > Template	Specify the template file (*.amr) that defines the report contents and items. Default: installation folder\Language\English\Report Template\Template.amr
④ View Point	View point for capturing report figures: Isometric1, Isometric2, Front, Back, Top, Bottom, Left, Right, Current
⑤ Simplified Generation Mode	When checked (default), the report is generated directly with the template layout. When unchecked, the Report Generation window opens so the layout can be edited before generating.

2. Note
   - MS Word and MS Excel are required to generate reports. Close any open Word documents before generating.
   - If a case without a result file is selected, a message indicating that no result file exists for the selected case is displayed.
   - The default template contains items for structural, thermal, and fluid analyses; items without results in the current analysis are omitted from the report.

11-2. Report Function: Report Generation Window (Template Tab)

This window opens when Simplified Generation Mode is unchecked and OK is clicked. Choose items in [Contents for Report] on the left, and set the order and structure of the report in the tree of [Layout of Report] on the right.



Report Generation window – Template tab

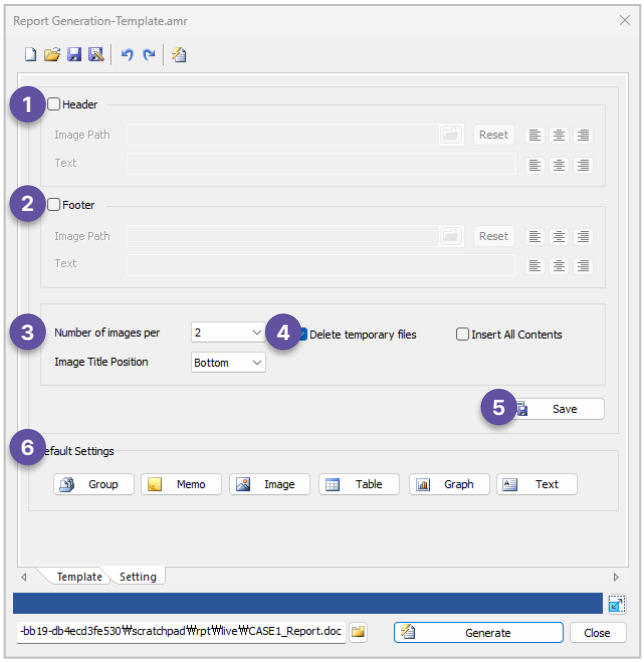
1. Window Layout
   - ① Toolbar: New, Open, Save, Save As (template *.amr), Undo, Redo, Generate
   - ② Contents for Report: select the type (Figures, Tables, Graphs) and result category to list the items that can be added.
   - ③ Layout toolbar: insert Cover, Content, Group, Memo / Page Splitter / insert Text, Image, Word file / Delete, Up, Down
   - ④ Contents tree: cover, contents, chapters (groups), and the figure/table items under them. Items without results in the current analysis are grayed out.
   - ⑤ Output file path (default: model folder\AnalysisCaseName_Report.doc) and [Generate] button

2. Items by Content Category

Category	Figures	Tables (max/min, etc.)
Model Information	Geometry/mesh shape, material/property colors, boundary/load/CFD boundary condition labels, analysis case	Analysis summary, mesh set/material/property/function lists
Nodal Displacement	Displacement, rotation, velocity, acceleration (total/X/Y/Z)	Max/min displacement, velocity, acceleration
Nodal Force/Moment	Constraint force, MPC force, applied force (force/moment)	Max/min reaction, MPC, applied force/moment
Element Force/Moment	Rod, bar, and shell element forces	Max/min rod, bar, shell force/moment
Nodal Stress	Mesh density error	Solid/shell max convergence error
Element Stress / Element Strain	Rod, bar, shell, and solid stress/strain, von-Mises, safety factor	Max/min stress, strain, safety factor
Thermal Results	Temperature, thermal gradient, heat flux	Max/min nodal, 1D, 2D, 3D results
Optimization	Material density	Max/min material density
CFD Result	Pressure, velocity, density, turbulence, wall shear stress, vorticity, temperature, heat flux, level set, water level, displacement, electric potential/current density, radiation	Max/min pressure, displacement, velocity, energy and others

11-3. Report Function: Setting Tab and Generation

The Setting tab specifies the header, footer, and figure layout. Settings can be kept with [Save] and reused for later reports.



Report Generation window – Setting tab

1. Generation Procedure

1. Run Results > Special Post > Report
2. Select Analysis Case, Step, Template, and View Point
3. (When editing) Arrange contents in the Template tab and set the header and figure layout in the Setting tab
4. Check the output file path and click [Generate]: the Word report is created when the progress bar completes.
5. Enter the cover title, subtitle, date, and the project name in the header directly in Word.

2. Setting Items

Item	Description
① Header	When checked, an image (e.g. logo) and text are added at the top of each page. Specify the image path, text, and alignment (left/center/right).
② Footer	Specify the image and text at the bottom of each page in the same way as the header.
③ Number of images per page	Number of figures per page: 1, 2, 4 (default 2) / Image Title Position: Top, Bottom (default Bottom)
④ Delete temporary files	Deletes the temporary images used for figure capture after generation (checked by default). Insert All Contents: lists figure and table items in the table of contents as well.
⑤ Save	Saves the current settings.
⑥ Default Settings	Sets the default format (font, caption, etc.) of Group, Memo, Image, Table, Graph, and Text items.