

Multidisciplinary Integrated Analysis Solution for Optimal Design

midas NFX 2025 Introduction



midas NFX Introduction for update function 2025

Major update list (Release Version : 2025.02.07)

- > CAD Interface update
- > Graphic User Interface improvement (Ribbon menu, Tree menu)
- > Added Abaqus file (*.inp) import feature (for linear materials and structural elements)
- > Updated 3D Auto Mesh Generation Library
- > Added 3D Layered Mesh generation feature
- > Enhanced 3D Auto Mesh internal surface functionality
- > Added automatic 2D fillet welding generation feature
- > Added automatic Fluid-to-Structure model generation feature
- > Improved rendering speed in preprocessing environment
- > Enhanced usability for frequency/random response analysis
- > Added memory extension for structural analysis solver
- > Other improvements

midas NFX provides fully integrated and linked analysis for structure, thermal, fluid, and optimization within a single working environment using a single model. It features a Windowsbased GUI localized in Korean and offers a user-friendly environment with structured training and technical support through the MIDAS customer value system.



CAD Interface Update

The CAD Interface has been updated according to the CAD version update. Support for the latest versions may be delayed depending on the update environment of partner companies. If the latest version is not supported, please convert the file to Parasolid format for use. We will do our best to guickly incorporate the latest CAD versions.

Туре	File format	Applied version
Parasolid	x_t, xmt_txt, x_b, xmt_bin	9.0 ~ 37.0.118
ACIS	sat, sab, asat, asab	R1 ~ 2024.1.0
STEP	stp, step	AP203, AP214, AP242
IGES	igs, iges	Up to 5.3
Pro-E / Creo	prt, prt.*, asm, asm.*	16 ~ Creo 11.0
SolidWorks	sldprt, sldasm, slddrw	98 ~ 2024
CATIA V4	model, exp, session	4.1.9 ~ 4.2.4
CATIA V5	CATPart, CATProduct	V5 R8 ~ V5-6R2024
Unigraphics	prt	11 ~ NX2406
Inventor Part	ipt	V6 ~ V2025
Inventor Assembly	iam	V11 ~ V2025
SolidEdge	par, asm, psm	V18 ~ SE2024



Graphic User Interface improvement

<Development Purpose and Usage>

The GUI (Graphic User Interface) provided by NFX has <u>been completely redesigned to make the entire process of</u> <u>geometry, meshing, and analysis more intuitive and convenient</u>. Instead of displaying all available icons at once, they have been grouped and categorized based on user workflows, allowing for easier identification and application according to the required analysis.

The interface is structured <u>to follow the user's workflow step by step, progressing from left to right</u>. This ensures that only the necessary functions are displayed at each stage, enabling users to complete each step efficiently. Additionally, the interface supports multiple languages, including Korean, English, and Chinese, allowing users to switch as needed.

• Tree menu simplification: Quickly access only the necessary items!



• Shape tab icon reorganization: <u>Only the necessary items based on the workflow!</u> CAD / Creation / Modification / Dimension Change / Tools

Geometry Mesh S	tic/Heat Analysis Dynamic/Trans.Heat Analysis CFD Analysis Results Tools
Import Export → □ ∅ → CAD File → □ ↓ □ ↓	1 → 1 →
🚺 🔹 Geometry Mesh	Structural Static Analysis Structural Dynamic Analysis CFD Analysis Results Tools HKMC-RPS
🍇 🍇 📲 -••-	🖍 🗂 🗇 💷 🖓 🗊 💠 🖯 🎅 🔚 🗇 🕼 🧭 🛝 🥔 🦯
Import Export Work Point Plane *	Line Surface Solid Line Surface Solid Extrude Translate Scale Project Super Sub Shape Shape Shape Connect Extraction Extraction Elem-Face
CAD File Geor	ttry Creation Geometry Modification Rank Modification Tools

• Element tab icon reorganization: Only the necessary items based on the workflow! FE Model / Material & Properties / Generation Control / Creation / Modification / Tools

Geometry Mesh Static/Heat Analysis Dynamic/Trans.Heat Analysis CED Analysis Results Tools Application					
Matl. Prop. Comp. Prop.	.	(TOPO. ∰Extract ∰ Measure eters ∰Divide China & Smooth & Profile Shell Element Topole			
Attribute	Control Generate Proclude Pransion Node	Element 1005			
💕 🔹 Geometry	Mesh Structural Static Analysis Structural Dynamic Analysis CFD Analysis Results Tools HKMC-RPS				
	🖳 🖳 🔜 🖂 🛹 👁 🐨 🔜 🖬 📽 💊 🗰 🔍 🚺	I 🔟 🛛 🗣 📅 🎒 🍘 慉			
NFX Nastran ABAQUS	Matl. Prop. Comp. Control Match Layer 1D 2D 3D ETC Extrude Remesh Transform Node Connection Element Divid	le Parameters Check Mesh Renumber Measure Connect Table Quality Geometry *			
FE Model	Attribute Element Creation Control Element Generate Modify Node/Element	Tools			

Graphic User Interface improvement

<Development Purpose and Usage>

The GUI (Graphic User Interface) provided by NFX has <u>been completely redesigned to make the entire</u> process of geometry, meshing, and analysis more intuitive and convenient. Instead of displaying all available icons at once, they have been grouped and categorized based on user workflows, allowing for easier identification and application according to the required analysis.

The interface is structured <u>to follow the user's workflow step by step, progressing from left to right</u>. This ensures that only the necessary functions are displayed at each stage, enabling users to complete each step efficiently. Additionally, the interface supports multiple languages, including Korean, English, and Chinese, allowing users to switch as needed.

• STR / Static Load : More intuitive classification based on types of static loads!



• STR / Dynamic Load : More intuitive classification based on vibration and transient analysis types!

Geometry Mesh Static/Heat Ana	ysis Dynamic/Trans.Heat Analysis CFD	Analysis Results Tools				Application * Style *
Image: Self Matl. Prop. Matl. Prop. Matl. Prop. Matl. Prop. Manual Pin/Bolt Contact	Init Time Freq. Usp. Disp. Disp. Conduct Load	Joint Time Load Sing Time	Narker ody Load	Convec. (유 Cavity Cavity 유료ia. 상 Sensor 을 Define Set 중ource 蹉 Pipe Cooling Transient Heat Load	Transient Specific Transient Electric Potential Electric Current Transient Electric Load	General Analysis Case
d Compton High	Characterial Chattin Appeluite	Charles I Danaria A		Analysia Danaka Taala		
Geometry Mesh	Structural Static Analysis	Structural Dynamic Ana	ysis CFD	Analysis Results Tools	НКМС-КР5	
闷 🙀 🚑	: 🎰 🔛 🙌			上 🌺 🏬 👫	🚟 🏹 🎼 湃	🛃 💾 🎼
Define Structural Contact Pin/Be	olt Constraint Constraint Init Equation Velo	Time Freque	ncy Response	Temp. Flux Source Convec.	Radia. Cavity Pipe Electric Cooling Potential	Electric From Sensor Current Results
Common Condi	tion	Structural Dynamic A	alysis	Structural Analysis HeatTra	nsfer/Joule Heating (Transient Sta	te) Tools

• CFD Analysis : More intuitive classification based on fluid analysis modules!





Added Abaqus file (*.inp) import feature

<Development Purpose and Usage>

Support for Abaqus Input File (*.inp) Import has been expanded beyond the existing NFX and Nastran Standard Format. <u>This extension allows users to import and work with Abaqus files</u>. The development includes key materials, properties, 1D/2D/3D elements, connections, and special elements. Detailed importable items can be checked using keywords below.



keyword			
*ASSEMBLY *ENDASSEMBLY	*FASTENER		
*BEAM GENERAL SECTION	*FASTENER PROPERTY		
*BEAM SECTION	*INCLUDE		
*BOUNDARY	*INSTANCE *ENDINSTANCE		
*CLOAD	*KINEMATIC		
*COHESIVE SECTION	*MATERIAL		
*CONDUCTIVITY	*NODE		
*CONNECTOR SECTION	*NSET		
*CONNECTOR BEHAVIOR	*ORIENTATION		
*CONNECTOR ELASTICITY	*PART *ENDPART		
*COUPLING	*PLASTIC		
*DENSITY	*RIGID BODY		
*DISTRIBUTING	*SHELL GENERAL SECTION		
*DLOAD	*SHELL SECTION		
*DSLOAD	*SYSTEM		
*ELASTIC	*SPECIFIC HEAT		
*ELEMENT	*SURFACE		
*ELSET	*SYSTEM		
*EXPANSION	*TRANSFORM		

*FASTENER : Connetor

*ELASTIC : ISOTROPIC, ENGINEERING, LAMINA

*CONDUCTIVITY : ISOTROPIC, ORTHOTROPIC

3D Auto Mesh Generation Library Update

<Development Purpose and Usage>

To improve the success rate of mesh generation on curved surfaces in 3D Auto-Mesh creation based on the Parasolid Kernel, **enhancements have been made to address singularities and correct errors in commonly occurring geometries** such as Torus, Bsurf, and Sphere. Additionally, failed mesh element sets during generation are now separated and marked for clear identification.

Ongoing refinements and corrections are being applied to the Parasolid Kernel and Mesh Generation Library to further enhance the stability and reliability of automatic mesh generation.



midas NFX 2025

midas NFX 🚺

Added Layered Mesh generation feature

<Development Purpose and Usage>

The wall function option is commonly used to simulate velocity changes in the viscous sublayer region near the wall. Even when using wall functions, an element mesh that accurately represents the boundary layer characteristics is essential for precisely capturing velocity variations with respect to the wall distance. Now, with the boundary layer meshing function, you can easily generate multiple layers of fine mesh in regions with high physical variable gradients.

Generate mesh(Face) X				
Map-Area Boundary-Layer-Mesh(2D)				
Select A Tar	get Face			
Select Non-Lay	ered Edge(s)			
Mesh Size	30	mm		
Layer Control				
Number of Layers	3	▲ ▼		
 First Element Size 	2	mm		
🔘 Total Layer Size	0	mm		
Layer Growth Rate	1.5			
Property				
Mesh Set Boundary-Layer-Mesh(2D) ~				
Image: Concel Apply Apply >>				

Generate mesh(Solid)	×			
Auto-Solid Map-Solid 2D->3D Auto-Layered				
From Geometry	~			
Select O	bject(s)			
Size Method				
Size ODivision	30 <			
First Layer Height	0.5			
Number of Layers	3			
Layer Growth Rate	1.3			
Total Layer Height	1.99			
Except Surface				
Select O	bject(s)			
Inner Element Generation	Туре			
• Tetra dominant (Hexa dominant			
Collision Avoidance Metho	bd			
O Compression O Decrease				
Smooth Element Normal	Vector Transition			
Multi-Normals Angle 30				
Delete Source Mesh Set				
Property				
1	~ HD			
Mesh Set Boundary-Layer-Mesh(3D) ~				
🔯 🥖 📫 OK Cancel Apply >>				



- Two layer specification options are provided (first element size, total layer size).
- Boundary layer exclusion face selection function is available (maintains boundary layer thickness at the edge)
- Includes geometry-to-mesh relationship settings.

[3D layered mesh generation]



MIDAS

Enhanced 3D Auto Mesh internal surface functionality

<Development Purpose and Usage>

The internal face function, one of the 3D Auto Mesh features, is widely used in fluid analysis. Errors were identified in cases where the geometry was split, and improvements were made by extending the functionality to **include single cutting**, **orthogonal division**, **and multi-face splitting**.

It <u>can be used for thin plates that may degrade element quality, guide vanes affecting fluid flow,</u> <u>and internal fan conditions requiring pressure jumps.</u> Previously, it was limited to single face (Face) types, but now it has been expanded to include shell (Shell) type assemblies.





Added automatic 2D fillet welding generation feature

<Development Purpose and Usage>

Various welding functions are used across many industries. Following the implementation of spot welds, <u>a 2D fillet weld feature has been added for application to 2D shell elements</u>. Based on the user-specified 2D element edge (From), the system automatically performs remeshing on the target element (To) while considering the fillet size and type.

Users can preview the placement based on the From element or custom-defined values, allowing them to check the automatic alignment and connection results.



MIDAS

Added automatic Fluid-to-Structure model

<Development Purpose and Usage>

In real-world scenarios, most cooling phenomena occur in fluids, except for electrical effects like Peltier. One of the primary purposes of CFD analysis in industries is to evaluate cooling efficiency and temperature distribution in key components.

Previously, FSI INTERFACE and coupled analysis cases were provided, but defining mesh and conditions for overlapping regions was somewhat inconvenient. To streamline the process and improve usability, a new feature has been added that <u>automatically defines a structural element</u> <u>model sharing the same nodes and a volume-based FSI INTERFACE using user-defined structural material and property values.</u>

Even when performing the analysis manually, users can easily map node-based data using the general result conversion function.

Remesh CFD->Str CFD->Str CFD->Str	Mesh Move/Copy X Cfd->Structure (2D) Cfd->Structure (3D) Select Mesh Set(s) Select Object(s) FSI Interface Make FSI Interface (Temperature) Property 1 Mesh Set Copied Mesh Set Image: Copied Mesh Set Image: Copied Mesh Set Image: Copied Mesh Set <th></th>	
ICED Model1	• Auto generate FSI interface(volume)	ESI INTEREACE - Volume1



Added automatic Fluid-to-Structure model

<Development Purpose and Usage>

In real-world scenarios, most cooling phenomena occur in fluids, except for electrical effects like Peltier. One of the primary purposes of CFD analysis in industries is to evaluate cooling efficiency and temperature distribution in key components.

Previously, FSI INTERFACE and coupled analysis cases were provided, but defining mesh and conditions for overlapping regions was somewhat inconvenient. To streamline the process and improve usability, a new feature has been added that <u>automatically defines a structural element</u> <u>model sharing the same nodes and a volume-based FSI INTERFACE using user-defined structural material and property values.</u>

Even when performing the analysis manually, users can easily map node-based data using the general result conversion function.



Improved rendering speed in pre-processing

<Development Purpose and Usage>

As large-scale models have become more common, increased use of complex geometries and mesh elements has led to a decline in rendering speed when manipulating models. To address this, <u>the body information processing method in the existing rendering system has been restructured to allow parallel processing.</u>

Validation was conducted on various cases, including repetitive geometric shapes, large mesh structures, automotive assemblies with over 2,000 parts, element models with over 2 million elements, and large-scale CFD models. <u>The results showed an average rendering speed</u> <u>improvement of over 400%</u>.

Case 1 : 50,000 Cubes (600,000 Elements)
 : 548 ms (1.8 fps) → 28 ms (35.7 fps) … 1,857% Improve



• Case 5 : CFD model : 378 ms (2.6 fps) → 40 ms (25.0 fps) … 845% Improve



FPS Improvement (Before vs After)



* Case 2/3/4 Due to security concerns, real-world models cannot be disclosed. (Over 2,000 Parts, Over 2 million elements model)

Enhanced usability for frequency/random response analysis

<Development Purpose and Usage>

Recently, as the complexity and scale of frequency/random response analysis have increased in industries such as ESS, defense, and automotive, there has been a growing need for clusterbased methods and definitions for irregular forms. Previously, in NFX, users had to manually input irregular frequency sets, but this has now been **improved to a table format that allows data to be copied and pasted from Excel.** This enables quicker verification and application of necessary data alongside mode table results.

Additionally, in random response analysis, structural safety and vibration fatigue evaluation are typically conducted using moment, RMS, and NPX values, reducing the need for single-response outputs. As a result, single-response data is now <u>hidden by default, and users can selectively</u> <u>output results for desired frequency sets through subcase analysis settings.</u>

- Frequency Set × Add Frequency List Modify Method Discrete Delete Frequency (Cycle/sec) 47.3827 Copy & Paste 58.5865 63,9838 63.9838 110.6198 110.7369 121.9946 251.2217 251 2316 .232152e-12 .427395e-09 .003147e-04 .551437e-04 .331462e-12 1.697376e-0 5.443071e-7.909024e-5.480075e-0 3.6528936-0 1.526492e-0 2.054441e-0 4.031794e-0 3.442087e+0 7.215006e+0 1.013853e+0 6.057311e-0 9.126125e-0 2.200700-1.163999e-0 1.265157e-0 3.875907e-1 2.715248e-(1.019702e-105305e-04 1.689601e-0 9.191318e+0 7.312890e+0 Method 876742e-221754e-02610e 009019e-893319e-2.831774e-0 .652277e+0 3 005797e-03 3 005797e-03 3 005797e-03 2 445159e+01 4 111565e+01 1753344e+01 Close
- > Changed the definition method for discontinuous frequency sets (Copy & Paste enabled).

► Modified the result addition method in random response analysis (Default: Moment, RMS, NPX output).

→ Users can right-click on the subcase > Add Analysis Results

는 過 Analysis Case	Insert Analysis Results	X Insert Analysis Results X
	Insert Analysis Results Step Results Ster 1 Interval Apply Name Interval IcAon 1: FREQ=1.9555e+03 IcAon 1: FREQ=1.9555e+03 IcAon 1: FREQ=1.9555e+03 IcAon 1: FREQ=1.9556e+03 PSD : FREQ=1.9684e+03 PSD I: FREQ=1.9684e+03 PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist) PSD MOMENT(rist)	Insert Analysis Results A Step Results Velocities Velocities Nodel Miss. Results Deplacements Grid Forces Shell Bernent Stresses Sold Stresses Sold Stresses Sold Stresses T1 VELOCTTY [P] (V) T3 VELOCTTY [P] (V) R2 ANGULAR VELOCTTY [P] (V) R3 ANGULAR VELOCTTY [P] (V) Sel All Unsel All Sel All Unsel All
	OK Close	OK Close



Added memory extension for structural analysis solver

<Development Purpose and Usage>

Added Memory Extension option for structural analysis solver to allow users to manually adjust memory allocation. If physical memory is insufficient, users can enable virtual memory using the hard disk to prevent slowdowns. Recommendation: <u>Ensure sufficient disk space before enabling</u> <u>virtual memory, as excessive memory usage may overload the model.</u>

If the hard disk has enough free space (e.g., 100 GB), the software will automatically allocate the required memory for analysis. <u>Helps compensate for memory shortages and improve analysis</u> <u>speed.</u>



Test model (2D	Shell Element + RBE2, Weld)
Elements	: 1,891,593
Nodes	: 1,941,628
DOFs	: 11,340,769
Equations	: 11,297,653

STR. Analysis Option Control X	Case	Available Memory (GB)	Memory Extension (GB)	Usage Memory (GB)	Working Time (sec)	CPU Clock (GHz)
Resource Utilization Plan		(00)	(00)	(00)		
Number of Processors 8	Α	32	0	20.33	1585.28	2.5
Enable Memory Extension 102400 [MByte]	В	32	<mark>32</mark>	<mark>26.99</mark>	1819.17	2.5
Enable GPU Acceleration	С	32	<mark>100</mark>	<mark>49.02</mark>	786.00	<mark>2.5</mark>
[Notice] If memory extension enabled, enter the sum of physical and virtual memory.	D	128	0	49.08	499.77	<mark>3.6</mark>
If memory extension disabled, available memory will be estimated and used. Analysis may be interrupted due to incorrect memory extension.	A. 8 CPU TH [SYSTEM II NUMBER (MAXIMUM AVAILABLI TOTAL CP WALL CLO TOTAL WA	DIFERENCE OF THREADS : 8 MEMORY USAGE MEMORY : 20 UTIME : 2387. DCK TIME : 1583 ARNINGS : 17	ry Extension) : 158 : 11217 MB(MAIN), 826 MB .56 sec(MAIN), 110.2 5.28 sec	5.28 sec 20762 MB(MFSEXE 34 sec(MFSEXE.EXE		
Element Formulation B. 8 CPU Threads + Memory Extension 32GB : 1819.17 sec (114.7%) Hybrid (Accuracy) [SYSTEM INFO] Reduced (Efficiency) NUMBER OF THREADS : 8 MAXIMUM MEMORY USAGE : 11228 MB(MAIN), 27558 MB(MFSEXE.EXE) AVAILABLE MEMORY : 27633 MB						
Equation Solver	WALL CLO TOTAL WA	OCK TIME 1819	9.17 sec	SEC(INIFSEAE.EAE)		
O Auto ○ Multifrontal ○ Dense ○ AMG	<u>C. 8 CPU T</u>	nreads + Memory	Extension 100GB:	786.03 sec (49.6	<u>%)</u>	
Maximum Iteration 1000	[SYSTEM I	NFO] OF THREADS : 8				
Convergence Tolerance 1e-06	MAXIMUM MEMORY USAGE <u>50196 MB</u> AVAILABLE MEMORY SAGE <u>50197 MB</u>					
2D Element Setting	WALL CLO TOTAL WA	O TIME : 742.5 OCK TIME 786. ARNINGS : 18	003 sec			
Unique Shell Normal Generation 20 [deg]	D.8 CPU Th	reads (No Memor	ry Extension) : 499.	77 sec		
Control Transverse Deformation (Hybrid) Set Default OK Cancel	[SYSTEM II NUMBER C MAXIMUM AVAILABLE TOTAL CPU WALL CLO	NFO] DF THREADS : 8 1 MEMORY USAGE E MEMORY : 985 U TIME : 1290. CK TIME : 1290. CK TIME : 1499.	50253 MB 716 MB 66 sec 767 sec			



Other improvements

<Improvement in Geometry-Based CFD Monitoring Definition>

During CFD monitoring, an issue was identified where monitoring conditions remained even after the monitored surface was modified or removed, leading to abnormal termination. To address this, unused monitoring conditions are now automatically deleted, and a comment is logged in the output window. Additionally, previously inaccessible files due to this issue can now be opened properly.

<Improvement in Heat Flux (Convection) Handling in Mixture Flow Analysis Module>

When performing mixture flow analysis with flux-type loads, it was found that heat flux information was being ignored as the solver received both mixture flux and heat flux data. This issue has been fixed to ensure that heat flux (convection) is correctly applied even in mixture flow conditions.

<Modification of Fatigue Analysis Formula in Random Vibration Analysis>

An error in the calculation of certain parameters in the Steinberg method for fatigue analysis in random vibration analysis was identified and corrected. For existing analysis files, re-analysis of the fatigue analysis is required if verification using the updated formula is necessary.

<Unit Conversion Error in NFX CAD Export>

An issue was identified where certain models were being converted to 1/100 of their original size during CAD export using the Parasolid Kernel provided by NFX. This has been corrected to ensure that the models are saved according to the current working unit system.

<Correction of Missing Temperature Data in CFD .res Result Files>

An error was identified where temperature values were missing in the *.res result files of CFD analysis when used for load generation. This issue has been resolved by enabling the addition and conversion of node-based temperature data, allowing for faster data mapping in fluid-structure transformation functions.

<Expansion Function Enhancement for Element Visibility>

Previously, the Expand function in element visibility allowed checking only one adjacent target. The new "Expand All" feature enables full expansion, and welds and contacts are now included as expansion targets. This helps verify model connectivity, and the inverse function allows users to identify unconnected parts in advance.

<Improvement in Energy (W) Definition for Heat Flux and Heat Generation in Structural Analysis>

In structural analysis, heat flux and heat generation conditions were previously applied as secondary calculated values in W/m² and W/m³, causing inconvenience for users. A new feature has been added to allow direct definition of total energy (W), which is automatically applied by the solver based on the selected target.

Other improvements

<Addition of Multi-Node Connection and Flow Coupling Grouping in 1D Flow Analysis>

In 1D flow analysis, conditions are now inherited when copying elements. Additionally, flow coupling has been expanded to allow batch selection and application of 3D volume surfaces and 1D nodes in manifold structures, enabling faster multi-definition workflows.

<Fix for Graphic Distortion Issue When Checking Minimal Geometries in Simplification Feature>

An issue was identified where extremely small CAD items would cause graphic distortion or disappear when double-clicked for repositioning due to a scaling error. This problem, which resulted in a reduced screen view, has been identified and corrected.

<Improved Program Uninstallation Speed>

Previously, CAD libraries and font files from third-party applications were added to the system during installation, causing delays in the uninstallation process. To resolve this, the system no longer installs fonts. Depending on the user's environment, the previous uninstallation process could take up to 5 minutes. With the updated version, the process now starts immediately and completes within 30 seconds.

