MIDAS NFX STR

TOTAL SOLUTIONS FOR TRUE ANALYSIS-DRIVEN DESIGN

MIDAS NFX STR

MIDAS

MIDAS
NFX STR

TOTAL ANALYSIS SOLUTIONS FOR OPTIMUM DESIGN IN MULTI-DISCIPLINES

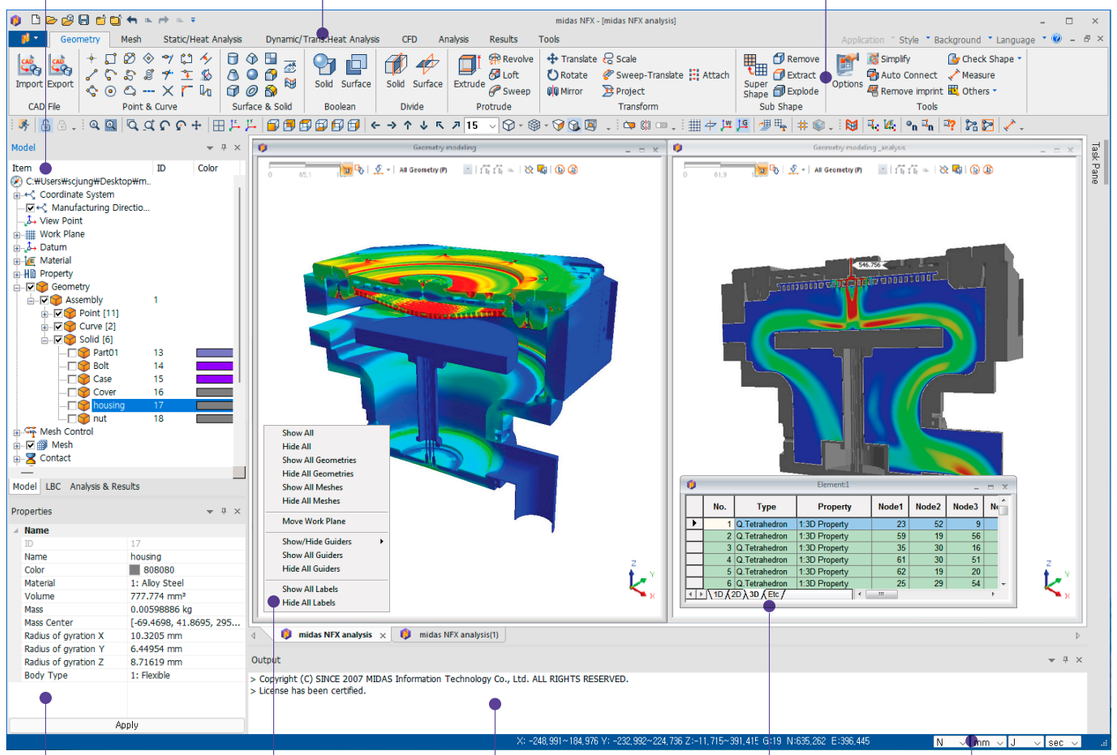
SIMPLIFIED FRAMEWORK
FOCUSED ON EASY USER INTERFACE

Operation Environment

Works Tree
Provides model data in a tree structure and useful functions for data management and modeling

Ribbon Menu
Provides related works in panel type

Tabbed Toolbar
Provides categorized toolbar icons



Properties Window
Provides various information and an editing function

Context Menu
Provides frequently used menus depending on selected entities

Message Window
Provides various information and operation results in modeling

Table Window
Provides input data and result values in Excel-like tables

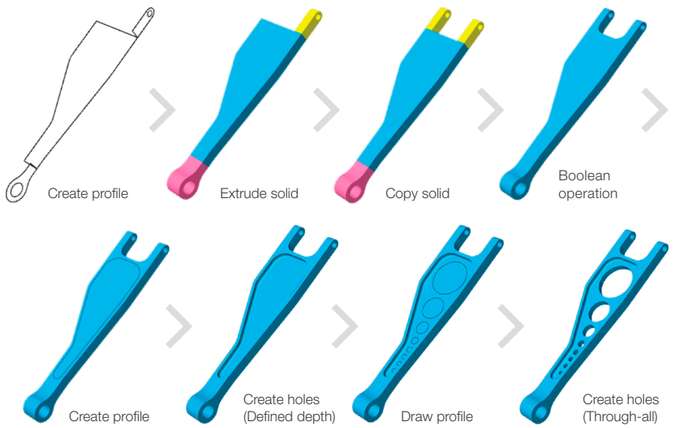
Unit Manager
Provides real time unit conversion

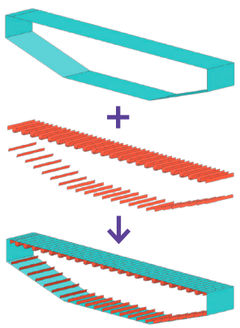
GRAPHIC USER INTERFACE

Geometric modeling

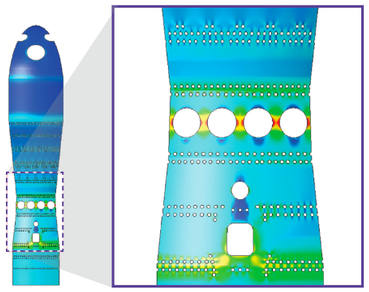
Mid-range of CAD modeling functions
for a variety of practical geometric modeling.

Solid Modeling Example

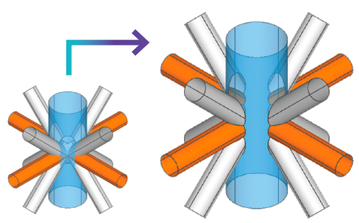




Connection of stiffners using sew and fuse functions of surface (Non-manifold Surface creation)



Various shapes and hole sizes by trim and surface split by line



Trim 2 surfaces based on intersection line

Surface

- Surface : plane, Coons, NURBS, point interpolation
- Extrude, revolve, sweep, loft
- Fillet, chamfer, offset
- Fuse, sew (end-connect, mid-intersect, approximate)
- Trim, extend, imprint of point/curve on surface
- Trim by surface/curve

Solid

- Primitive: box, cylinder, sphere, torus, cone
- Boolean operations: fuse, cut, common
- Extrude, revolve, sweep, loft
- Trim, divide, draft, shell
- Fillet, chamfer, create hole

Curve

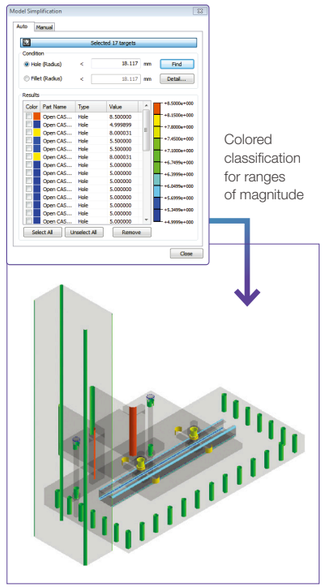
- Line, polyline, arc, circle
- Rectangle, polyline
- Spline, profile, spiral
- On-face curve
- Intersect line on surface, shortest line, tangent
- Trim, extend, fillet, chamfer, offset
- Merge, divide, make wire (grouping)

Geometry manipulation

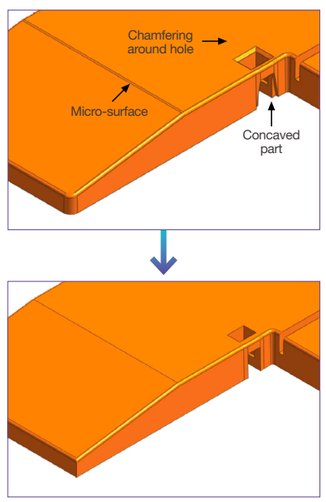
- Explode, compound
- Model check: topology, overlap
- Search/delete small surface/curve
- Measure: area, length, distance, angle
- Move: translate, rotate, mirror, scale
- Remove: hole, interior (imprint) point/line

Automatic Cleanup

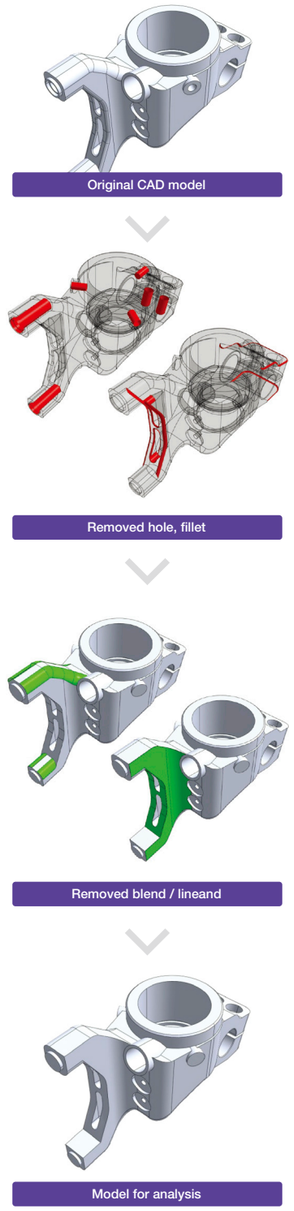
The automatic cleanup function of midas NFX can conveniently clean up features such as small holes and fillets that are not essential for analysis.



Intuitive interface for searching and distinguishing objects to be removed



Cleanup process for various shapes/parts (automatic/general)



The cleanup function can be applied automatically when importing a CAD model. Or features can be conveniently searched, checked and deleted in the cleanup wizard without any complicated manual work.

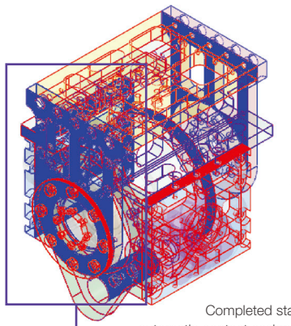
Main automatic cleanup functions

- Remove holes, fillets, projections
- Remove/merge micro-surfaces
- Check and modify topology

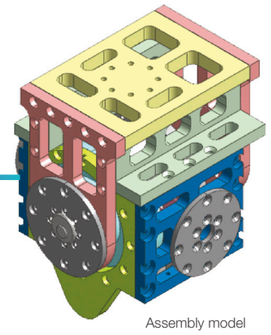
GRAPHIC USER INTERFACE

Definition of Contact

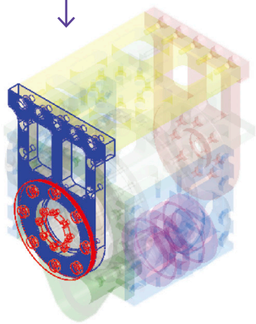
Automated contact definition suitable for complex,
large scale assemblies and convenient visualization and management.



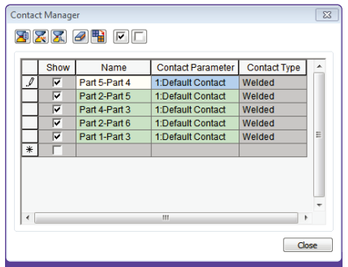
Completed state of
automatic contact assignment
(total model check)



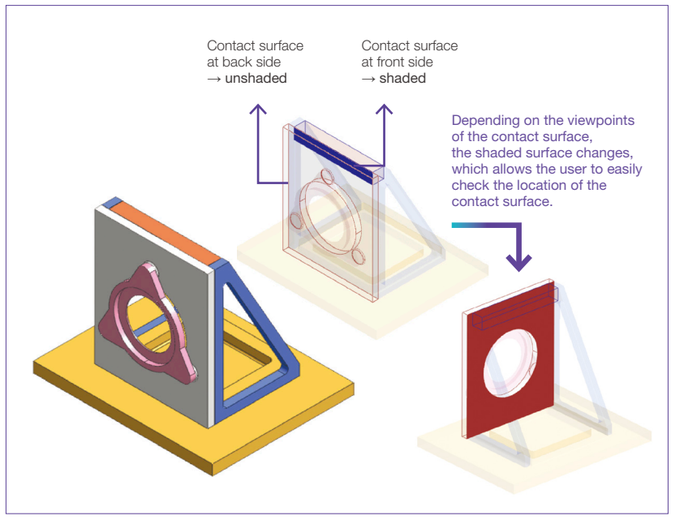
Assembly model



Individually check each contact definition



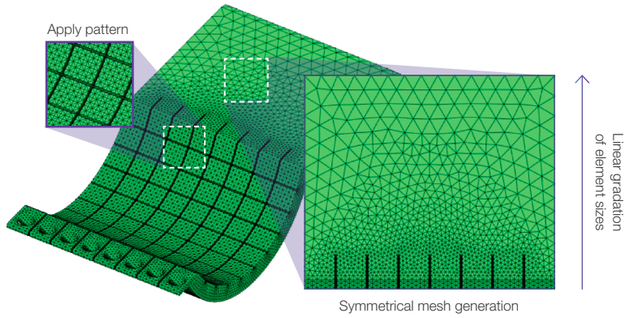
Contact manager to conveniently check, revise and manage contact definitions



Even for a complex assembly model, contacts are established by automatic calculation of distances between the parts without having to check every contact condition between the parts. The defined contacts can be clearly checked through visual representations.

Also, by using the contact manager function, the essential information of the defined contacts can be readily checked and simply revised.

Mesh Generation



Automatic Generation

Auto-mesh generation
: Surface, solid & plane domains
: 2D → 3D. Element based regeneration
: Densification including internal points/
curves Mapped mesh generation
: Surface, solid, 4-nodes
: Curve/surface defined volume

Density Control

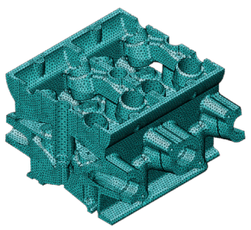
Density Control
: Element length, number of divisions,
length ratio
: Linear gradation, symmetrical distribution
: Mouse click assignment, table input,
matching Property assignment and check
Default value assignment and use
Division of patterns and density control

Protruded Generation

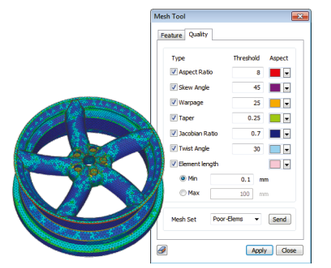
Extrude, revolve, sweep project, offset, fill
Node → 1D element extrusion
Curve → 2D element extrusion
2D element/solid element surface
→ 3D extrusion
Equal, unequal interval extrusion
Extrusion based on geometric entities, nodes and elements

Manipulation Function

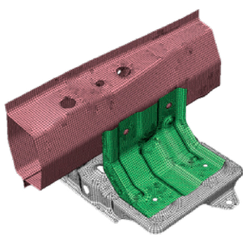
Node/element table
Node/element group
: Definition, name change, Boolean operation
Change in element parameters (order, etc.)
Check : connection condition, element quality
Move : translate, rotate, mirror-copy, scale
Group calculation : union, intersection,
difference of sets, XOR



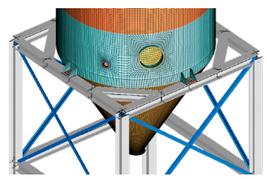
Auto-meshed solid elements



Graphics based convenient element mesh quality check
and group classification



Auto-meshed shell elements



Freedom of combining of solid, shell and frame elements
(frame cross-sections displayed)

midas NFX enables both expert and novice users to easily generate optimum mesh for analysis through a number of mesh generation options.

- Surface Auto-Mesher
- Solid Auto-Mesher
- Map-Mesher
- Manual extraction of higher order element mesh
- Element based mesh regeneration
- Element mesh generation including internal points and curves
- Assignment of mesh densities to internal element meshes
- Offset element creation around internal holes
- Adaptive analysis reflecting geometric shapes

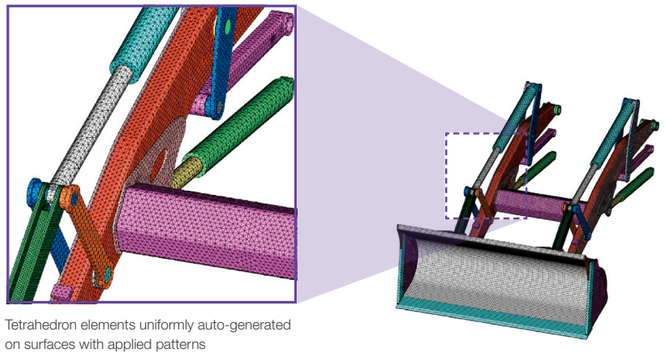
midas NFX contains practical functions to improve and manipulate element mesh of high quality. midas NFX also provides various management and checking functions to conveniently manage complex models.

- Automatic group creation by parts
- Element mesh check
- Element mesh quality testing
- Checking and aligning element coordinate systems
- Division of element patterns
- Renumbering nodes/elements

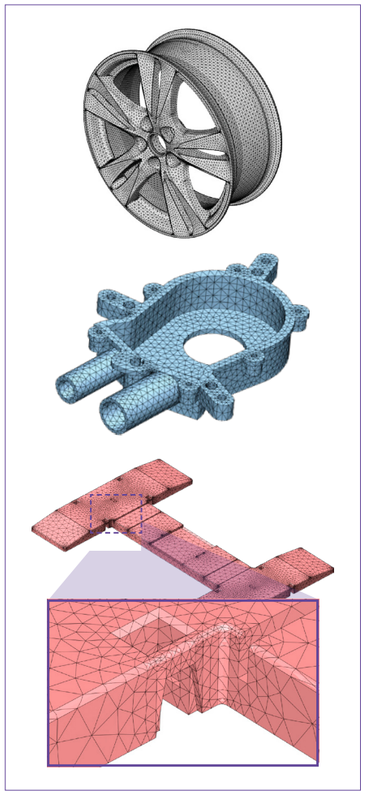
GRAPHIC USER INTERFACE

Solid Automatic Mesh

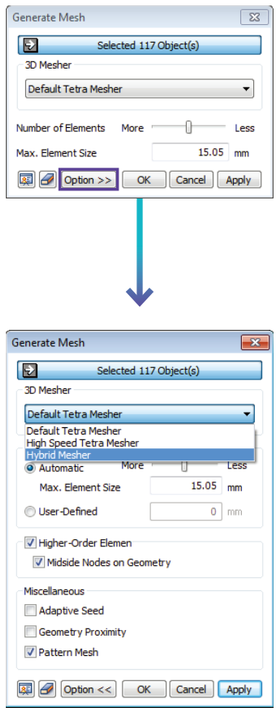
Total Solutions for True Analysis-driven Design High-performance, high-quality auto-mesh generation for optimum finite element mesh.



Tetrahedron elements uniformly auto-generated on surfaces with applied patterns



Auto-generated tetrahedron mesh with the automatic control of mesh density reflecting the shape characteristics such as curvature and proximity to holes

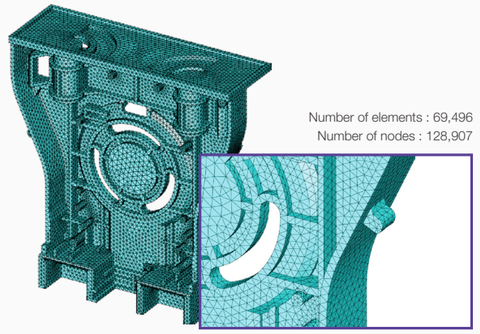


Various practical control options for element mesh density and shape

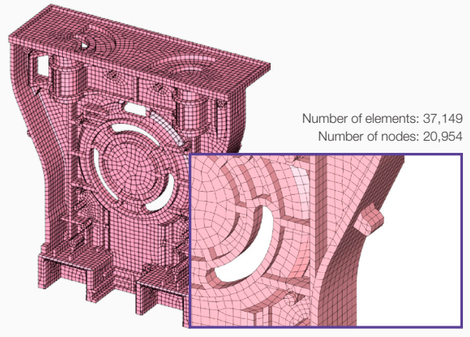
midas NFX generates high-quality, hexahedron dominant element mesh even for solid models of complex shapes. As such, the number of nodes and hence the analysis time become significantly reduced. Especially the boundaries generally consist mostly of hexahedra, the results of which are superior to other element types.

midas NFX supports parallel processing that utilizes multi-cores during mesh generation. Even for a large scale assembly model consisted of tens and hundreds of parts, many parts are simultaneously meshed, which results in a significant reduction in the total mesh generation time.

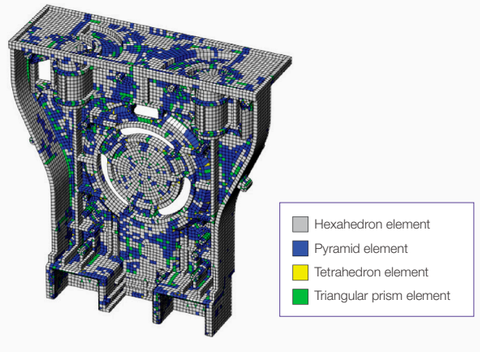
Hybrid Element Mesh (hexahedron-tetrahedron hybrid element mesh)



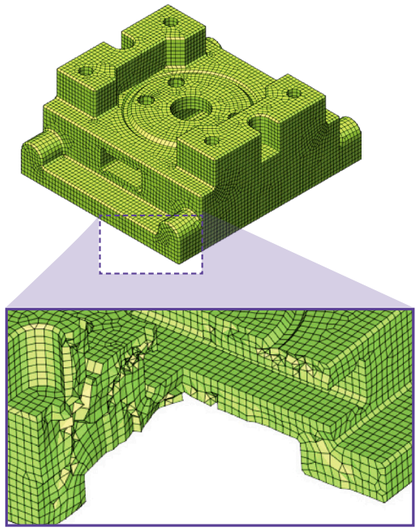
Tetrahedron element mesh modelgenerated by Tetra Mesher



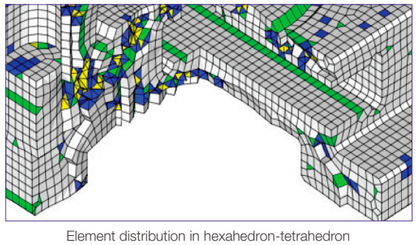
Hexahedron-tetrahedron hybrid element mesh model generated by Hybrid Mesher
(50% reduction in the number of elements & 80% reduction in the number of nodes compared to traditional mesh of similar element sizes)



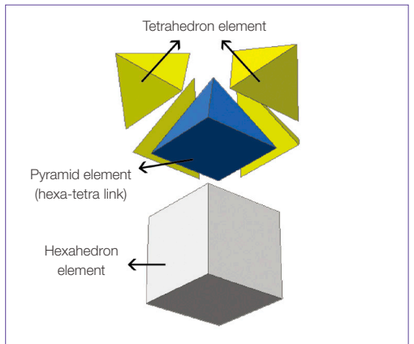
Distribution of hexahedron-tetrahedron hybrid elements
(colored display based on element types)



Hexahedron elements are primarily generated at the boundaries where high stresses are resulted. Tetrahedron elements are partially generated at interiors where stiffness and mass calculations are more meaningful.



Element distribution in hexahedron-tetrahedron hybrid element mesh
(colored display based on element types)



Composition of hybrid element mesh

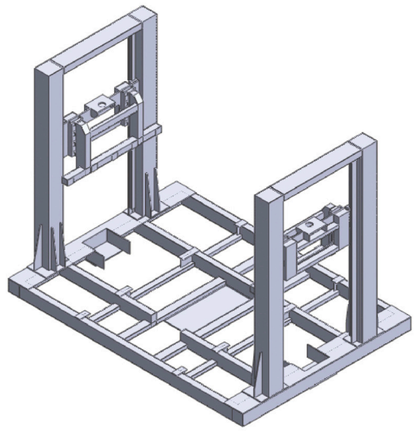
GRAPHIC USER INTERFACE

GUI

Mesh generation by parallel processing

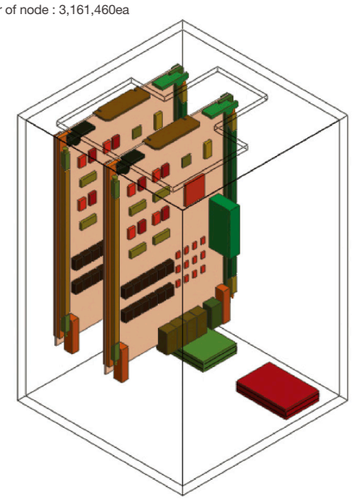
Comparison of automatic mesh generation speed according to the number of cores **(1)**
(assembly model of a total of 23 parts)

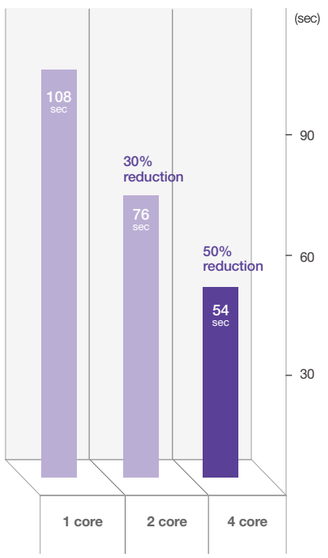
number of element : 580,076ea
number of node : 930,327ea

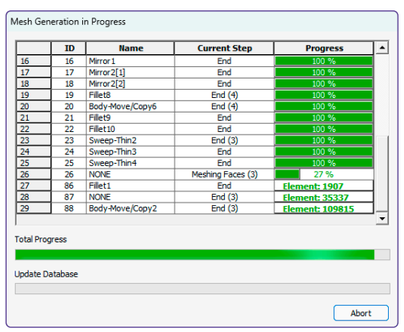


Comparison of automatic mesh generation speed according to the number of cores **(2)**
(assembly model of a total of 129 parts)

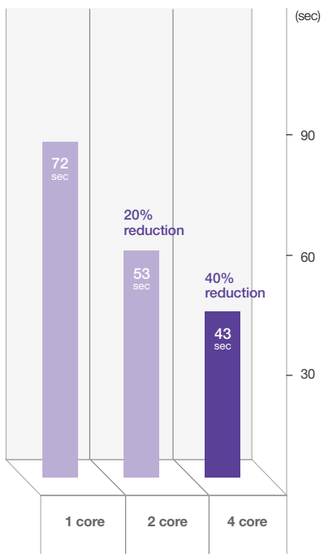
number of element : 2,115,758ea
number of node : 3,161,460ea







Parallel mesh generation process using multi-core
for large-scale assembly models graphic

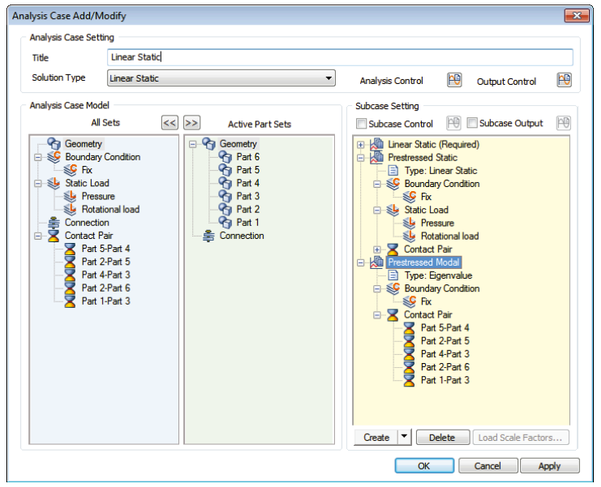


ANALYSIS FUNCTION

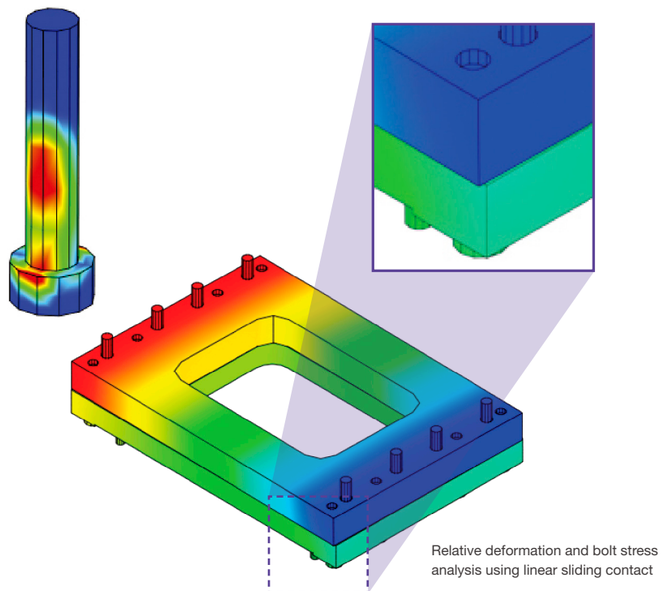
SOLVER

Linear Static Analysis

Using the superb analysis performance and the linear contact function of the high performance parallel processing solvers (multifrontal & AMG), models of any complexity can be analyzed quickly and accurately.



Multiple number of analysis cases for a single project model
and the results of the analysis cases compared after analyses
(Intuitive user interface consisted of a tree structure and Drag & Drop method)



Relative deformation and bolt stress
analysis using linear sliding contact

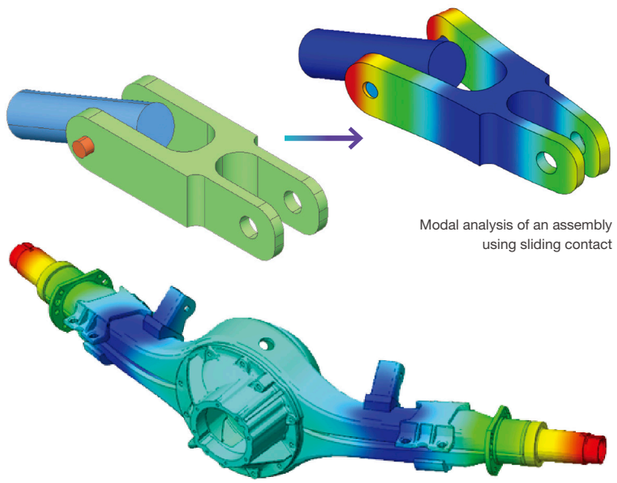
- Linear stress, displacement and safety factor calculations
- Linear contacts : single-body motion, sliding, interpolation link
- Pre-stress function
- Diverse and yet practical loads and boundary conditions
  - Loads : self-weight, centrifugal force, concentrated load, moment, temperature, pressure, beam load, pipe internal pressure, remote load, bolt load, etc.
  - Boundary conditions : constraint condition, symmetrical condition, MPC condition, etc.
- GUI based subcase definition, calculation of results and transformation of result coordinate system
- Outstanding analysis speed due to high performance parallel solvers
  - Direct method : multifrontal solver
  - Iterative method : AMG solver
- Checking practical analysis results (convergence error caused by mesh density, etc.)
- Extraction of stress results using surface elements
- Inertia Relief

ANALYSIS FUNCTION

MIDAS NFX STR

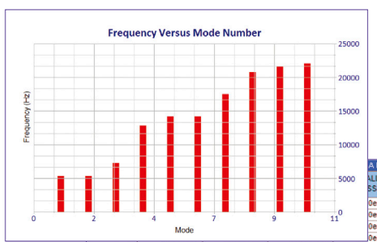
Modal / Buckling Analysis

Using the Block Lanczos solver, fast eigenvalue analysis becomes possible for a large scale model. In a complex assembly model, the modes of behavior can be effectively calculated using the linear contact function reflecting the relative motions between the parts.



Modal analysis of an assembly using sliding contact

Modal analysis of an automobile axle (7th mode, Free-Free condition)



				ALUES			
				ALIZED SS	GENERALIZED STIFFNESS	ORTHOGONALITY LOSS	ERROR MEASURE
				0e+000	1.14e+009	0.00e+000	1.93e-009
				0e+000	1.14e+009	4.87e-010	1.34e-010
				0e+000	2.10e+009	1.59e-011	1.13e-011
				0e+000	6.49e+009	2.80e-010	8.41e-011
5	7.97e+009	8.93e+004	1.42e+004	1.00e+000	7.97e+009	2.36e-011	3.52e-011

MODAL EFFECTIVE MASS

MODE NUMBER	T1	T2	T3	R1	R2	R3	
1	2.76e-006	5.15e-003	0.00e+000	3.10e+001	1.66e-002	1.23e+001	
2	5.15e-003	2.76e-006	0.00e+000	1.66e-002	3.10e+001	1.35e+001	
3	0.00e+000	0.00e+000	0.00e+000	7.52e-015	2.06e-015	1.05e+001	
4	0.00e+000	6.56e-015	6.01e-003	1.50e+001	1.50e+001	2.22e-011	
5	3.49e-005	1.48e-003	0.00e+000	1.88e-001	2.54e-003	4.93e+000	
TOTAL	5.19e-003	6.63e-003	6.01e-003	4.62e+001	4.61e+001	4.12e+001	
TOTAL IN MODEL	7.86e-003	7.86e-003	7.86e-003	0.00e+000	0.00e+000	0.00e+000	

PERCENTAGE MODAL EFFECTIVE MASS

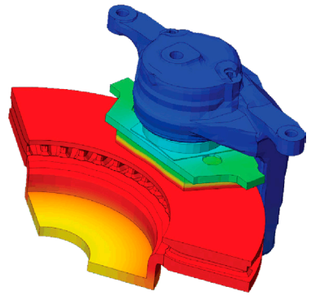
MODE NUMBER	T1	T2	T3	R1	R2	R3	
1	0.04%	65.49%	0.00%	0.00%	0.00%	0.00%	
2	65.49%	0.04%	0.00%	0.00%	0.00%	0.00%	
3	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%	
4	0.00%	0.00%	76.45%	0.00%	0.00%	0.00%	
5	0.44%	18.85%	0.00%	0.00%	0.00%	0.00%	

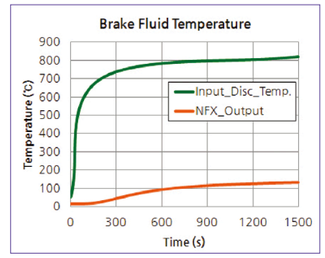
Example of numerical results table and graph for a modal analysis

- Natural frequency, mode shape, mode participation factor, effective mass results and calculation error check Define the range of eigenvalues to be calculated
- Sturm Sequence check (check for missing eigenvalues)
- Linear contact function : single-body motion, sliding, interpolation link
- Prestress considered (prestress modal)
- Mode Assurance Criterion (MAC)
- Consistent mass, lumped mass
- Results check identical to that of linear analysis (stress, strain, etc.)
- Buckling analysis possible for all the elements including composite material solids

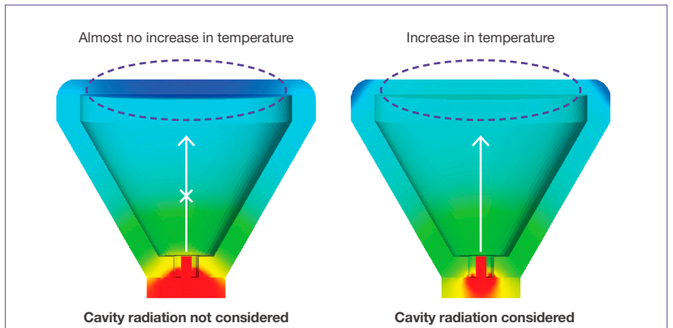
Heat Transfer / Heat Stress Analysis

NFX STR offers practical heat transfer and heat stress analysis capabilities. Especially heat stress analysis is provided as an independent analysis case. As such, a single analysis can produce temperature results by heat transfer and thermal deformation/thermal stress results.

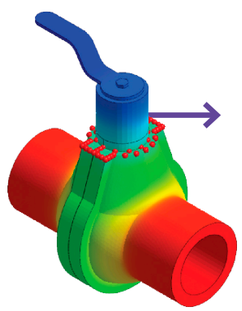




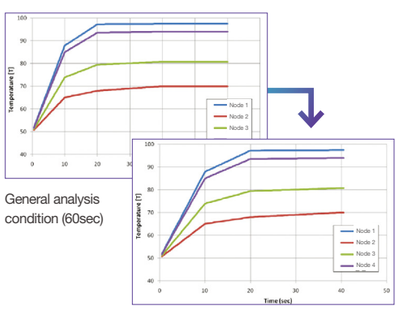
Heat capacity analysis of a brake system (transient heat transfer, heat contact applied)



Heat transfer and temperature distribution by heat cavity radiation inside an LED lamp



Transient heat transfer analysis of a ball valve (using sensors)



General analysis condition (60sec)

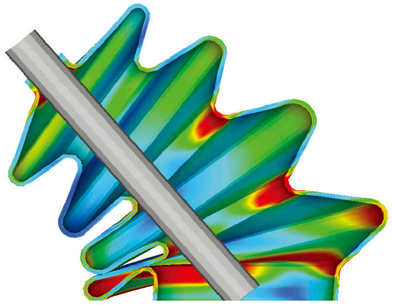
Termination of results at 40.5 sec when using sensors

- Steady and transient heat transfer analyses
  - Restart function
- Nonlinear heat transfer analysis function considering temperature-dependent materials and conditions
- Various load conditions
  - Heat generation, conduction, convection,radiation, heat flux, joule heating, initial temperature, fixed temperature conditions
- Thermal contact function to simulate heat conduction between discontinuous parts
- Heat transfer analysis function considering cavity radiation
  - Open/closed conditions
  - Radiation shape factor calculation
- Effective transient heat transfer analysis using sensor
- Automatic termination of analysis based on standards
- Minimum / maximum / average temperatures in a selected domain defined under the sensor conditions

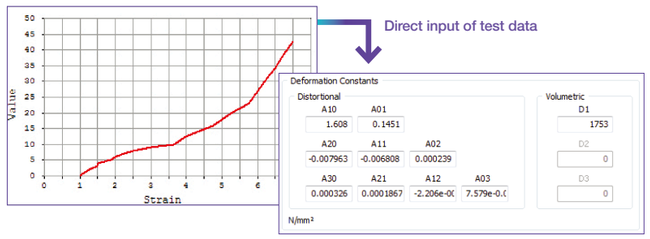
ANALYSIS FUNCTION

Nonlinear Analysis

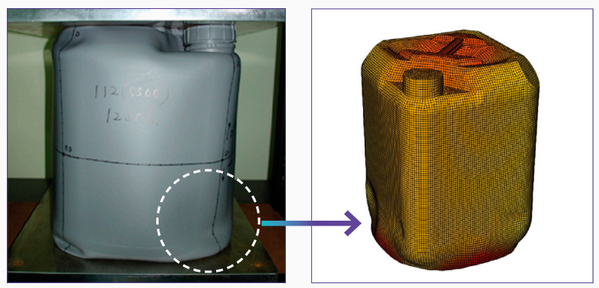
NFX STR provides excellent convergence and effectively undertakes material, geometric and contact nonlinear analyses.



Non-linear contact analysis of boot seal (single-sided contact)

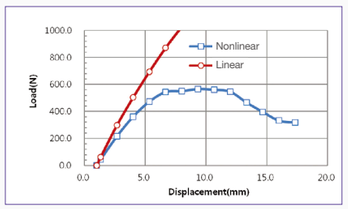


Rubber material property definition



Test results

NFX analysis results



Comparison with linear analysis

Material nonlinearity

- Material models : elastoplastic, hyperelastic
- Hardening behaviors : isotropic, kinematic, combined
- Hyperelastic material models : Mooney-Rivlin, Neo-Hookean, Polymoial, Ogden, Blatz-Ko, etc.

Geometric nonlinearity

- Large displacement and large rotation considered using the Updated Lagrangian method
- Follower force : pressure, gravity force, concentrated load, etc.

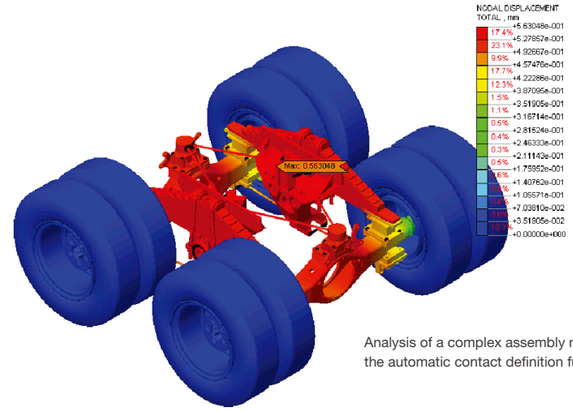
Contact nonlinearity

- Two-dimensional point-line/line-line contact
- Three-dimensional surface-surface/ lineline/point-surface contact, single surface contact
- Contact behaviors : single-body motion, sliding, rough contact, general contact, interpolation link, friction

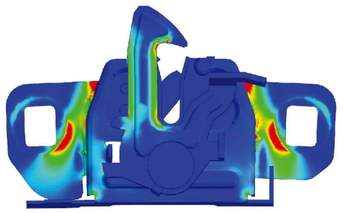
Various load increments

- Automatic load increments
- Quasi-static load increments using functions

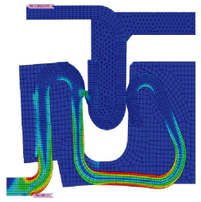
: Various iterative methods, stiffness update method and convergence criterion method
: Composition of continuous / independentload conditions
: Status of convergence and interim results during analysis, re-analysis (restart)



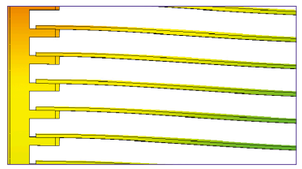
Analysis of a complex assembly model using the automatic contact definition function



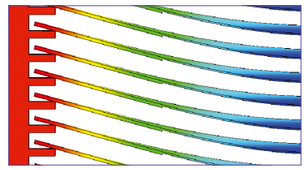
Nonlinear contact analysis of a car's Hood Latch



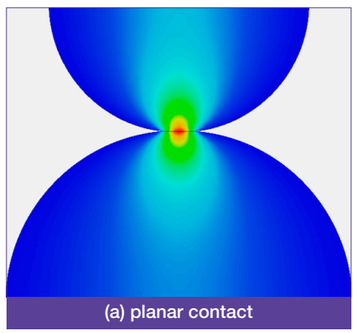
Connector fastening analysis using nonlinear contact



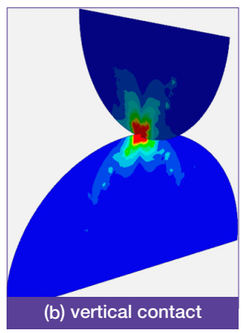
Linear contact (single-body motion)



Nonlinear rough contact (separation)



(a) planar contact



(b) vertical contact

3D line-line contact case : Hertz contact

Contact Analysis

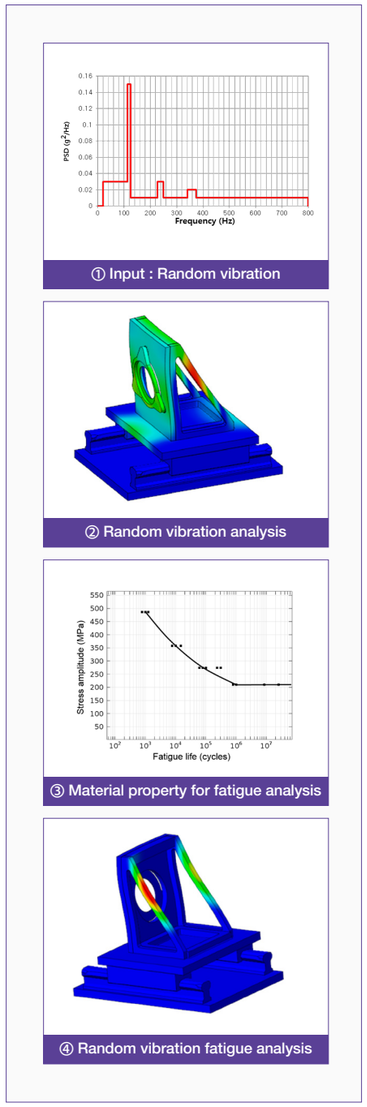
NFX STR uses the latest contact analysis function to analyze complex assembly models and nonlinear contact motions. Contact surfaces are autosearched from which contact conditions are subsequently defined in an assembly model of any complexity.

- Two-dimensional point-line/line-line contact
- Three-dimensional surface-surface, point-surface, single face contacts
- Various methods to define contacts
  - Automatic definition for each analysis case
  - Contact definition wizard, manual definition
- Contact behaviors suitable for practical work
  - Single-body motion, sliding, general and rough contacts, interpolation link
- Coefficient of friction, modulus of rigidity, possible to define shell thickness to simulate contact on both sides of shells
- Various results including contact force and contact stress
- Heat contact to simulate heat conduction between discontinuous parts

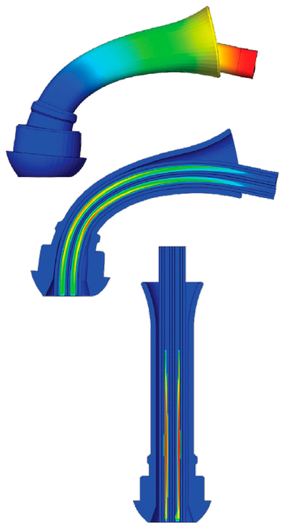
ANALYSIS FUNCTION

Fatigue Analysis

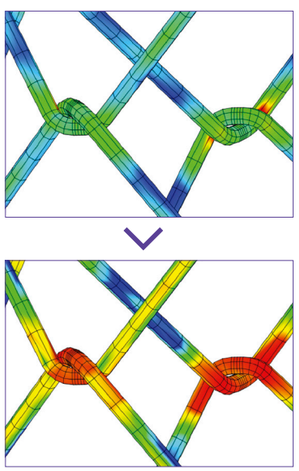
NFX STR can conveniently examine fatigue and durability using an independent post-processing function. Fatigue analysis can be conveniently performed with only minimally required input data. The structural analysis domain can now extend from traditional strength checks to durability checks.



Random vibration analysis
and random vibration fatigue analysis process



Deformation / strength
and service life analysis of cord assembly

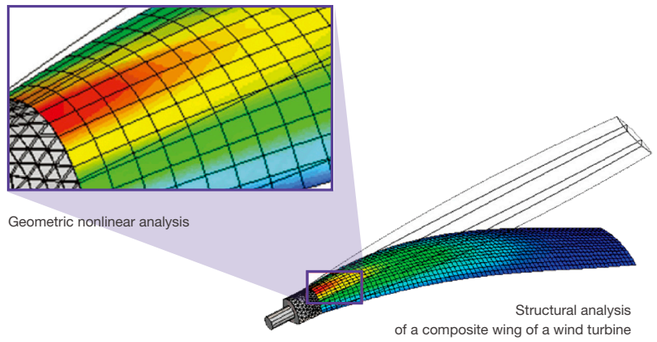


Static analysis using automatic contact
function and examination of fatigue life of a
medical stent using the static analysis results

- Fatigue analysis in time domain
  (fatigue analysis by timedependent load and stress history)
- Damage level, fatigue life results
- Fatigue analysis according to random vibration analysis
- Analysis objects designated
  (boundary, global, user-defined, etc.)
- Rainflow counting, mean stress correction options
- Selection of evaluation stress
  (Signed von-Mises, absolute maximum principal stress)
- Linear/multi-linear S-N curve

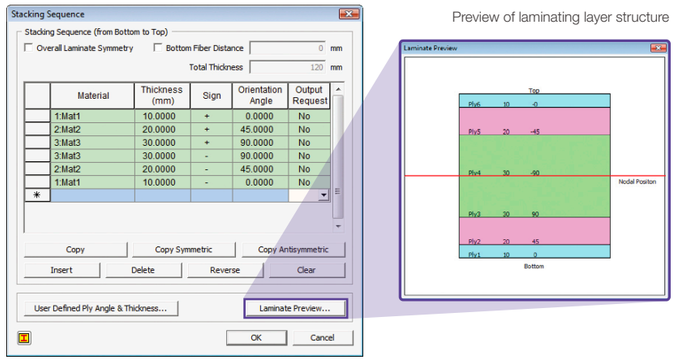
Composite Material Analysis

NFX STR can check 2D and 3D composite material elements together with an intuitive GUI for defining composite materials.



Geometric nonlinear analysis

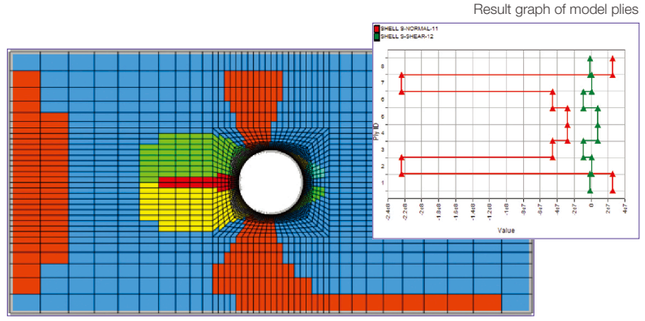
Structural analysis
of a composite wing of a wind turbine



Preview of laminating layer structure

Intuitive GUI for defining a laminated layer structure
(compatible with MS-Excel)

Result graph of model plies



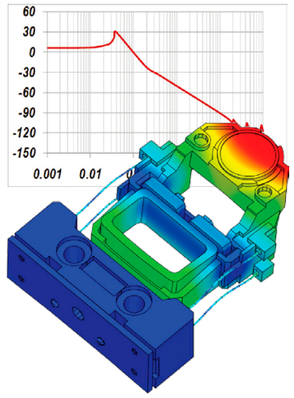
Ply maximum/minimum results (Contour, Iso-line)

- Failure theory
  - Hill, Hoffman, Tsai-Wu, maximum stress, maximum strain, NASA LaRC02
- Failure index
  - Failure Index, FE Failure Index, Strength Ratio
- 3D composite material solids and nonlinear materials supported
- Global Ply ID supported and material property matrices (A, B and D) calculated Top / bottom fiber results per ply produced
- Various ways to define material directions
  (angle, coordinate system, vector, etc.)

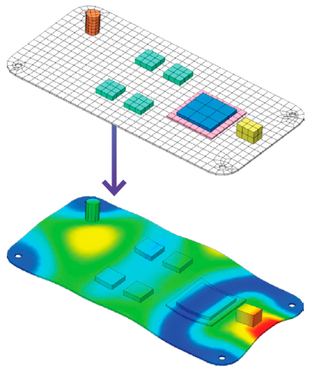
ANALYSIS FUNCTION

Linear Dynamic Analysis

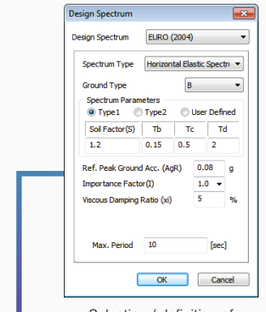
NFX STR can perform practically the most excellent and reliable dynamic analysis. Both direct integration and modal methods are provided with reliability and efficiency.



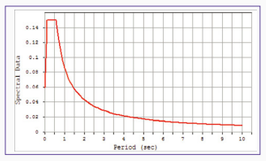
Response analysis of DVDROM due to magnetic force
(frequency response analysis)



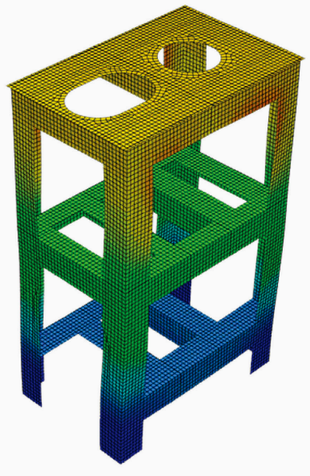
Random vibration analysis of PCB
(RMS results)



Selection / definition of design spectrum



Automatic generation of response spectrum seismic

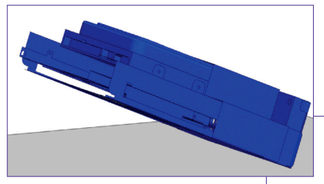


Seismic analysis

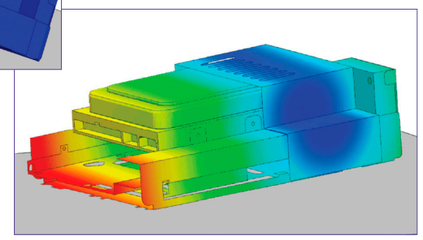
- Direct method and mode method
  - Transient response analysis
  - Frequency response analysis
  - Random vibration analysis
  - Response spectrum analysis
  - Enforced motion analysis
  - Import results of modal analysis
- Conversion function from static to dynamic loads
  - Analysis function considering various load conditions
- Automatic time increments
- Analysis function considering pre-stress
- Various damping effects
  - Modal, structural, material, Rayleigh, frequency-dependent
- Design spectrum database implemented

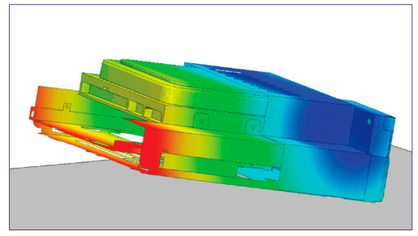
Explicit Dynamic Analysis

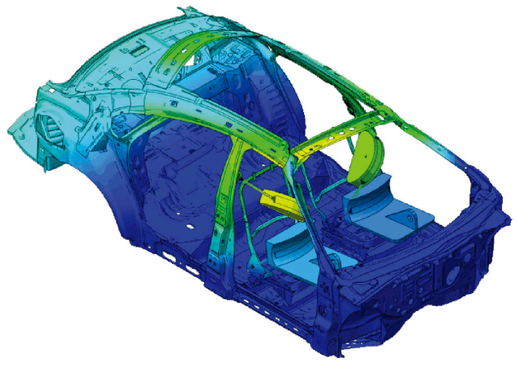
By using implicit and explicit time integration, NFX STR effectively calculates complex material, geometric and contact nonlinear phenomena of large scale assembly models. Accurate analysis can be conveniently carried out using various element types including hexahedron elements, pyramid elements and higher-order tetrahedron elements.



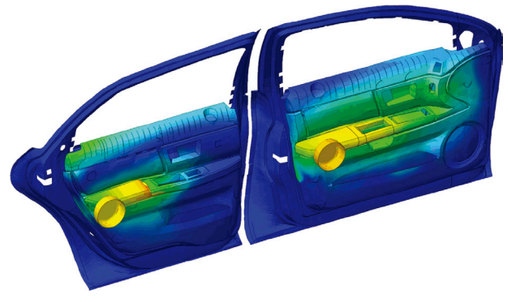
Hard disk drop test







Seat belt anchorage analysis



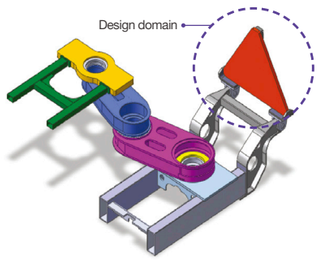
Door trim impact analysis

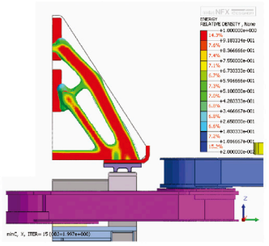
- Diverse nonlinearity
  - Material nonlinearity: elastoplastic, hyperelastic (Mooney-Rivlin, Neo-Hookean, Polymoial, Ogden, Blatz-Ko, etc.) models
  - Geometric nonlinearity : large displacement, large rotation, follower force
  - Contact nonlinearity : various contact behaviors considering three-dimensional contact and friction
- Mass scale
  (Only applied explicit time integration analsysi)
  - Scaling by individual element groups
  - Time step based mass adjustment
- Automatic calculation of safe time step by elements
- Checking the status of convergence and results in the interim steps during analysis
- Restart function using subcases and parallel processing function using multi-cores

ANALYSIS FUNCTION

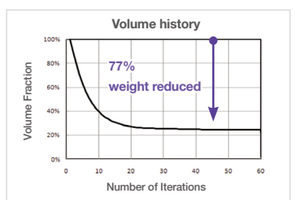
Topology Optimization Design

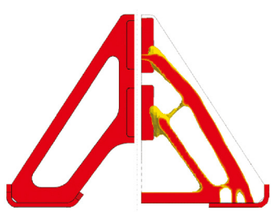
NFX STR provides practical topology optimization analysis considering static/dynamic analyses and manufacturing processes.
By linking linear static, modal and frequency response analyses, all of which are widely used in practice, optimization analysis is performed considering structural safety and economy.



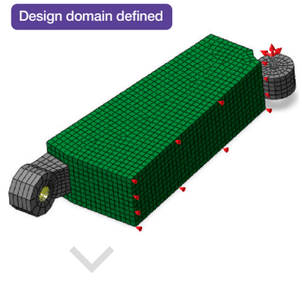


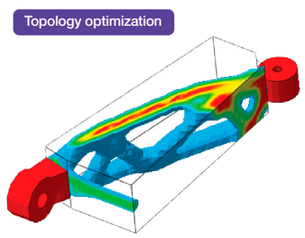
Conceptual design using topology optimization
(linear static analysis, weight reduction through minimizing volume)

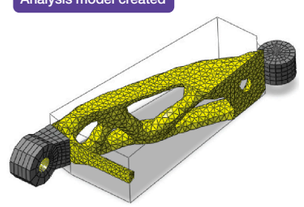


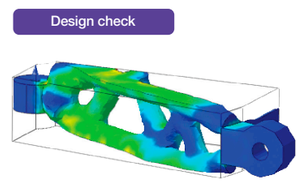


Review against original design







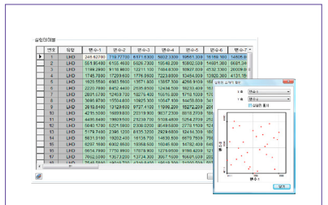


Process of using topology optimization design

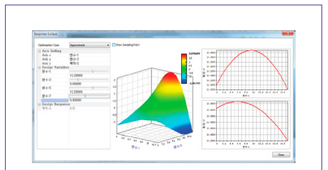
- Optimization analysis function linked with static and dynamic analyses
  - Linear static analysis
  - Modal analysis
  - Frequency response analysis
- Analysis function considering conditions of manufacturing processes
  - Setting design limit/constraint conditions such as stress, displacement, volume, draw direction and symmetrical conditions
- Simultaneous optimization analysis considering various operation and load conditions
- Automatic regeneration of analysis model without separate CAD work and mesh smoothing function
- Other practical convenience functions
  - Mode trace, definition of design/non-design domains, automatic initial value setup

Size Optimization

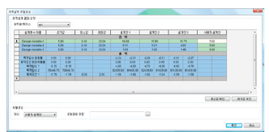
NFX STR provides size optimization analysis based on estimation and verification of each material and property's influence.
NFX STR can determine an optimal material/property composition to minimize stress, volume or weight of the designed model.



Sampling point output/analysis

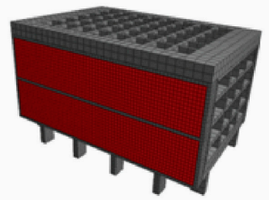


Approximate model generation

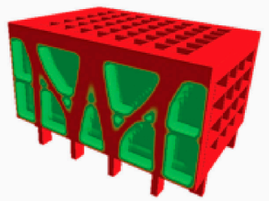


Capacity estimation/verification & Optimized model generation

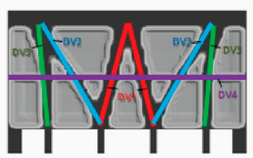
Approximate model-based size optimization design process



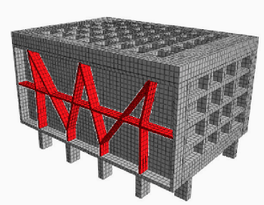
Assign Design Domainoptimiz



Topology Optimization



Approximate model Generation



Size Optimization

Size optimal design using topology optimization

- Size optimization for all types of thermal/structural analysis
- Property and material design variables
  - Intuitive assignment of variables for size optimization
  - Section size and thickness, composite material lamination thicknesses and angles, spring stiffness, damping, mass, modulus of elasticity, etc.
- Design Sampling
  - Various Methods (FFD, CCD, OA, LHD) & 1D Parameter Study
  - Correlation between Design variables & Analysis Response
- Size optimization design based on approximate models
  - Approximate modeling techniques (Kriging model, Polynomial Regression model)
  - 2D/3D Graphic tool for approximate model analysis
  - Optimization design estimation and analysis result verification
  - Automatic optimized model generation

MIDAS NFX Specification

Details		
Structural	Linear Static Analysis	Linear Static Analysis
		Modal Analysis
		Buckling Analysis
		Composite Materials Analysis
	Nonlinear Static Analysis	Nonlinear Material Analysis
		Nonlinear Geometry Analysis
		Nonlinear Contact Analysis
	Heat Analysis	Heat Transfer Analysis
		Heat Stress Analysis
		Joule Heating Analysis
	Linear Dynamic Analysis	Transient Response Analysis
		Response Spectrum Analysis
		Frequency Response Analysis
		Random Vibration Analysis
	Nonlinear Dynamic Analysis	Explicit Dynamic Analysis
		Implicit Dynamic Analysis
	Optimization	Topology Optimization Analysis
		Size Optimization Analysis
	Fatigue Analysis	**S-N curve** (Stress-life Method) **/ ε-N curve** (Strain-life Method)
		Thermal Fatigue Analysis
		Ramdom Vibration Fatigue Analysis
CFD	General Fluid Flow Analysis	Steady/Unsteady Fluid Flow Analysis
		Compressible/Incompressible
		14 Turbulence models
		Porous Media
		1-D Pipe Model
		Fan Boundary Condition
		MRF (Moving Reference Frame)
	Heat Transfer Analysis	Conduction/Convection/Radiation
		Conjugate Heat Transfer/1-way FSI
		Joule Heating/PCB Heat Resistance Model
	Mesh Deformation Analysis	Stretchable Mesh
		Sliding Mesh
		Overset Mesh
	Mixture Analysis	Species transport
	Multi-phase Analysis	Level Set
		Wave Elevation Analysis
		Discrete Phase Model
FSI (Fluid-Structual interaction)		Thermal 1-way coupled Analysis
		Structural 1-way coupled Analysis
		Structural 2-way coupled Analysis

MIDAS NFX STR

MIDAS

MIDAS
NFX STR

MIDAS

nfx.midasuser.com