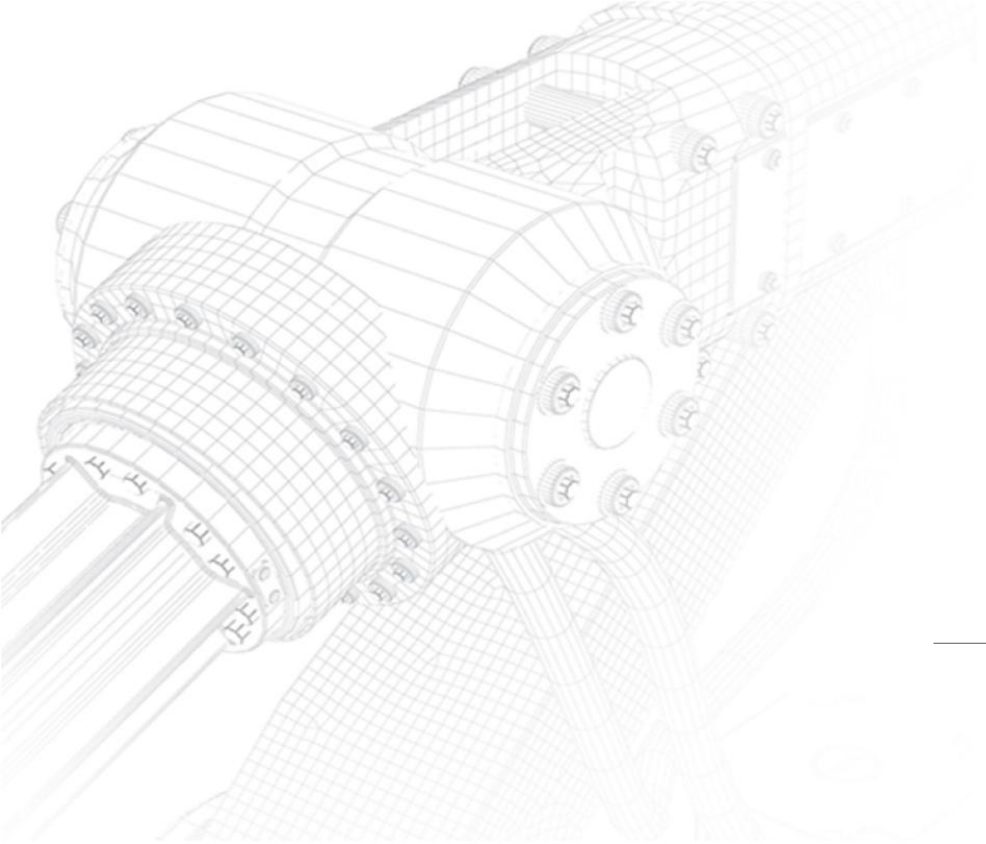
An insight into NFX 2012

Total Analysis Solutions for Multi-disciplinary Optimum Design

MIDAS

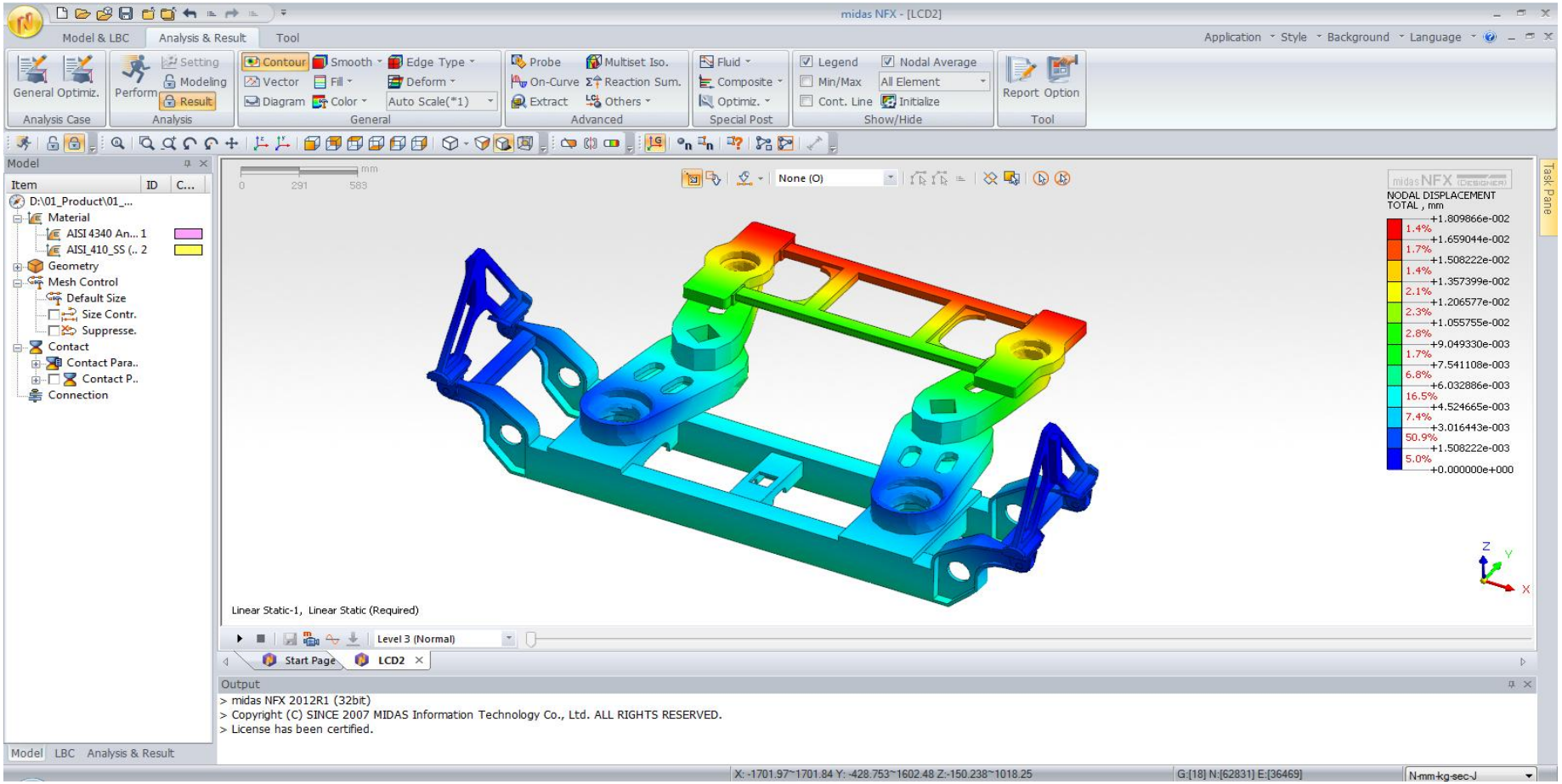


midas NFX

Total Solutions for True Analysis-driven Design

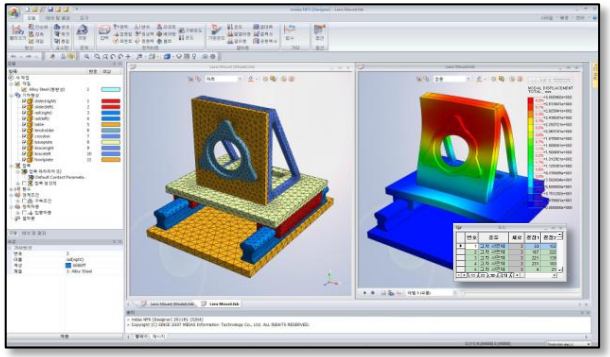
Contents

midasNFX provides multi-disciplinary CAE solutions totally integrated into one unique work environment



Simplified framework focused on intuitive user interface

midas NFX (Beginner Mode)



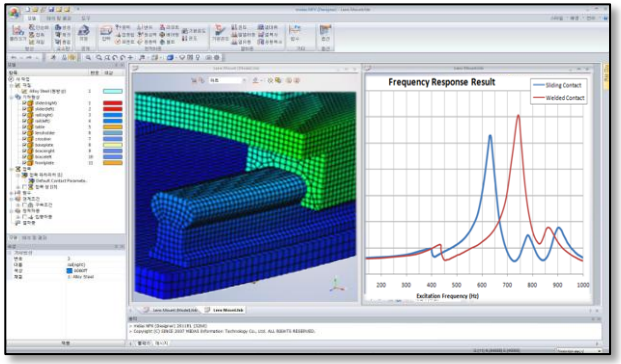
Simplified Mode

(Basic Operations on CAD Model)

- CAD model-based work environment for designers
- Fast and simple way to perform simulation design with various automated functions and minimal input requirement

Analysis Mode can be easily changed

midas NFX (Expert Mode)



Advanced Full Mode

(Precise Modeling and Analysis Control)

- From CAD Model creation to meshing finite elements for all types of mechanical analyses. Expert Mode is designed for simulation engineers and mechanical analysis specialists.
- With complex geometric modeling and various mesh generators, accurate modeling and analysis, along with extensive result analysis functions, are provided.

+

NFX Solvers

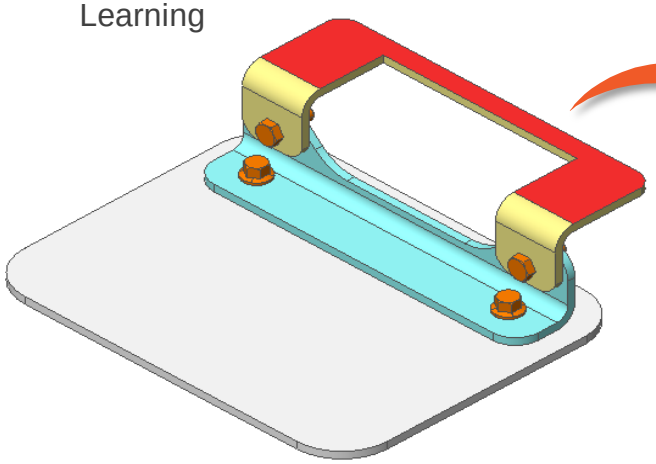
Linear Static (with contact)
Heat transfer/Thermal Stress (Steady/Transient)
Multi Body System (Rigid/Flexible Body)

Nonlinear Static (Material/Geometry/Contact)
Linear/Nonlinear Dynamic (Explicit included)
CFD (Thermal/Fluid, Moving Mesh)

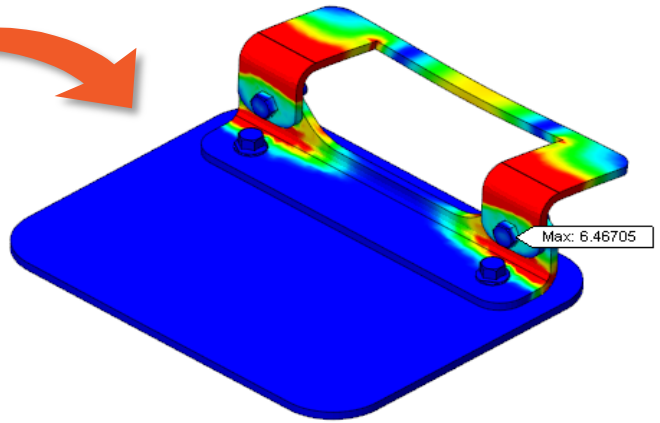
Modal/Buckling (Prestress)
Fatigue (S-N/e-N)
High Perf. Parallel Solvers (32/64 Bits)

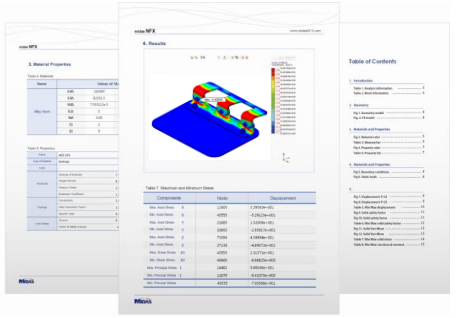
Effective analysis, standardized result evaluation & suggested design improvement

1 Application in practice with Minimal Learning

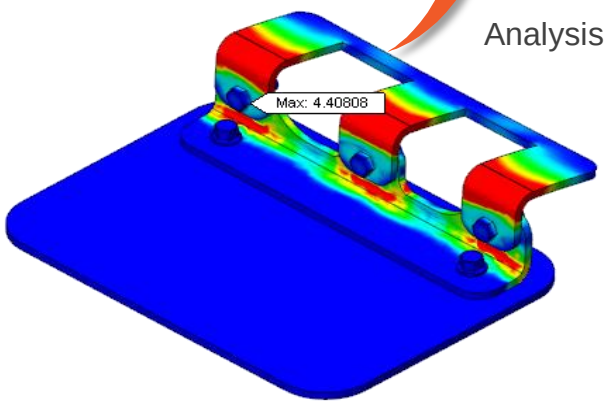


2 Fast Analysis and Results Evaluation with Minimum Operations

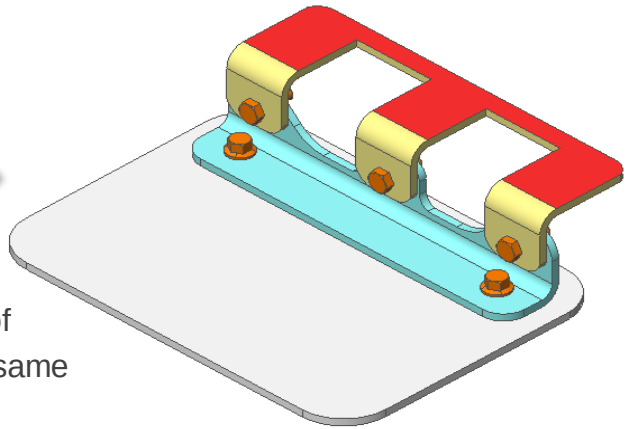




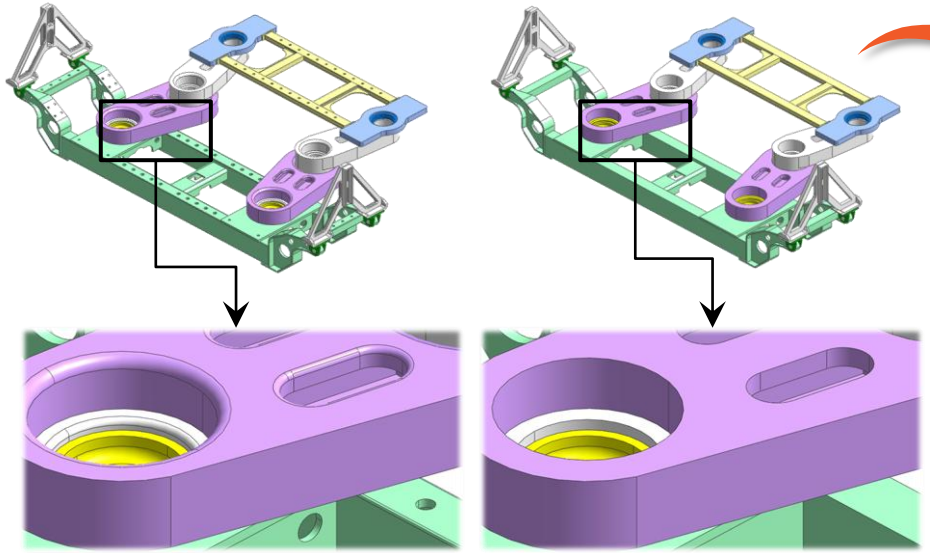
4 Auto generation of Customized Analysis Report



3 Immediate re-analysis in case of change in CAD model with the same data transferred from the step 1



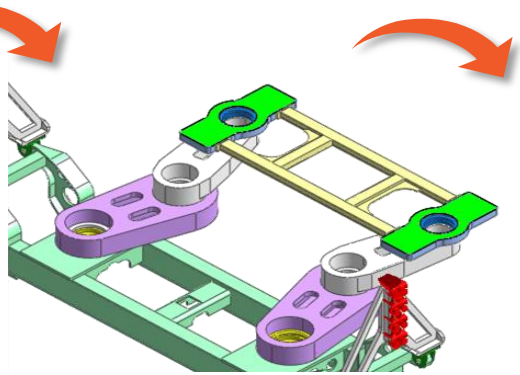
Intuitive Workflow with minimum mouse operation



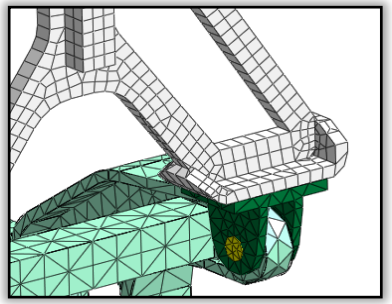
CAD Model Import

Model after clean-up and contact creation

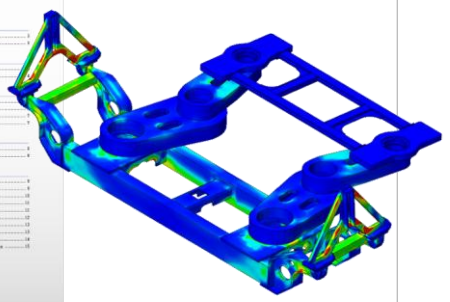
- Automatic Contact Generation
- Removal of useless Holes and Fillets by Cleanup



Loads and Boundary Conditions Definition



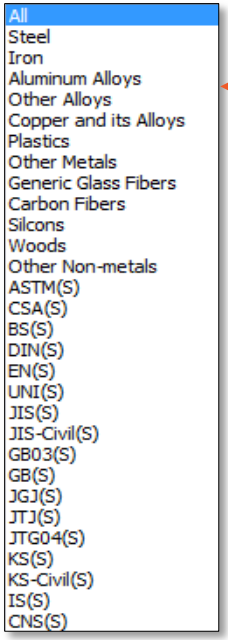
Auto Meshing



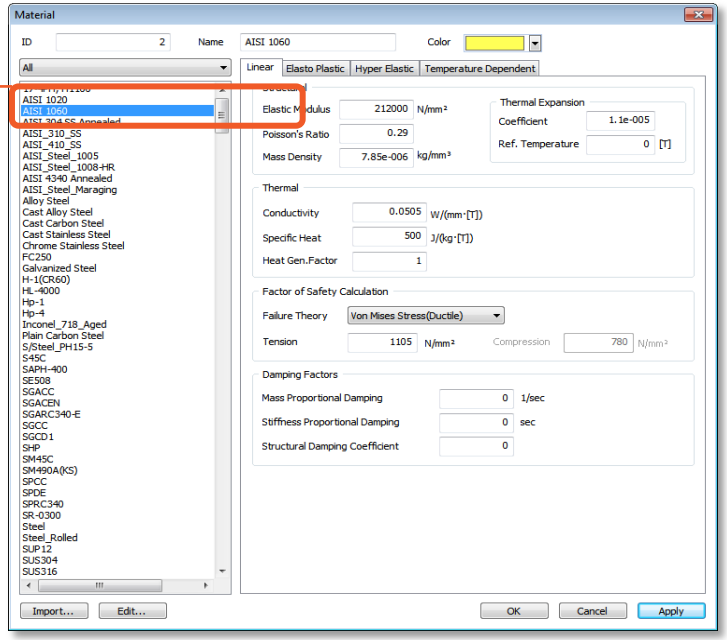
Checking Analysis Results and Customized Report Generation

All operations can be performed simply and conveniently in a few mouse clicks

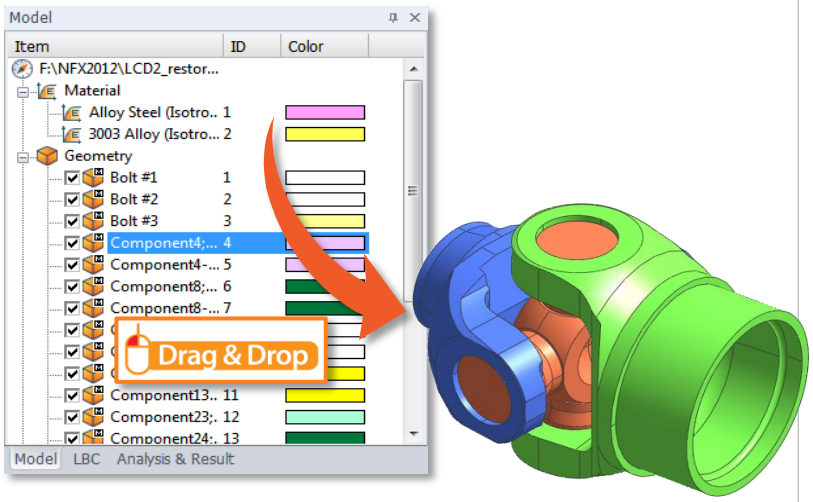
Large Material Database



Material properties DB

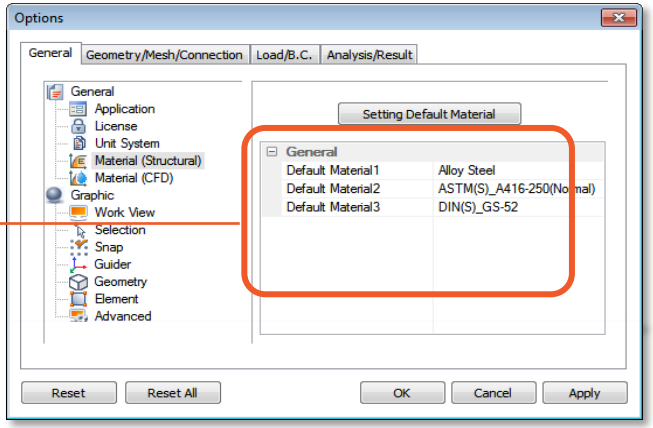


Properties for Structural/Fatigue/Thermal Analysis
(Safety factors included)

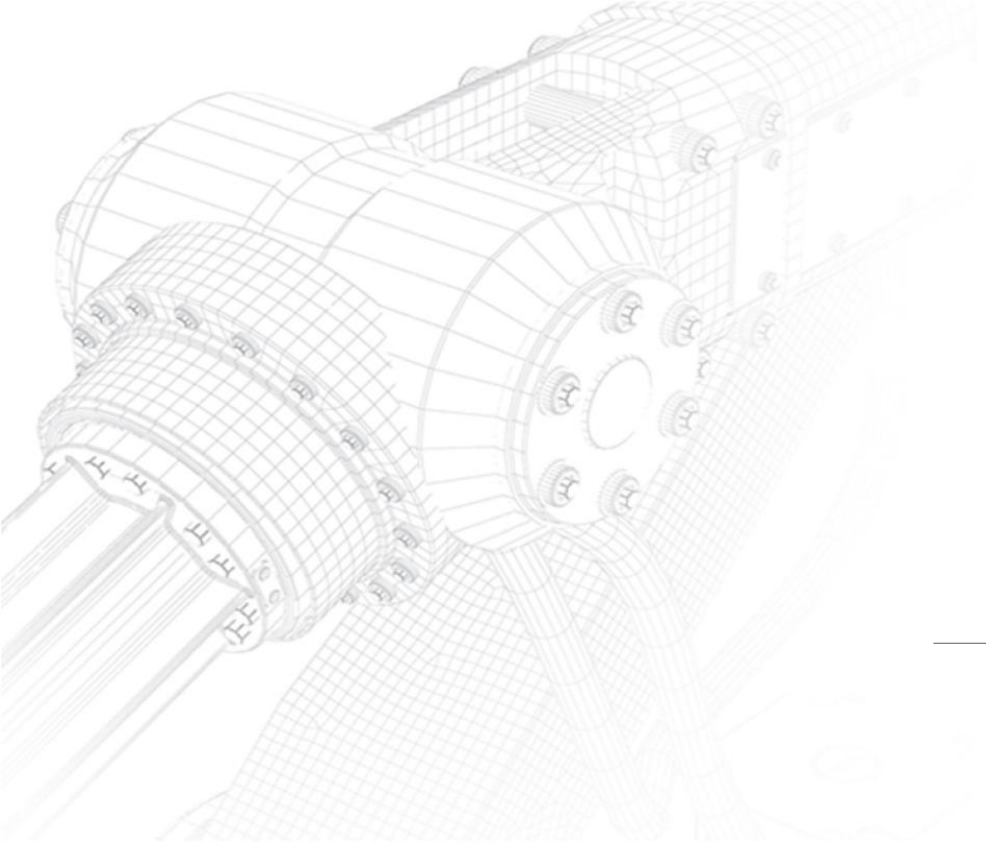


Assign Materials with Drag&Drop Feature

Upon registration of Basic Materials,
the Material database is
automatically added to New Project.



Database of More than 1000 Materials



midas NFX

Total Solutions for True Analysis-driven Design

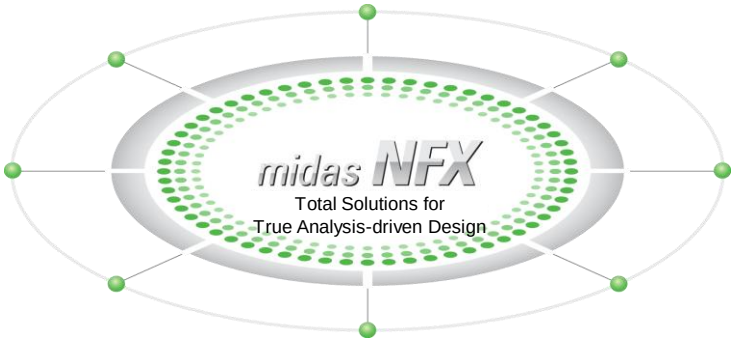
Contents

Multidisciplinary integrated structural/thermal/CFD analysis solutions for optimal design

Linear Static Analysis
(Linear Contacts, Boundary/Joints)

Modal/Buckling Analysis
(Prestress, Rigid/Joints)

Heat Transfer/Thermal Stress Analysis
(Steady State/Transient)



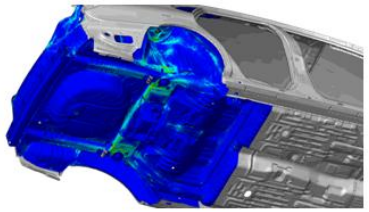
Nonlinear Static/Quasi-Static Analysis
(Material/Geometry/Contacts)

Fatigue Analysis
Composite Materials Analysis

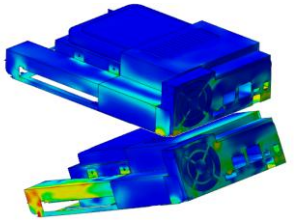
Dynamic Analysis
(Time, Frequency Response, Random, Nonlinear Explicit)
(Prestress, Rigid/Joints)

CFD Analysis
(Thermal/Fluid, Moving Mesh)

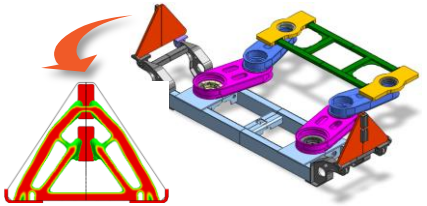
32/64bits
High Performance Parallel Solvers



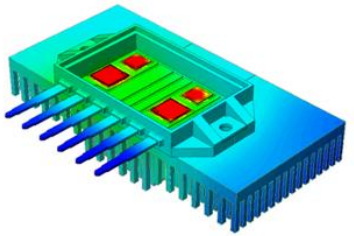
Evaluation of Strength/Durability of a Car Body
(Linear Static/ Fatigue)



Evaluation of Drop Impact of Hard Disk
(Nonlinear Dynamic, Contact)



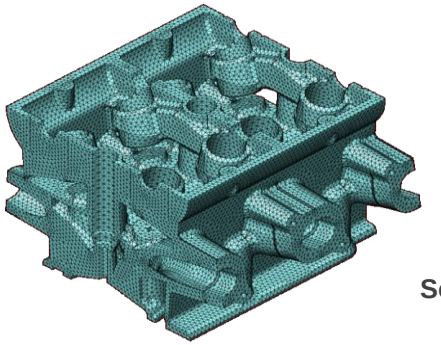
Optimal Design of a Connection Frame
(Topology Optimization)

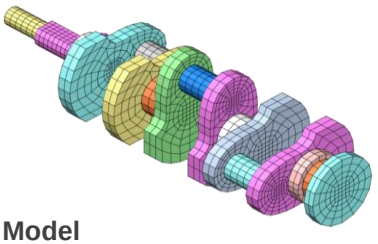


Evaluation of Thermal Deformation/Stress of a Heat Sink
(Linear Heat Transfer / Thermal Stress)

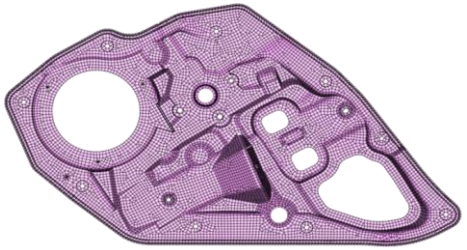
Extensive Element Library

- 3D Element
  - General Solid
    Tetrahedron, Pentahedron, **Pyramid**, hexahedron
  - Composite Solid (Pentahedron, Hexahedron)
- 2D Element
  - Shell, Plane Stress, Plane Strain, Axisymmetric
  - Composite Shell, Surface element
- 1D Element
  - Beam, Bar (Truss), Pipe
- Others
  - Spring, Mass, Damper, Fastener
  - Rigid Link, Interpolation Link, Bush

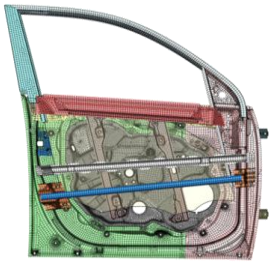




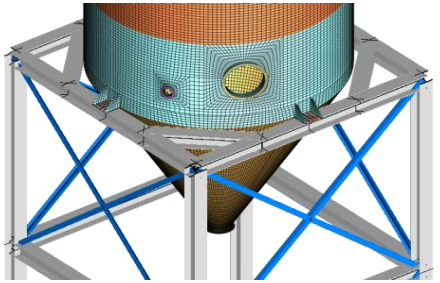
Solid Model



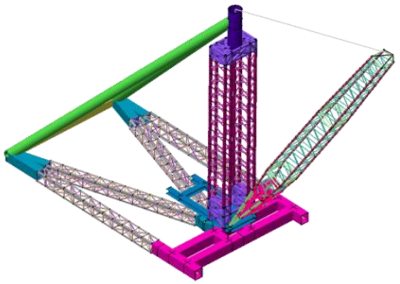
Shell Model



Hybrid Model (Shell/Solid/Mass/Link)



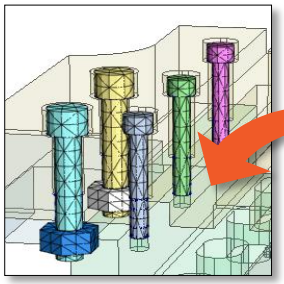
Hybrid Model (Solid/Shell/Frame)



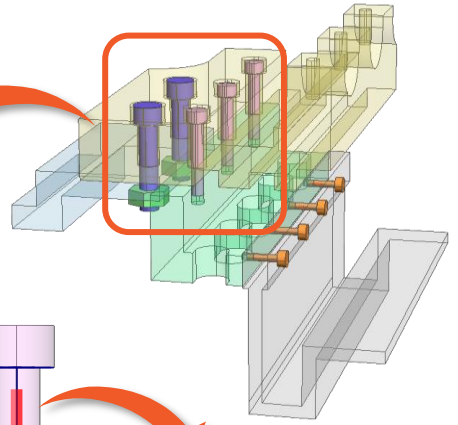
Frame Model

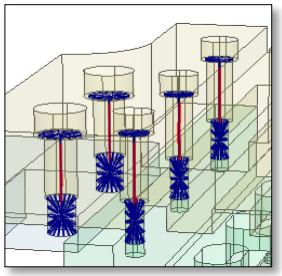
Various elements/loads/boundary conditions frequently used in practice

- **Link Elements /Modeling**
  Rigid Link, Bolt Link, Spring, Fastener
- **Loads**
  Self Weight, Concentrated Load, Moment, Pressure, Temperature, Displacement, Centrifugal Force, **Remote Load, Bolt Load, Bearing Load**
- **Boundary Conditions**
  Pinned Constraint, Rotational Constraint, Fixed, DOF Constraint, Symmetrical Constraint, Reference Shape based Constraint (Cylinder, etc.)

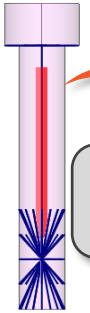


Bolt Model+ Contact

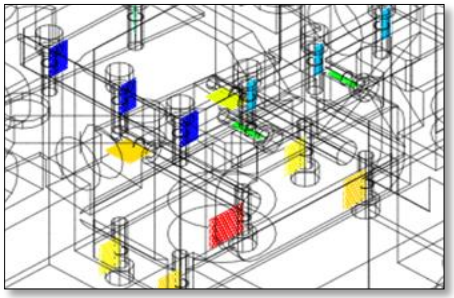




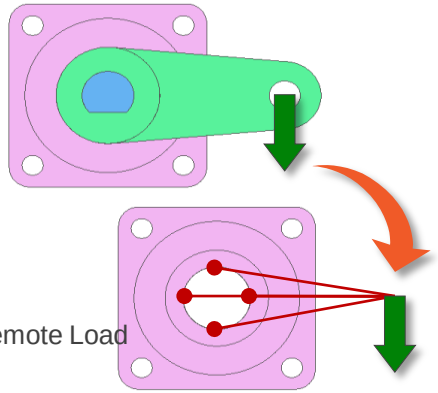
Bolt Links



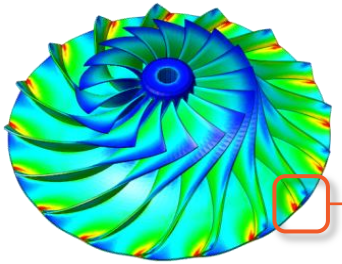
- Rigid Link+Beam Element
- **Bolt Load**

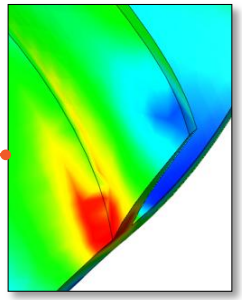


Check fastening forces from Axial Forces/Stresses in Bolt inks



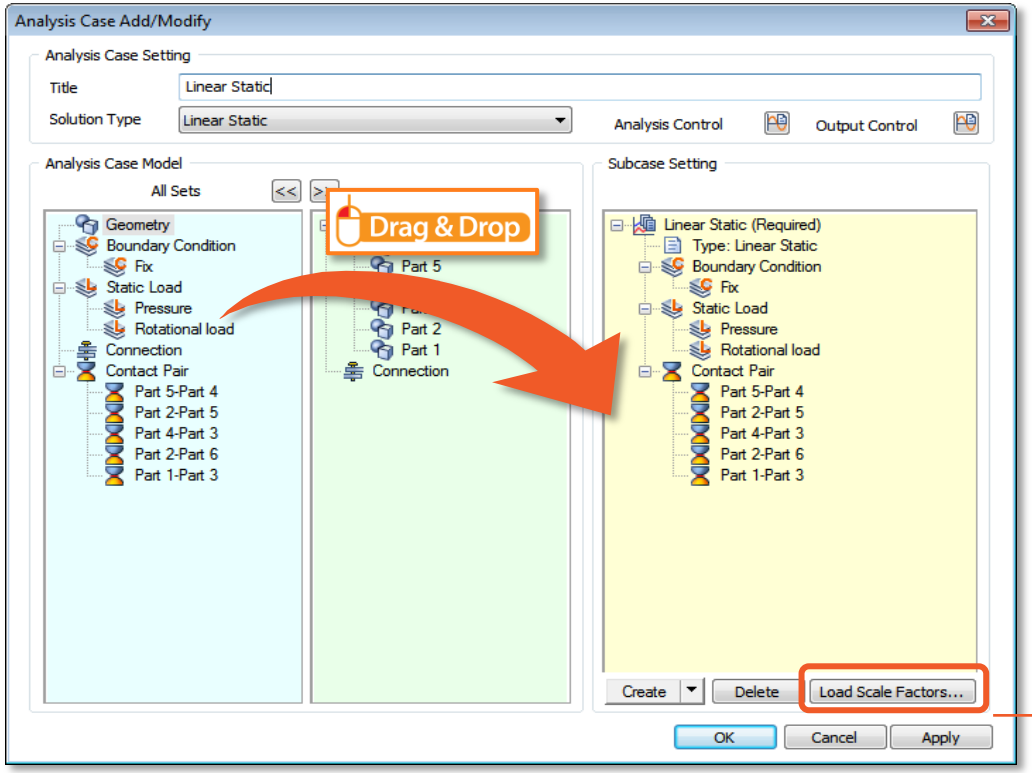
Remote Load





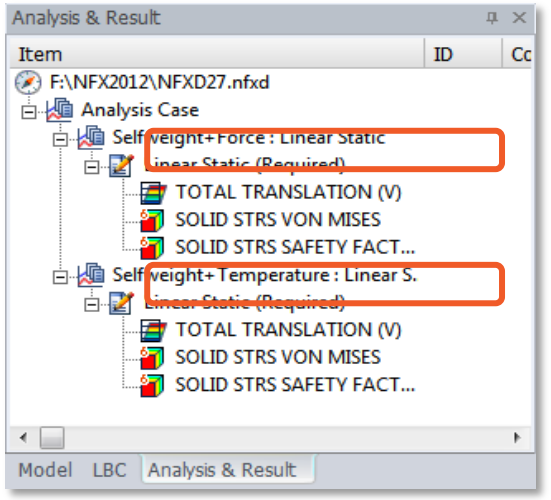
Deformation, Strength & Vibration of Impeller (Centrifugal, Prestress mode)

Subcases to analyze results of individual load cases and combinations



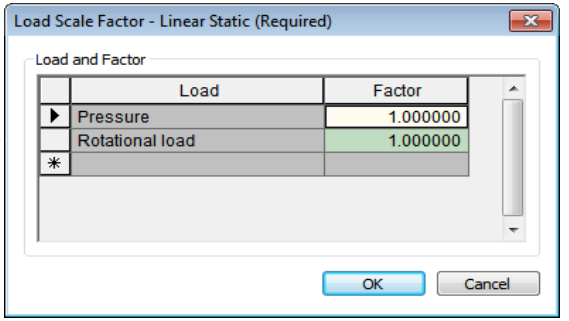
Assign Load/Boundary conditions to Subcases simply by Drag & Drop

(Possible to define multiple Subcases using a single Analysis Case)



Separate Results of Defined Subcases

➡ In case of unacceptable results of the total load, causes are identified by analysis of results for individual loads.

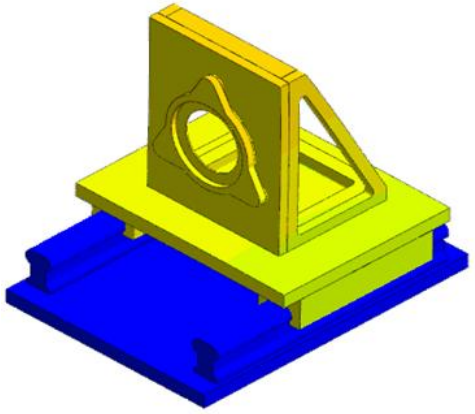


Combination Factors (Total Force=$\Sigma a_i P_i$)

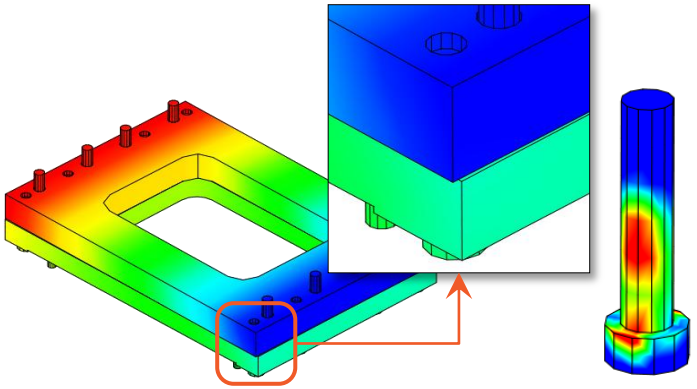
Subcases & Results Combinations of various load conditions

Fast and accurate analysis using high-performance elements, parallel solvers & contact

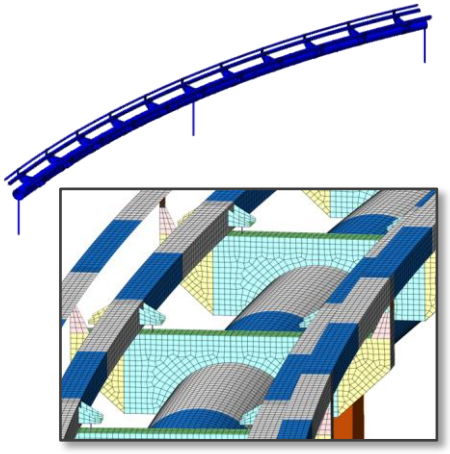
- Linear Static: Displacement /Stress /Safety Factor results, Thermal Displacement/Thermal Stress, Prestress, Subcases and Result Combinations of each loading case, Evaluation for suitability of Mesh density
- Modal /Buckling: Natural Frequency/ Mode Shape / Modal Participation Factor/ Effective Mass Results, Prestress, Assignment of Range of Eigenvalue calculation, Missing Eigenvalue check (Sturm Sequence)
- 32 / 64bit high-performance Parallel Solvers: Multi-Frontal, AMG, Block Lanczos
- Linear Contacts: Welded contact, Sliding contact, Interpolation link



Simulation of lens mount movement along rails
(Linear sliding contact)



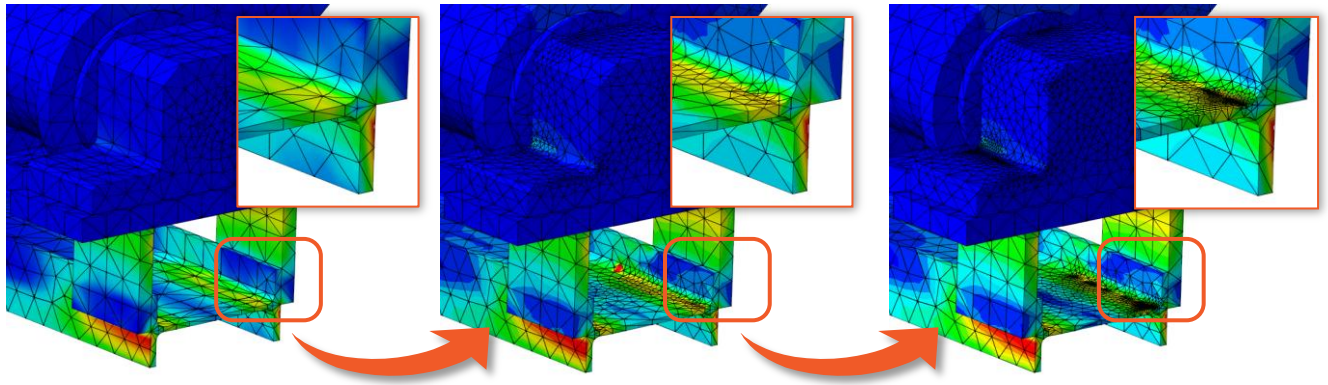
Relative deformation and bolt stresses
using linear sliding contact



Modal analysis of a track rail
(4th mode, Shell/Frame model)

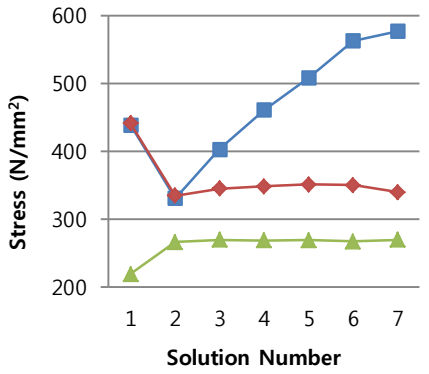
Adaptive mesh analysis for reliable analysis results

- Applicable to Linear static, Steady state heat transfer analysis
- Mesh refinement using the error criteria by Zienkiewicz-Zhu
- **Mesh convergence criteria:** Stress/Strain, Flux/Rate of change in temperature gradient
- **Auto-detection of singularities to prevent excessively fine mesh**
- **Local areas of interest can be selected**: curve, surface, solid



Initial Mesh in Region of interest

Mesh Refinement by Adaptive Analysis



Change in maximum stress in local area

Convergence-based reliable meshing that even beginners can use

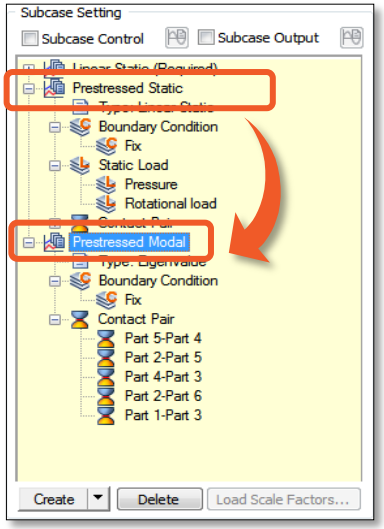
Modal Analysis considering prestress & relative motion between parts of an assembly

Prestress Modal Analysis

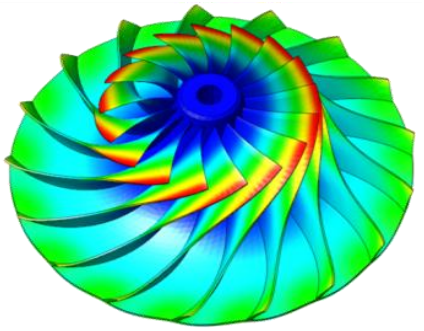
- Consecutive Analysis of two Subcases
  Analysis of prestress conditions reflected in the stiffness of the structure followed by eigenvalue analysis

Linear Contact supported

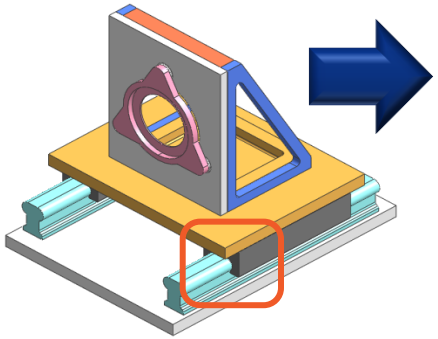
- Modal Analysis reflecting the relative motion between parts
- Welded contact, Sliding, Interpolation link



Prestress Modal Analysis composed of 2 Subcases

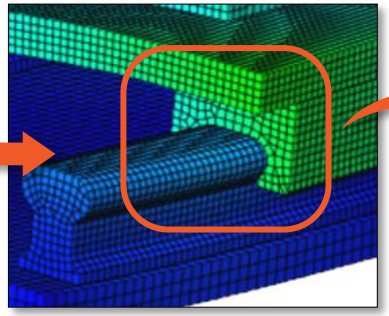


Modal Analysis of impeller for rotational condition
(Prestress Mode, Centrifugal Force)

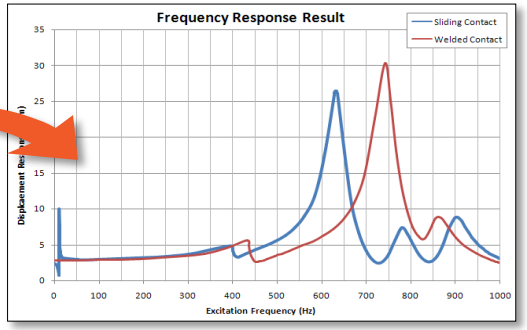


Contact Behavior Comparison
(Behavior of Rail Part)

General Welded Contact Behavior



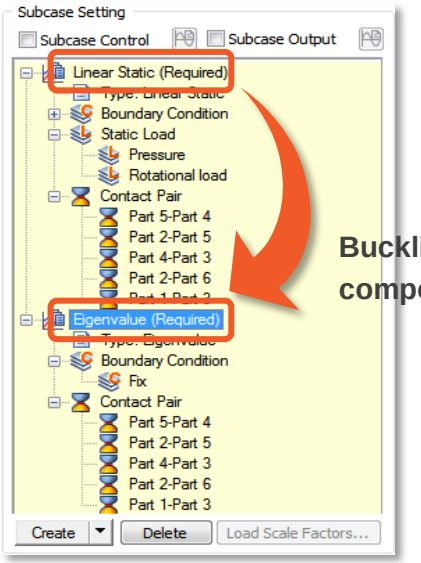
Sliding Contact
(Relative Motion)



Frequency Response Analysis Results
(Difference in maximum frequency responses)

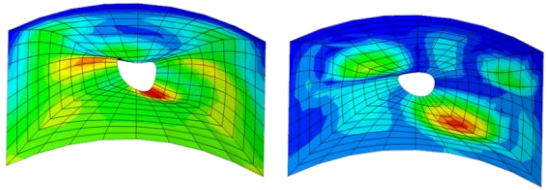
Buckling Analysis considering relative motion between parts of an assembly

- **Parallel Block Lanczos Solver**
- Applicable to all elements including composite materials
- Analysis Results
  - Eigenvalues (factor), Mode shapes
  - Stress, Strain, Strain energy, etc.
- Possible to define the range of eigenvalue calculation
- Missing Eigenvalues Check (Sturm Sequence)
- **Linear Contact supported**

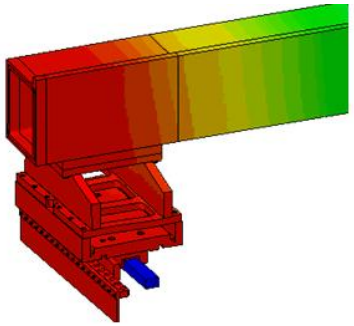


Material and geometric stiffness calculated from linear static analysis is reflected in the stiffness of the structure followed by eigenvalue analysis

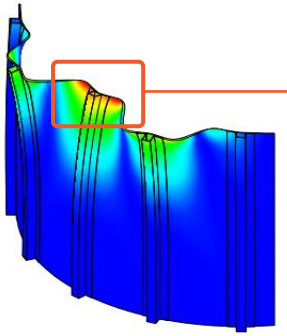
Buckling Analysis composed of 2 Subcases

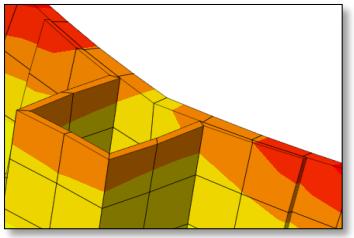


Buckled Shape of a composite panel containing an internal hole
(Composite shell elements used)



Analysis of a complex assembly model
(Linear Contact)



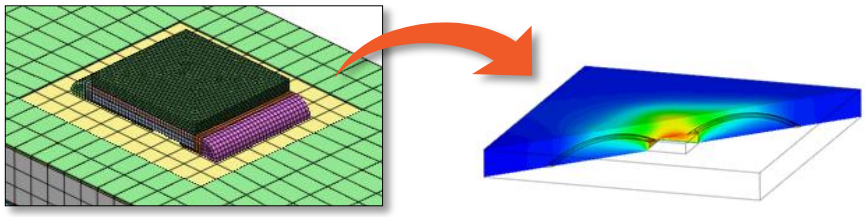


Discontinuous Hexahedral Mesh
(Continue result)

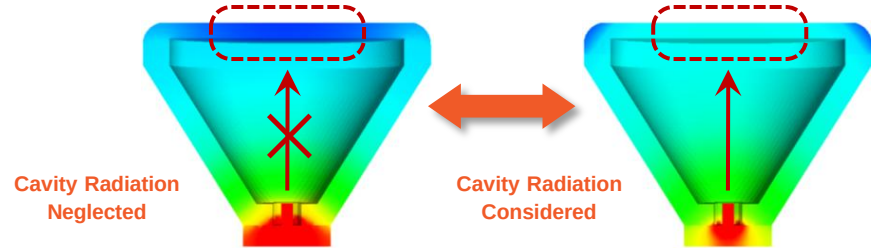
Buckling Analysis & Buckled Shape under the condition of welded contact

Heat Transfer/thermal Stress Analysis accomplished in only one Analysis

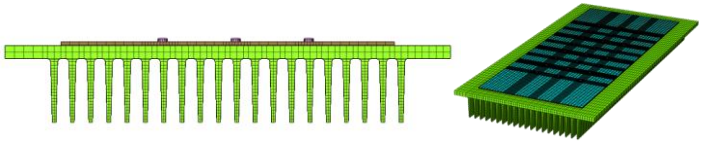
- Steady State /Transient State Heat Transfer Analysis (Temperature-dependent material model, Nonlinear Radiation)
- Heat Generation, Conduction, Convection, Radiation (Cavity Radiation: Closed/Open conditions, Radiation Shape factor automatically calculated)
- Thermal Contact supported (Conduction between discontinuous parts)
- Independent Thermal Stress Analysis Case provided (Heat Transfer→ automatic coupling with linear static)



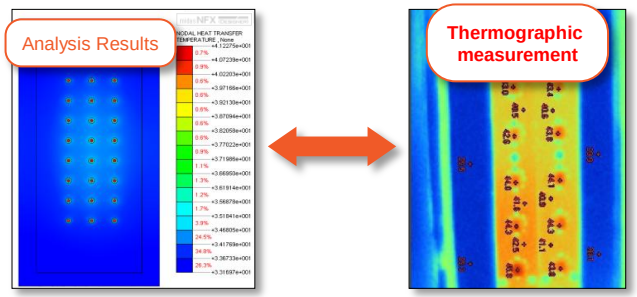
Heat transfer Analysis of BLU
(Application of thermal Contact to Hexahedral/Tetrahedral hybrid mesh)



Heat transfer & Temperature distribution due to heat cavity radiation within an LED Lamp



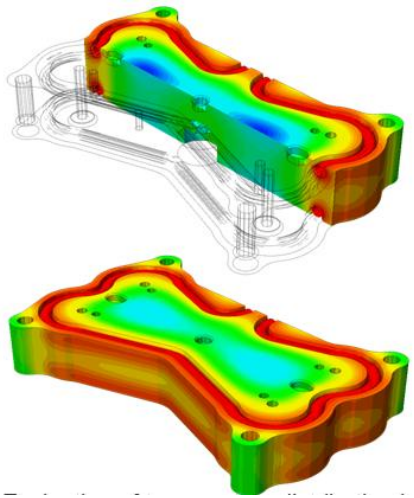
Heat transfer Analysis of LED street light assembly model
(Application of Thermal Contact)



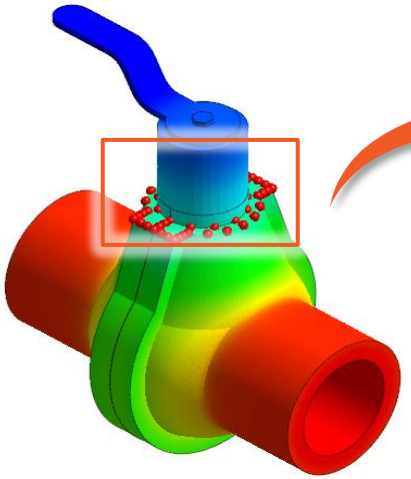
Comparison of Temperature distribution analysis
results and Thermographic measurements (Tolerance Level: 3%)

Effective Transient Heat Transfer analysis using Sensors

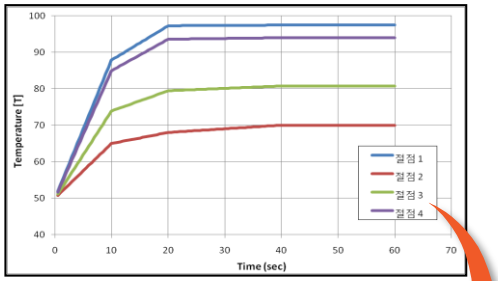
- Analysis automatically terminated based on defined criteria⇨ Reduction in Analysis Time based on imposed special conditions
- Sensor: Maximum/Minimum/Average criteria specified within specific domains
- Temperature based Sensor (below/over reference value)
- Analysis terminated when the rate of change in temperature is less than a specified reference value



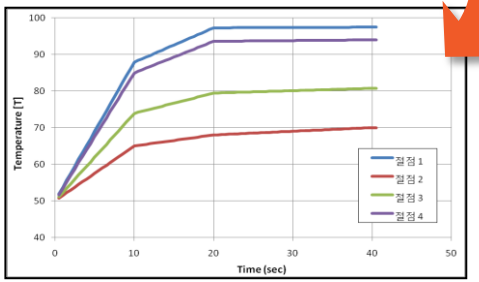
Evaluation of temperature distribution inside a Manifold
(Transient heat transfer analysis)



Transient heat transfer analysis of a ball valve (Sensor used)



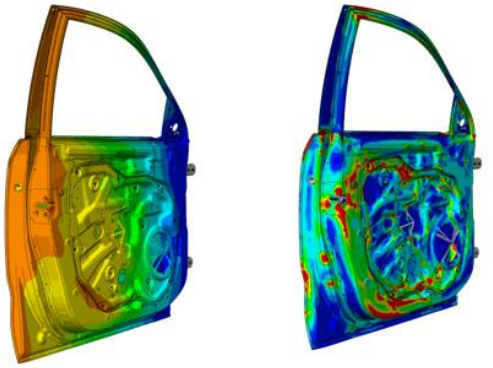
General Analysis condition (60 sec)



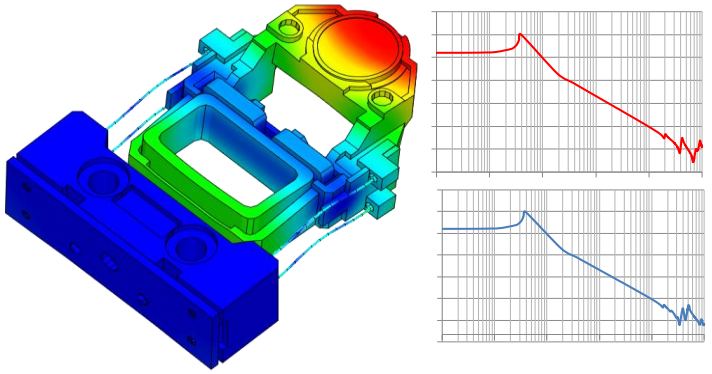
When sensor is used, Analysis is completed in 41 sec

Linear Dynamic Analysis of superior performance and practical applicability

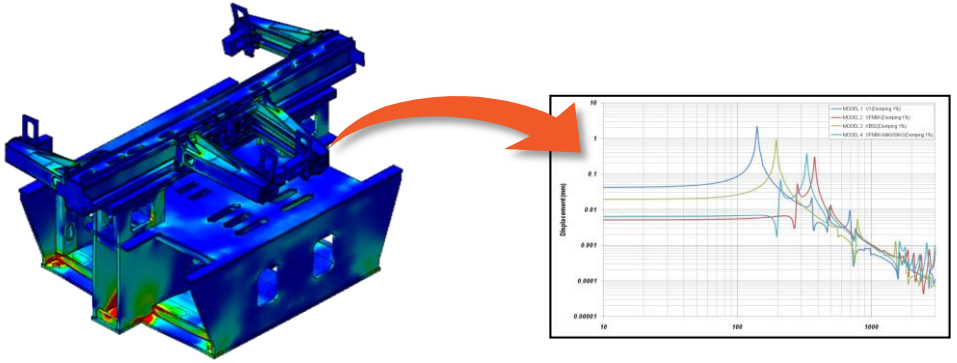
- Linear Dynamic Analysis by Direct and Modal Response methods
  - Transient Response
  - Frequency Response
- Possible to convert Static Load into Dynamic Load⇨ Various loading conditions converted
- Auto Time Step supported
- Random, Prestress conditions considered



Evaluation of stress and behavior for repeated opening and closing of a door (Transient Response Analysis)



Evaluation of frequency response of a DVD-Rom due to electro magnetic force (Frequency Response Analysis)

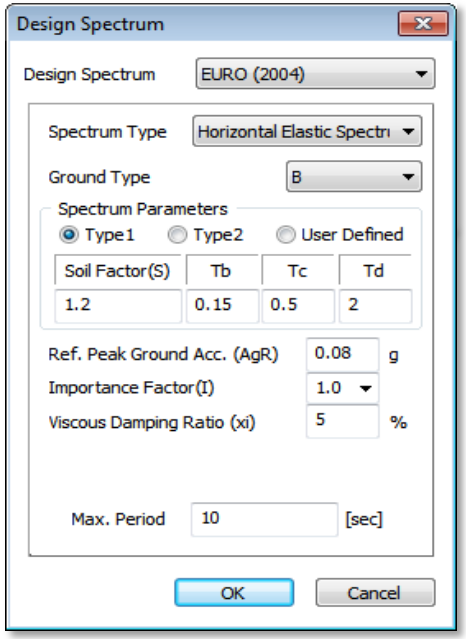


Evaluation of soundness of semiconductor equipment (Frequency Response Analysis)

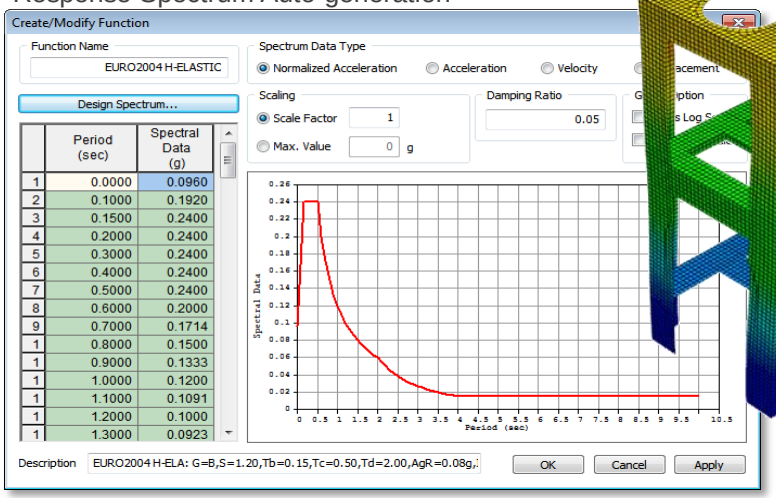
Database of design response spectrums

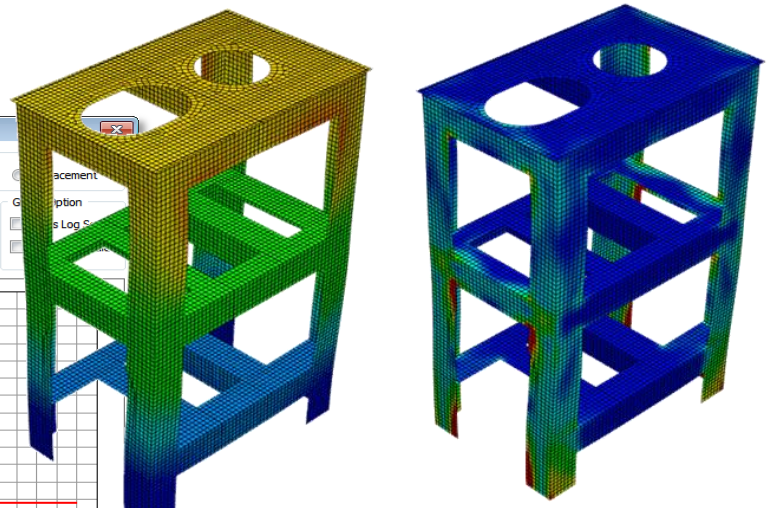
- **Various design spectrum database**
  - Euro code (2004), IBC2000 (ASCE7-98), UBC (1997)
  - Korea, Japan, China (JTJ)

Select/Define Design Spectrum



Response Spectrum Auto-generation

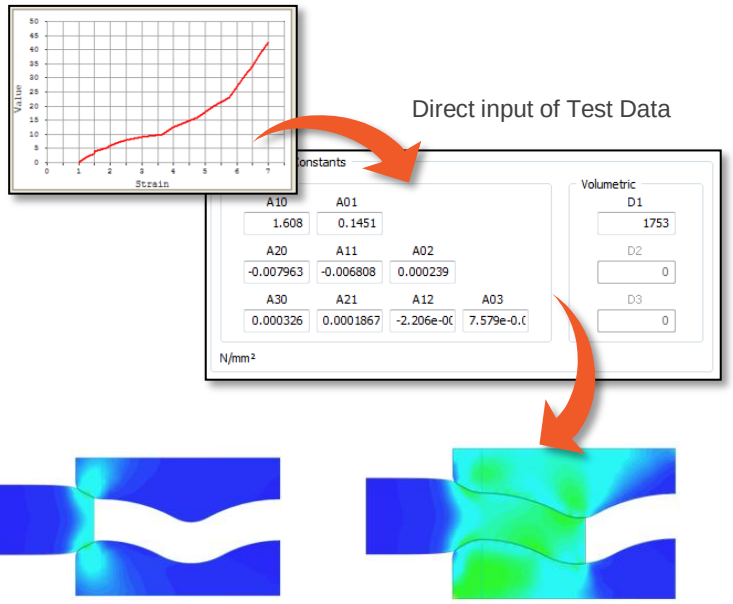




Response Spectrum Analysis

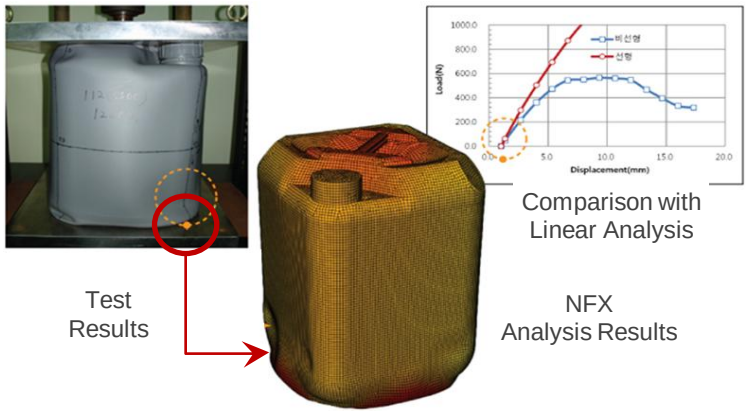
Nonlinear material/geometry/contact with excellent convergence and practical applicability

- Nonlinear Material: **Elasto-plastic model, Hyperelastic models (Mooney-Rivlin, Ogden, Blatz-Ko, etc.)**
- Nonlinear Geometry: Large deformation, Large rotation, Follower force
- Nonlinear Contact: Surface to surface contact/Single surface contact, Sliding/Rough/General Contact (friction supported)
- **Automatic load steps supported**, various Iterative methods/Stiffness update method and Convergence criterion provided
- **Status of convergence and interim results during analysis, re-analysis (restart)** provided



Forced inserted Rubber Analysis

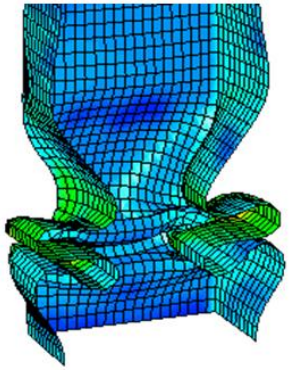
Comparison with real experiments: Reliability confirmation



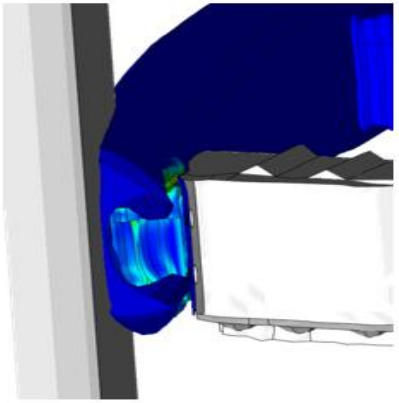
Buckling Analysis of polyethylene container (geometric nonlinearity).

Advanced Nonlinear drop/impact analysis using a single work environment & model

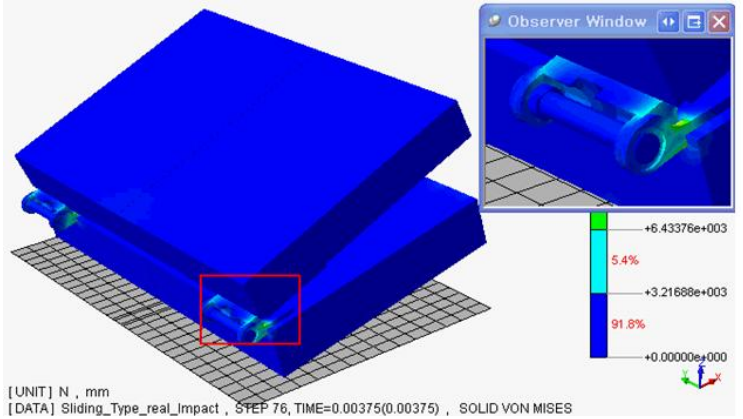
- Nonlinear Material: **Elasto-plastic model, Hyperelastic models (Mooney-Rivlin, Ogden, Blatz-Ko, etc.)**
- Nonlinear Geometry: Large deformation, Large rotation, Follower force
- Nonlinear Contact: Surface to surface contact/ Single surface contact, Sliding/ Rough/ General Contact (friction supported)
- Accurate and convenient analysis using various elements including Hexahedron, Higher-order Tetrahedron, Pyramid, etc.
- Auto-calculation of safe time steps by elements and various Mass Scale/Damping supported
- Applications to Molding/Processing Analysis using **Implicit Analysis and Sequentially Coupled Analysis**



Buckling Analysis of a box
Subjected to impact



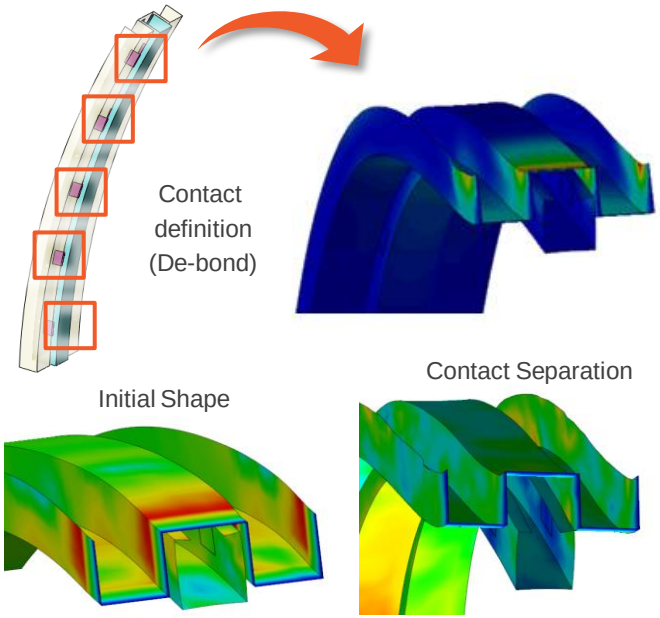
Low-speed front collision
(**Front Bumper**)



Drop analysis of mobile

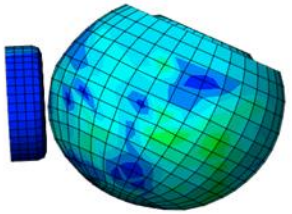
Various functions and outstanding performance for practical design analysis

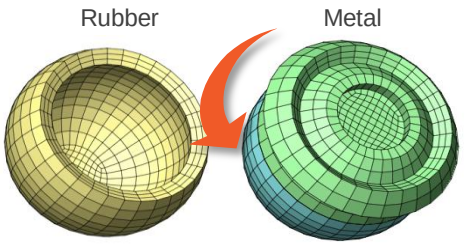
- Separation of contact/Re-contact functions provided
- Auto constraint of Rigid Body through selection of **Rigid Material** ⇨ Improved performance
- Re-start function provided to link to Subcases ⇨ Sequential Coupled Analysis supported
- **Enhanced Parallel processing using Multi-core processors**



Contact separation/re-contact

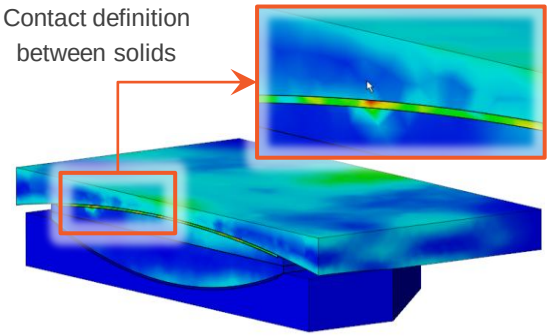
Enhanced performance using Rigid materials

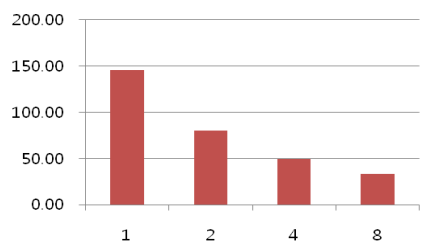




	Time step
Metal	1.5E-7 sec
Rigid material	2.5E-6 sec

Contact definition between solids

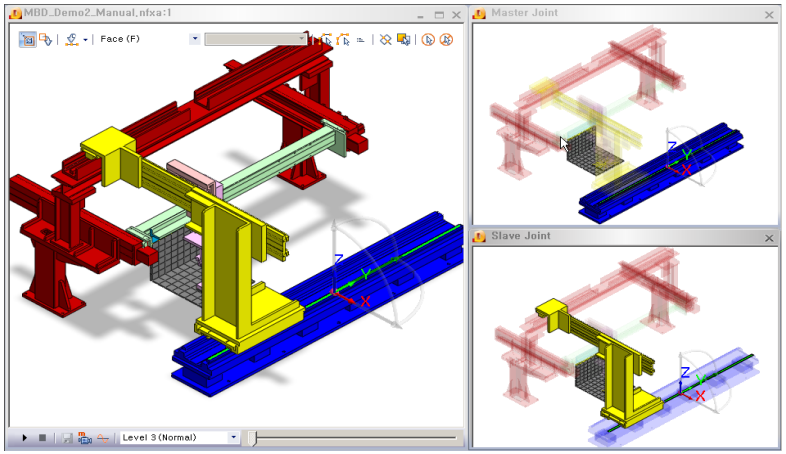




Performance improvement using Multi-cores

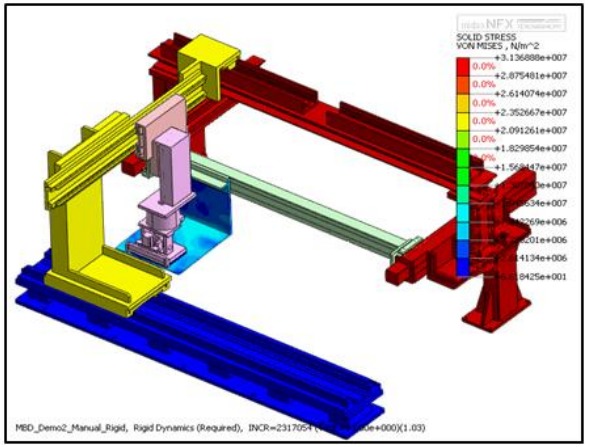
Simultaneous analysis of Rigid & Flexible bodies in a single work environment

- Modeling & Analysis of multi-bodies in the existing finite element mesh based work environment
- Simultaneous analysis of geometric shape based rigid body and mesh based flexible body
- Efficient analysis focused only on interested parts
- Existing nonlinear functions identically used ⇨ Possible to simulate Geometric, Material & Contact nonlinear behaviors

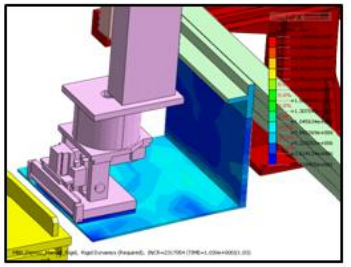


Work window for Modeling

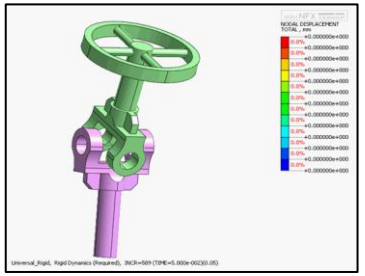
Detailed definition of joints



Results of explicit dynamic analysis



Stresses due to nonlinear contacts

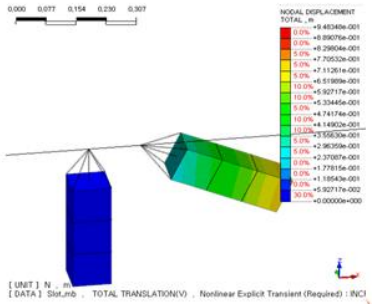


Rigid-Rigid Joint (Universal)

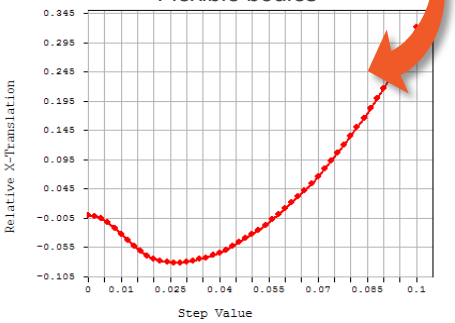
Definition of Rigid/Flexible assemblies with detail modeling of joints

Linear/Nonlinear Analysis using various joints

- Joints: Spherical, Cylindrical, Slot, Revolute, Planar, General, etc.
- Various options provided: Internal stiffness of joints, damping, etc.
- Various results generated: Relative displacements, rotations and reactions of joints, etc.

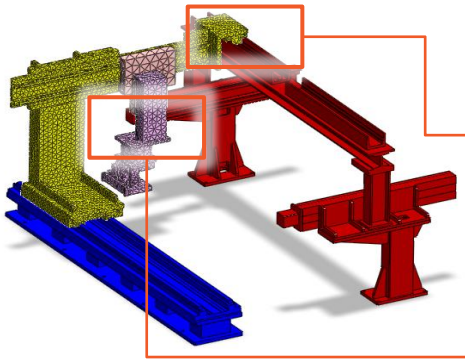


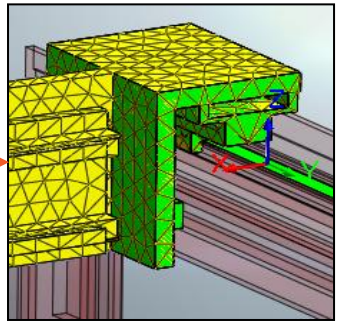
Relative motion between Flexible-Flexible bodies



Joint Relative displacement

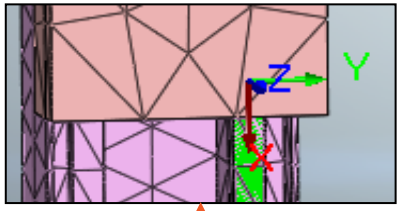
Modal Analysis using Rigid/Flexible bodies and joints

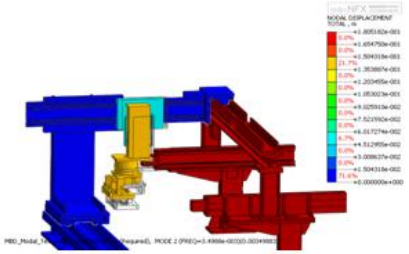




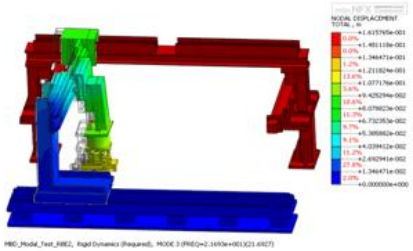
Rigid-Flexible body Connection

Flexible-Flexible body Connection

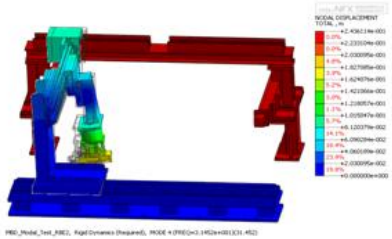




1st, 2nd Mode (Rigid Mode)



3rd Mode (21.7Hz)

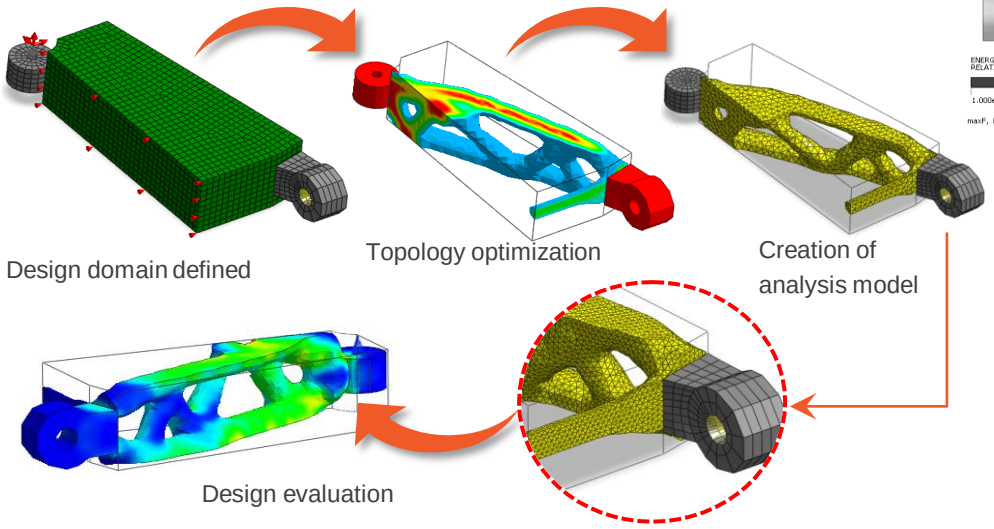


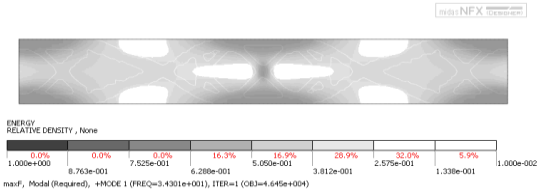
4th Mode (31.5 Hz)

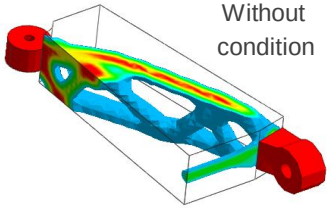
Topology Optimization reflecting static/dynamic analysis & manufacturing process

- Analysis types that can be coupled: Linear Static, Modal, Frequency Response
- Possible to specify Limitations and Design conditions such as Stress, Displacement, Volume or Manufacturing conditions (draw direction/symmetric conditions)
- Simultaneous optimization compatible with various operational and load conditions
- Analysis model auto-regeneration & Mesh smoothing functions (without separate CAD operation)
- Other practical features of convenience: Mode trace, Definition of design/non-design domains, Default value setup

Process of using topology optimization design

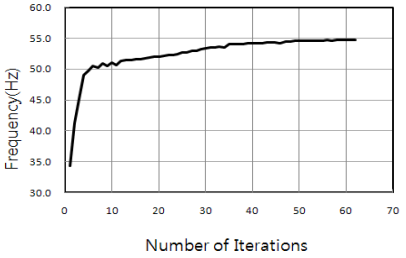






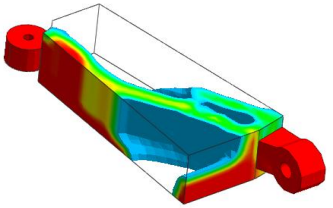
Without condition

Eigenvalues History



Optimized design retaining the maximal eigenvalue

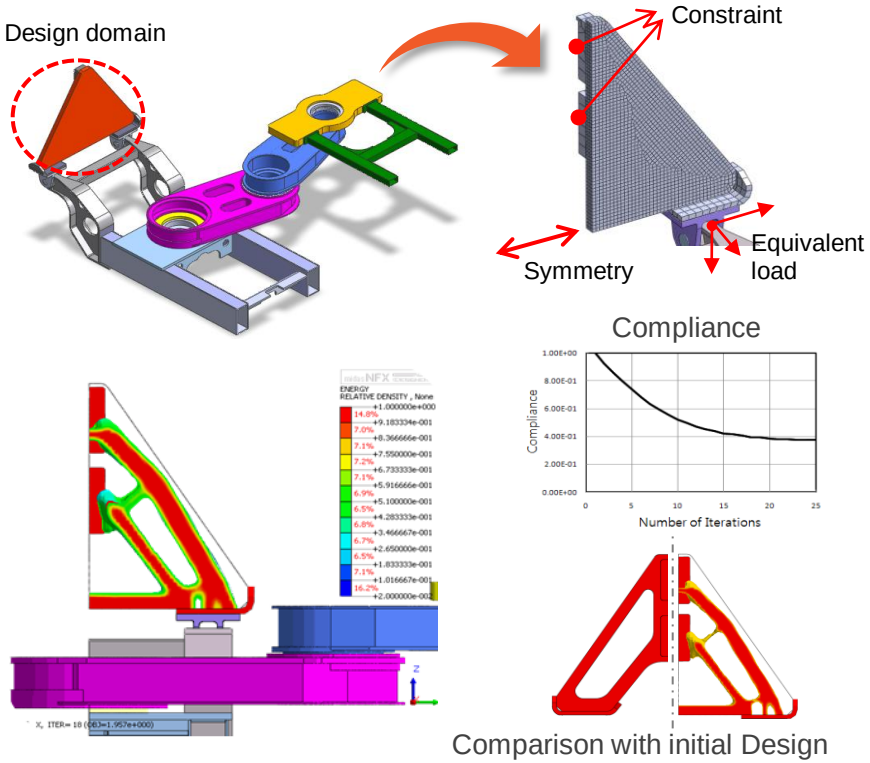
Manufacturing (Draw direction) condition



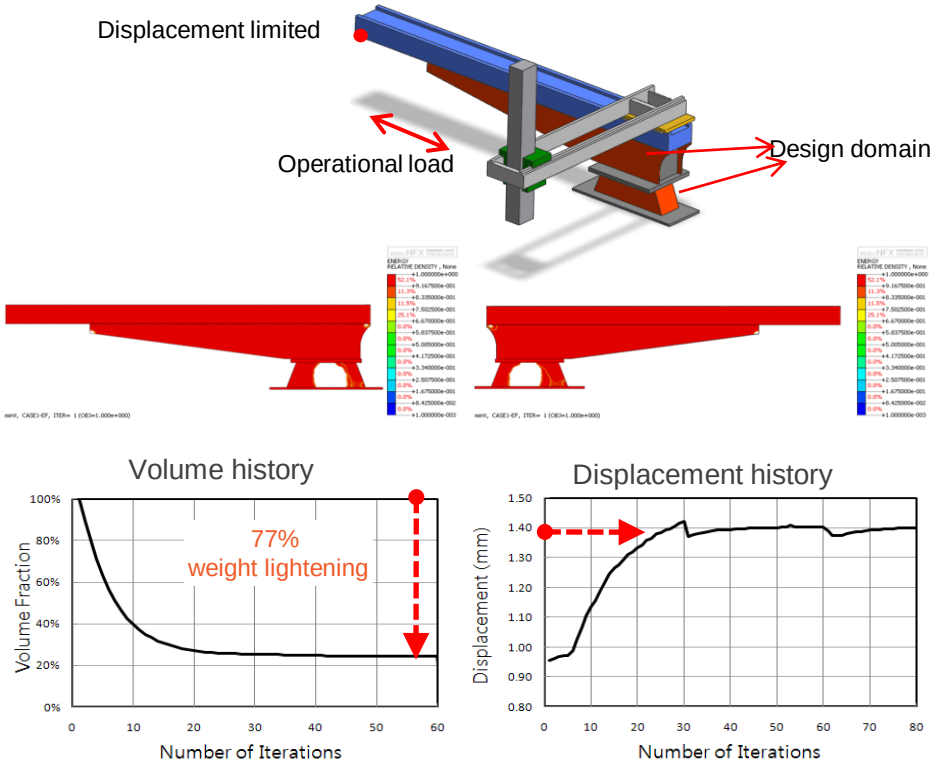
Topology Optimization considering manufacturing process

Case studies of Advanced Design Concept using topology optimization

- Design Objective: Maximize stiffness
- Analysis Condition: Linear static analysis
- Design Condition: Reduce weight by 45%

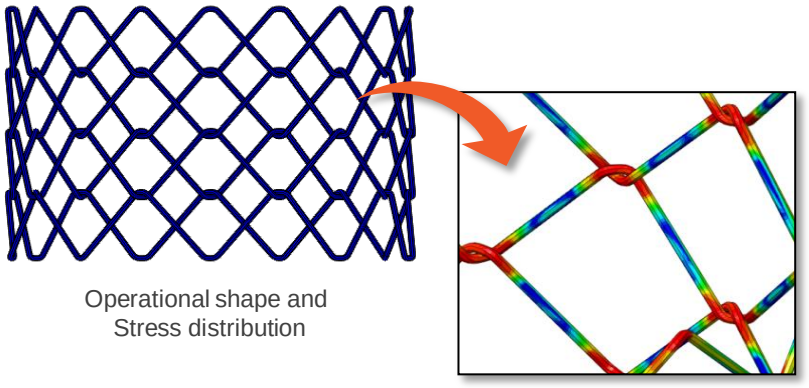


- Design Objective: Minimize volume
- Analysis Condition: Frequency response analysis
- Design Condition: Displacement limited to 1.4mm



Simple evaluation of Fatigue/Durability with minimum data

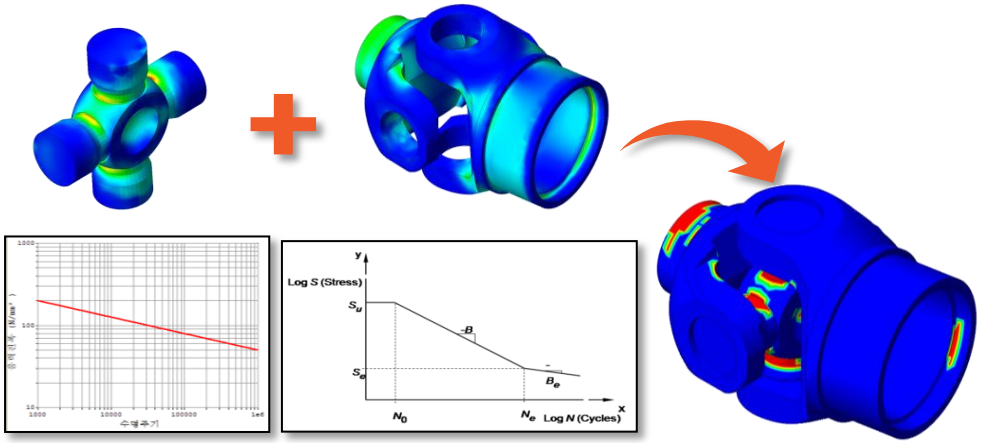
- Independent post-processing features provides (No separate/additional analysis required)
- **Fatigue Life**, **Results of Damage level provided**
- Linear/Multi linear S-N Curves by materials supported
- Stress Evaluation: Equivalent stress, Signed Von Mises (ductile materials), Principal stress (brittle materials), Shear force, maximum stress, minimum stress and mean stress
- Rainflow Counting, Mean Stress Correction (Goodman, Gerber) supported



Operational shape and
Stress distribution

Fatigue life

Evaluation of Fatigue Durability of a medical Stent



Test data input

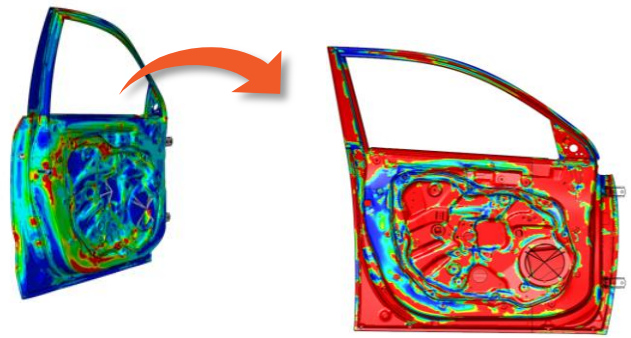
Direct input

Results of Fatigue life
Based on mixed definitions

Fatigue Analysis based on separate S-N Curve definitions for different materials

Enhanced Fatigue Analysis Functions

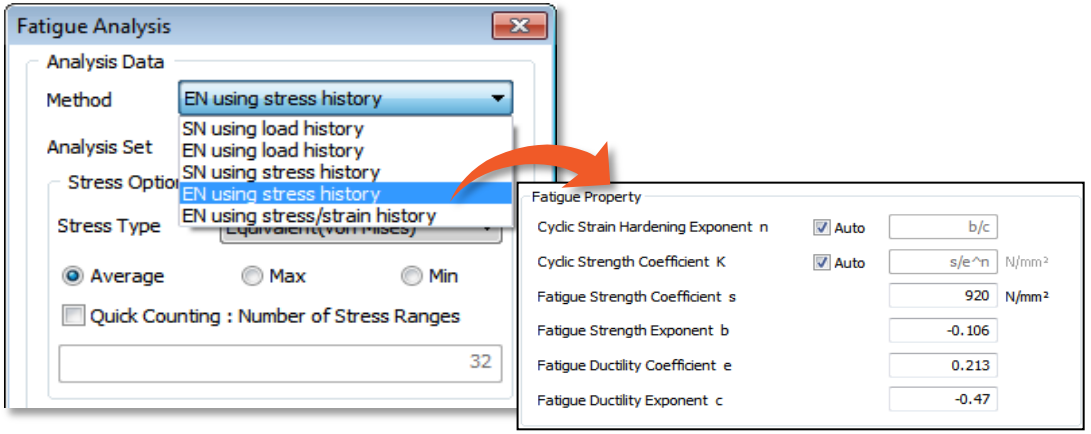
- **Addition of Strain-Life (e-N) Fatigue Analysis Method**
  - Mean Stress Correction: Morrow, SWT (Smith-Watson-Topper) supported
- Fatigue Analysis in Time Domain (Fatigue Analysis for time-dependent load/stress history)
  - Automatic definition of Fatigue Load for stress results at individual steps
  - Direct Use of Material nonlinear analysis results (Nonlinear stress/Strain history)
- Rainflow counting option (Quick Counting)
  - ASME Standard, Reduction in computation time



Operational shape and stress distribution

Fatigue life

Fatigue analysis for time-dependent loading

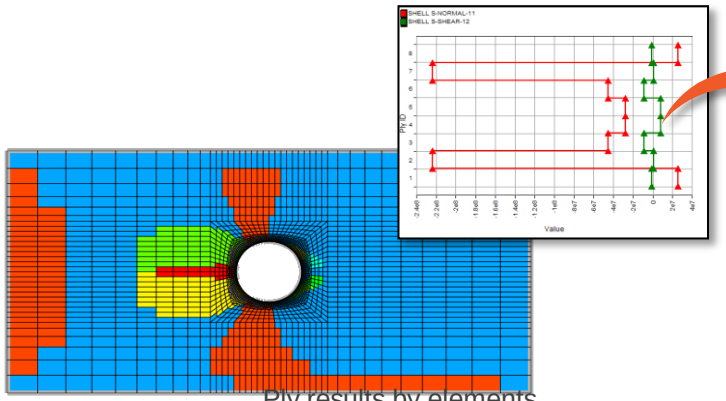


Fatigue Analysis dialog box

e-N Diagram dialog box

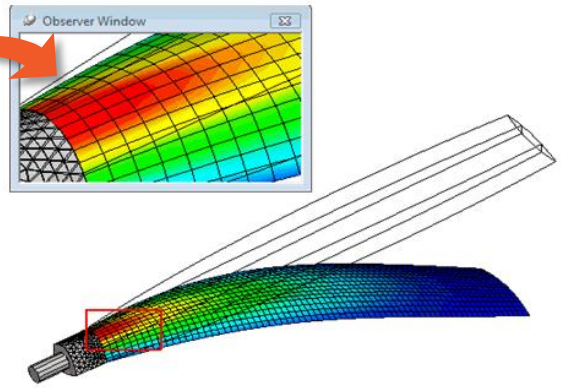
Intuitive GUI and dedicated pre/post-processing for modeling & practical analysis

- 2D / 3D Composite materials (Linear / Nonlinear) supported
- Failure Theory
  Hill, Hoffman, Tsai-Wu, Maximum stress, Maximum strain, NASA LaRC02
- Failure Criteria
  Failure Index, FE Failure Index, Strength Ratio
- Property Matrices (A, B & D) calculation, Definition of various material directions (Angle, Coordinate system, Vector, etc.)
- Global Ply ID supported, Top/bottom fiber results per ply produced

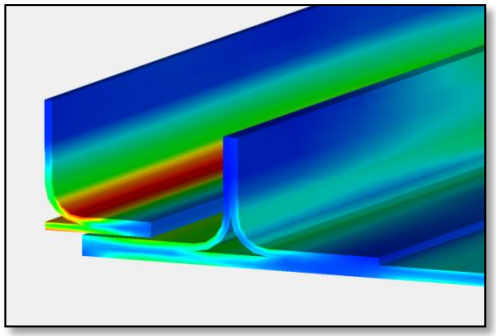


Ply results by elements

Ply Maximum/Minimum results

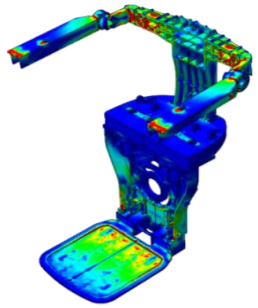
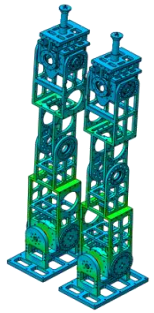
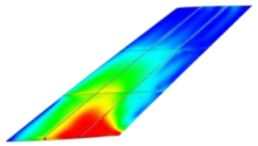
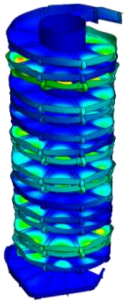
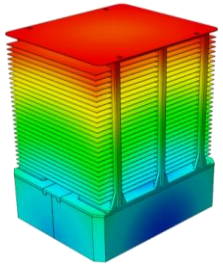


Composite wind turbine blade
(**Geometric nonlinear analysis**)



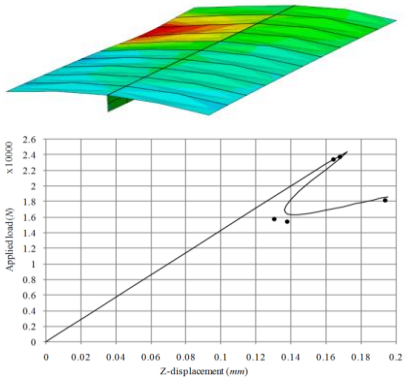
Example Analysis of composite material solid

World class high performance parallel solvers catered to large scale models

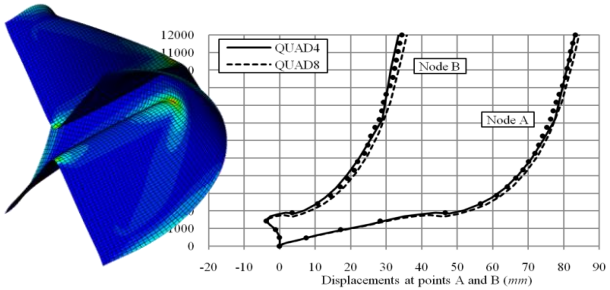
<table>
<tr>
<td colspan="3">  </td>
<td>  </td>
<td>  </td>
<td>  </td>
<td colspan="2">  </td>
</tr>
<tr>
<td colspan="3">Linear Static Analysis</td>
<td>Linear Static Analysis</td>
<td>Linear Static Analysis</td>
<td>Modal Analysis
(50 Modes)</td>
<td>Linear Static Analysis</td>
<td>Nonlinear Static Analysis
(nonlinear contact)</td>
</tr>
<tr>
<td colspan="3">Solid Mesh (156,862)
Nodes (272,597)
Degrees of Freedom (817,791)</td>
<td>Solid Mesh (163,143)
Nodes (303,347)
Degrees of Freedom (916,890)</td>
<td>Shell/Beam Mesh (156,862)
Nodes (272,597)
Degrees of Freedom (951,378)</td>
<td>Shell Mesh (170,123)
Nodes (162,391)
Degrees of Freedom (974,346)</td>
<td colspan="2">Solid Mesh (154,320)
Nodes (280,060)
Degrees of Freedom (840,180)</td>
</tr>
<tr>
<td>32bits</td>
<td colspan="2">64bits</td>
<td>32bits</td>
<td>32bits</td>
<td>64bits</td>
<td colspan="2">64bits</td>
</tr>
<tr>
<td>AMG</td>
<td>MFS</td>
<td>AMG</td>
<td>AMG</td>
<td>AMG</td>
<td>MFS</td>
<td colspan="2">MFS</td>
</tr>
<tr>
<td>232 sec</td>
<td>149 sec</td>
<td>138 sec</td>
<td>548 sec</td>
<td>160 sec</td>
<td>1.16 Hour</td>
<td>51 sec</td>
<td>1010 sec,
No of iterations: 27
→ 1 time: 37 sec</td>
</tr>
</table>

※ MFS (Multi-frontal, Direct Method), AMG (Algebraic Multigrid, Iterative Method)

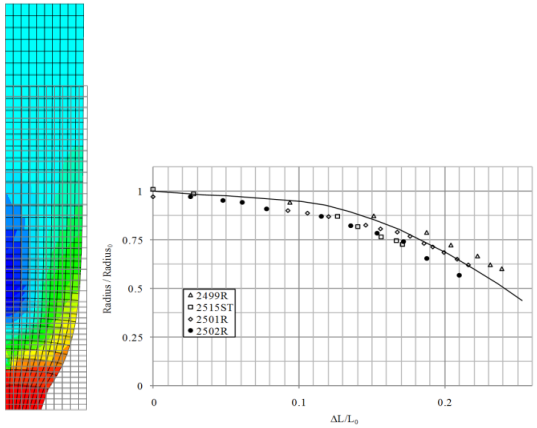
Reliable & excellent analysis results with the latest elements and analysis algorithms



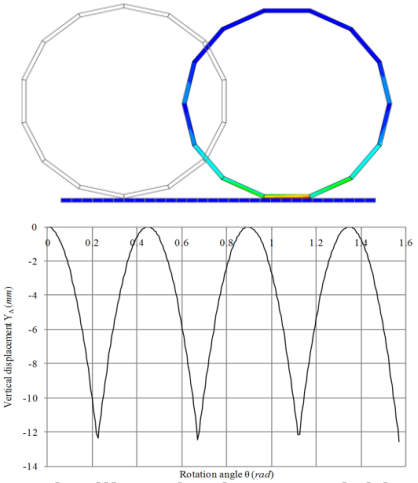
Stiffened cylindrical panel
(NAFEMS, Material/Geometric Nonlinearity)



Pinched cylinder with rigid diaphragms
(K.Y.Sze, W.K.Chan & T.H.H.Pian, Geometric Nonlinearity)

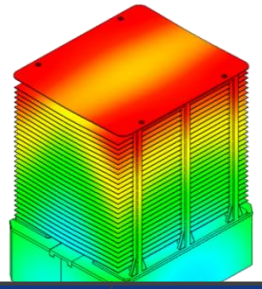


Necking of a circular bar
(J.C. Simo, T.J.R. Hughes, Material/Geometric Nonlinearity)



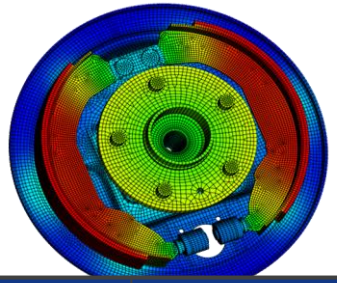
Sliding and rolling of a ring on a rigid surface
(NAFEMS, Boundary Nonlinearity)

Benchmarking Tests/Verification of Reliability for practical examples



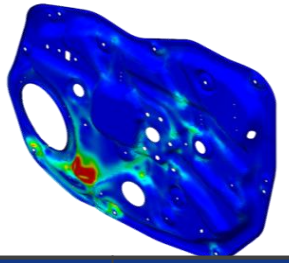
	Max. Deflection
midas NFX	0.67 mm
Program N	0.66 mm

Linear Static Analysis
(Solid element, Linear contact, Temperature load, Selfweight)



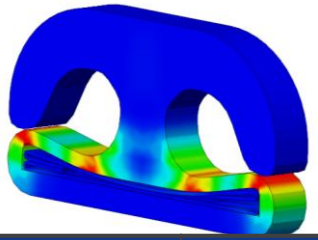
	Final Temperature
midas NFX	118.0 °C
Program A	117.6 °C

Heat Transfer Analysis (Transient)
(Solid element, Temperature condition, Convection)



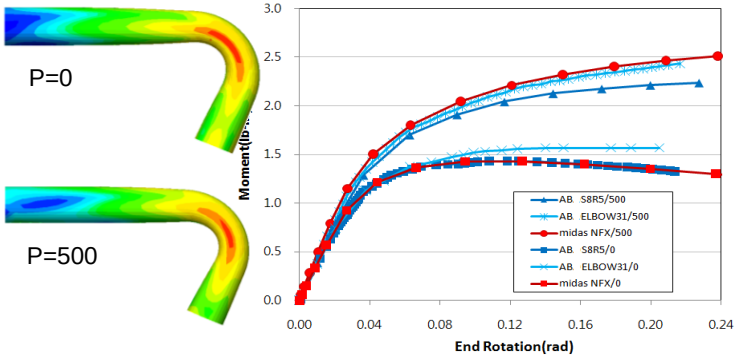
	Max. Stress
midas NFX	0.156 GPa
Program A	0.157 GPa

Material/Geometric Nonlinear Analysis
(Shell element, Large Deformations, Distributed Load)

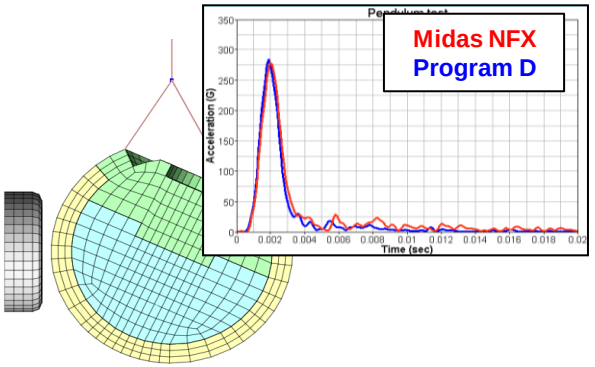


	Contact Load
midas NFX	10.88 N
Program N	10.87 N

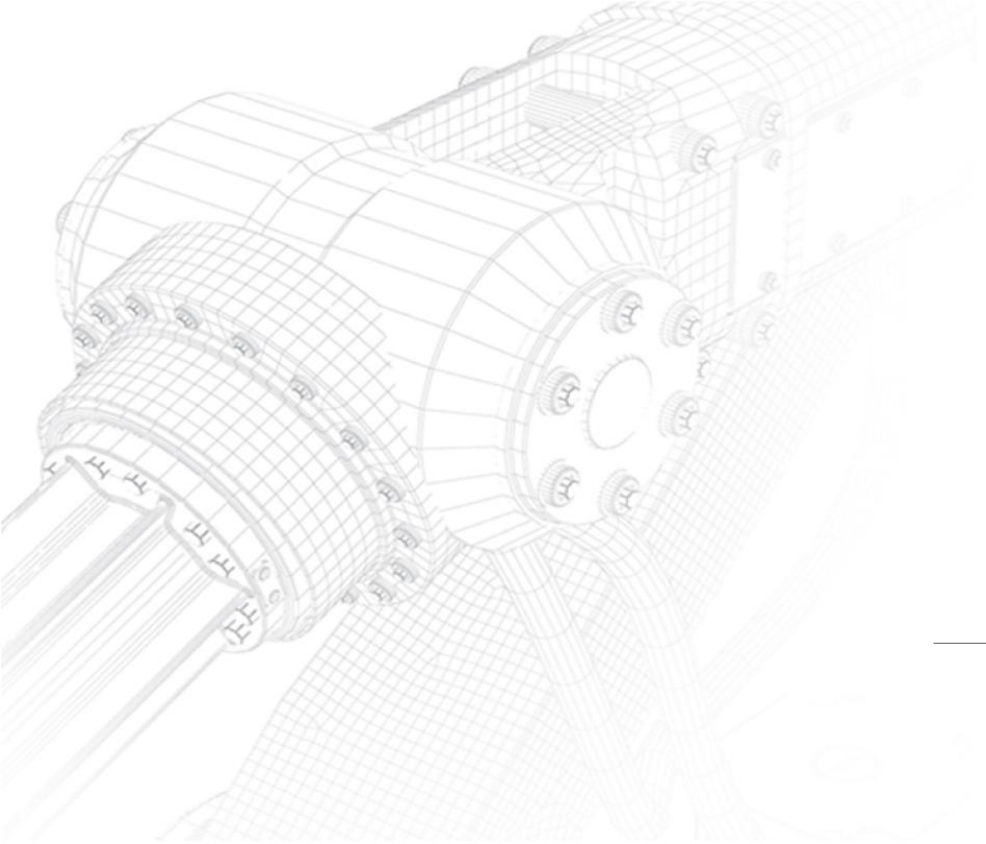
Nonlinear Contact Analysis
(Solid element, Enforced displacements)



Material/Geometric Nonlinear Analysis
(Shell element, Pressure load, Moment load)



Explicit Dynamic Analysis
(Solid element, Rubber material, Contact)

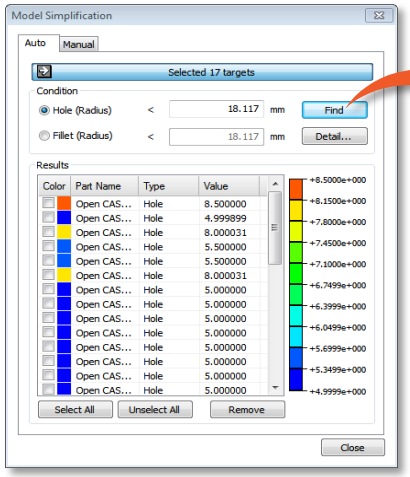


midas NFX

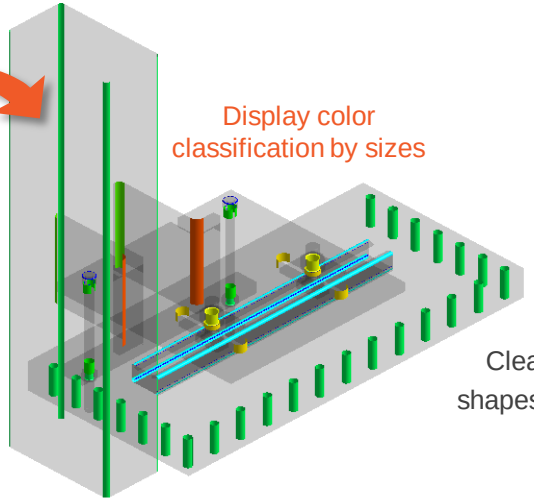
Total Solutions for True Analysis-driven Design

Contents

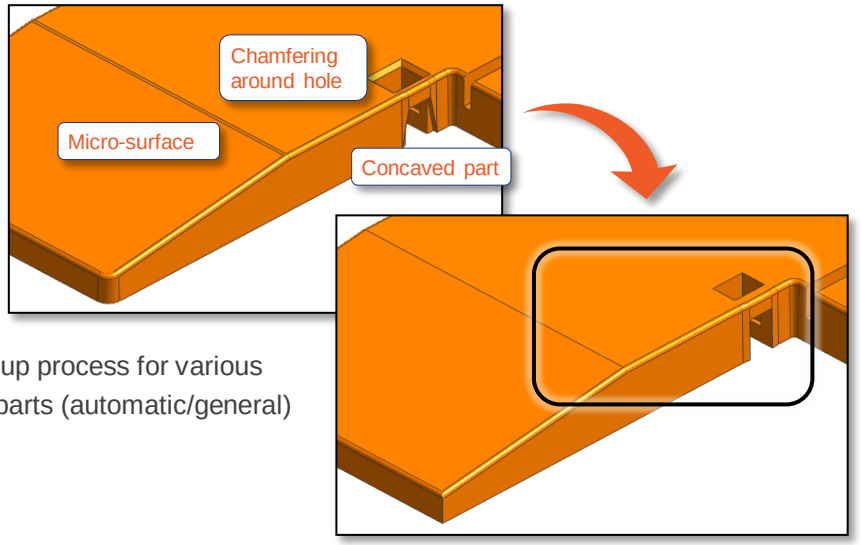
Intuitive and powerful cleanup feature to create efficient analysis models



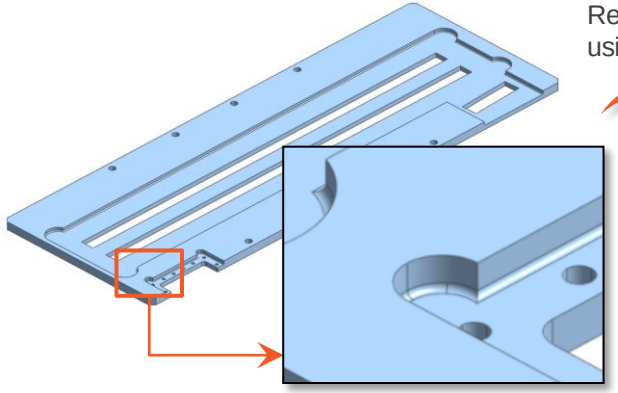
Intuitive interface for easy search and identification of objects (holes/fillets) to be removed



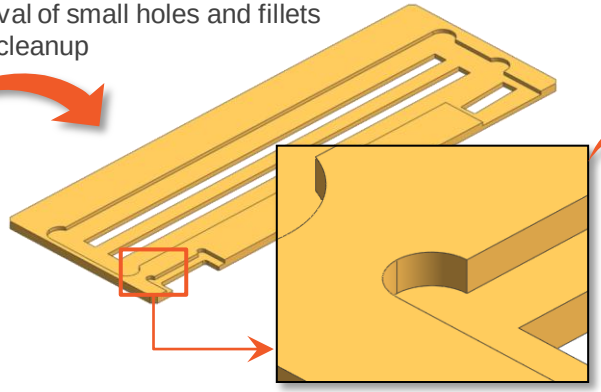
Display color classification by sizes

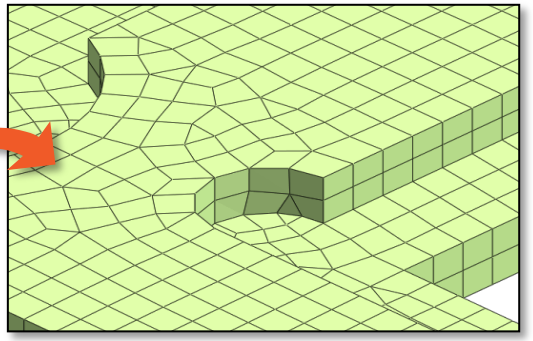


Cleanup process for various shapes/parts (automatic/general)



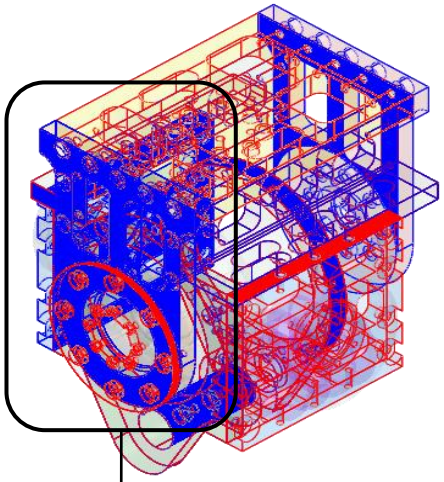
Removal of small holes and fillets using cleanup



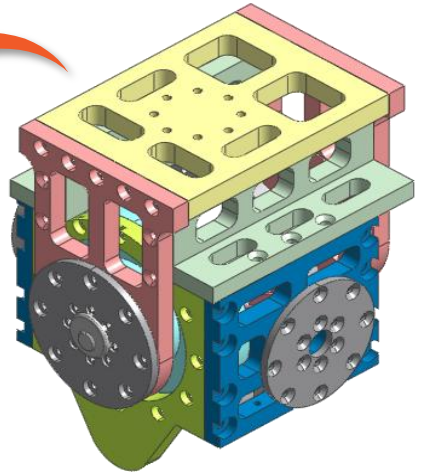


Hybrid mesh created after cleanup
(Reduction in the numbers of elements/nodes and improvement of element mesh quality)

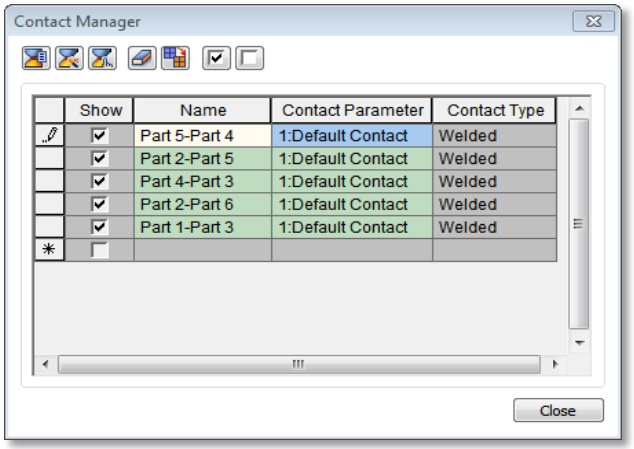
Automated contact definition suitable for complex, large scale assemblies and convenient visualization and management



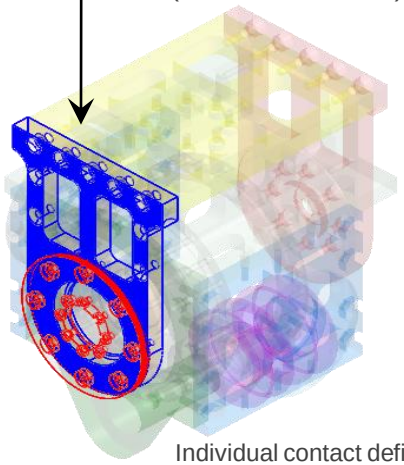
Completed auto contact definition
(Total model check)



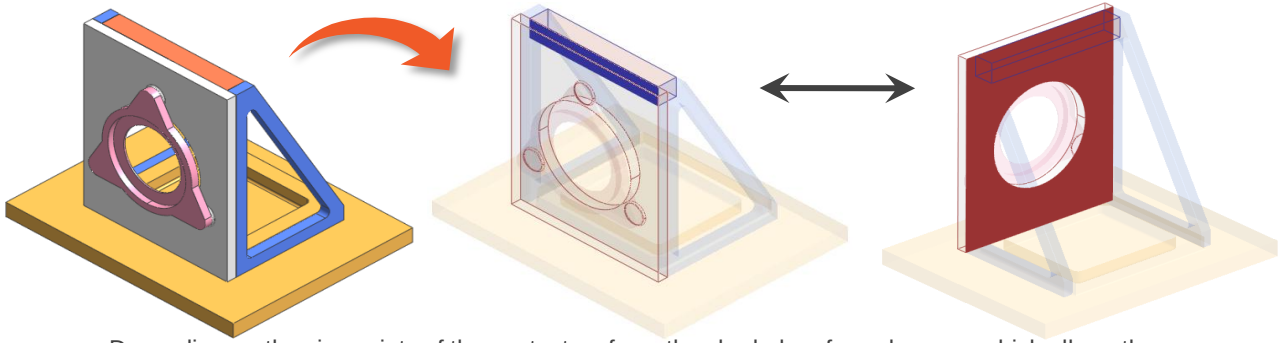
Assembly Model



Contact Manager to conveniently check, revise and manage contact definitions

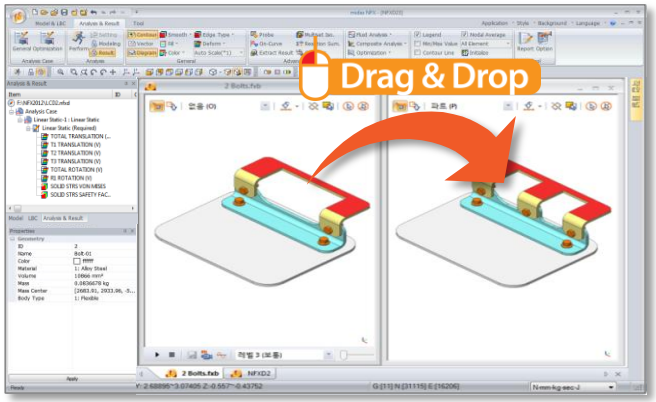


Individual contact definition check



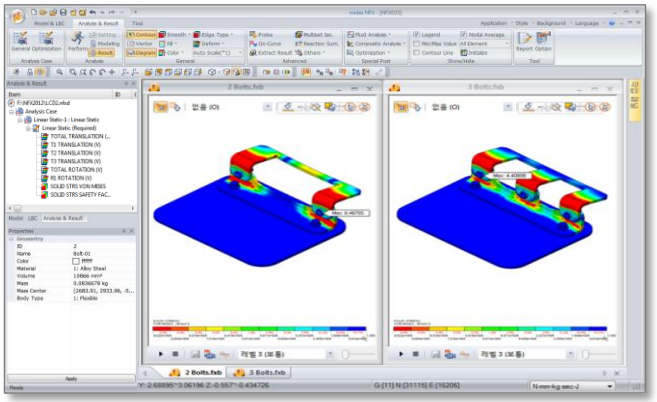
Depending on the viewpoints of the contact surface, the shaded surface changes, which allows the user to easily check the location of the contact surface.

Auto-update of analysis model for simple re-analysis in case of change in CAD model

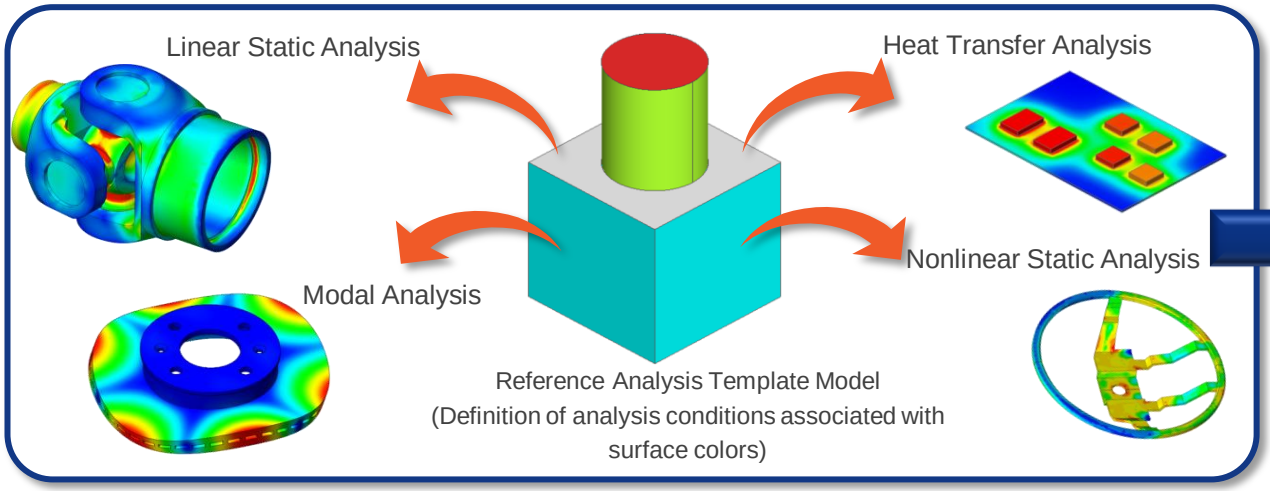


By simple Drag & Drop of the analysis cases of the present model to the revised model, all the loads, boundary conditions, analysis types and conditions are automatically transferred to the revised model.

Comparison of re-analysis results

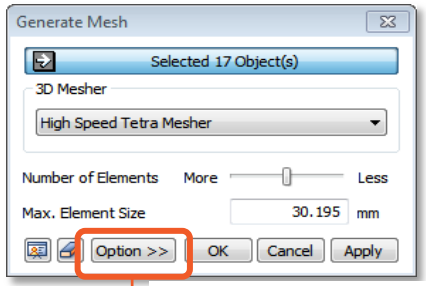


After re-analyzing the revised model, the results before and after the revision can be conveniently checked and compared in a multi-window view.

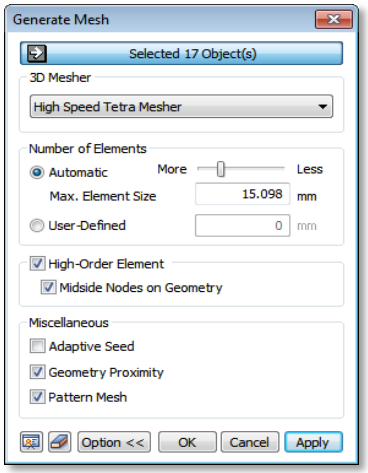


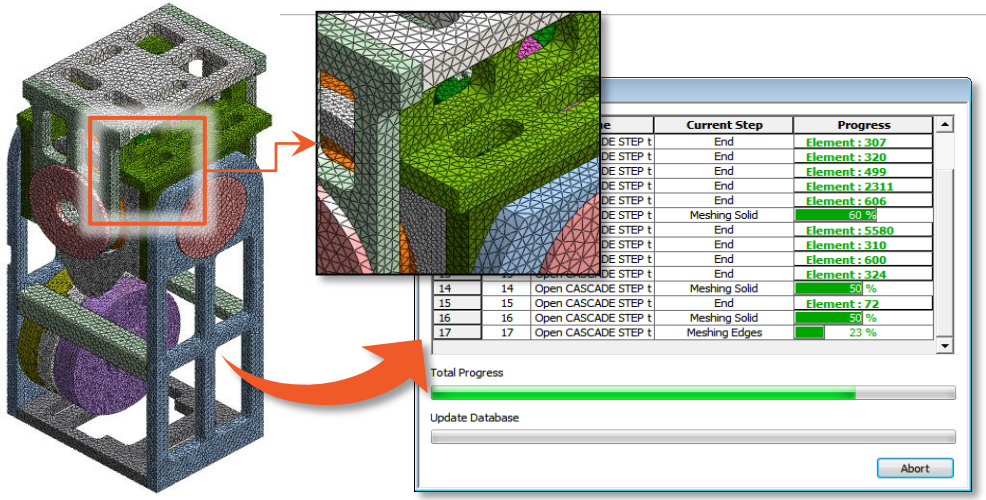
Automatic transfer of all the conditions based on the surface colors irrespective of the analysis types and similarity of the geometric shapes of models
→ Standardization of Analysis
→ Designer's analysis capability empowered

Parallel processing based high performance automatic mesh generation through a simple interface



Simple and convenient interface



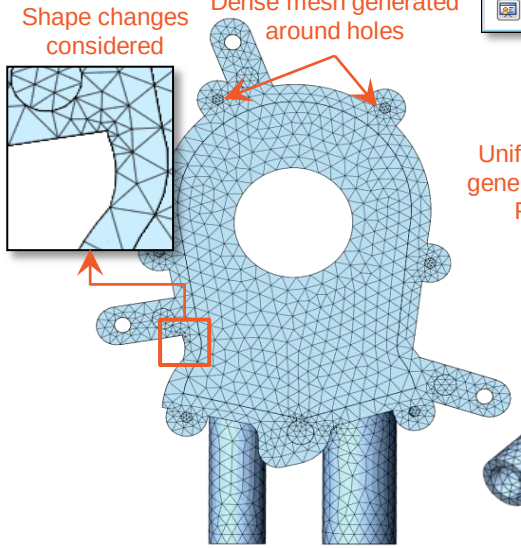


Assembly model of 113 parts
(Elements: 640,000, Nodes: 1.06 million)

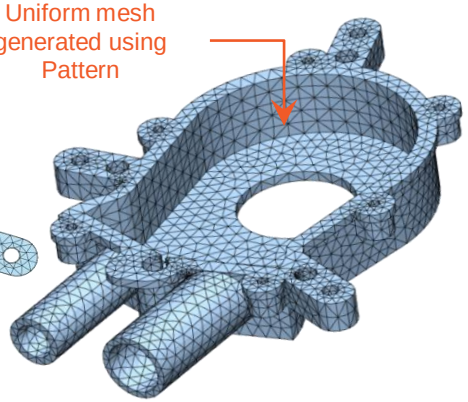
Mesh generation using multi-core parallel processing

Shape changes considered

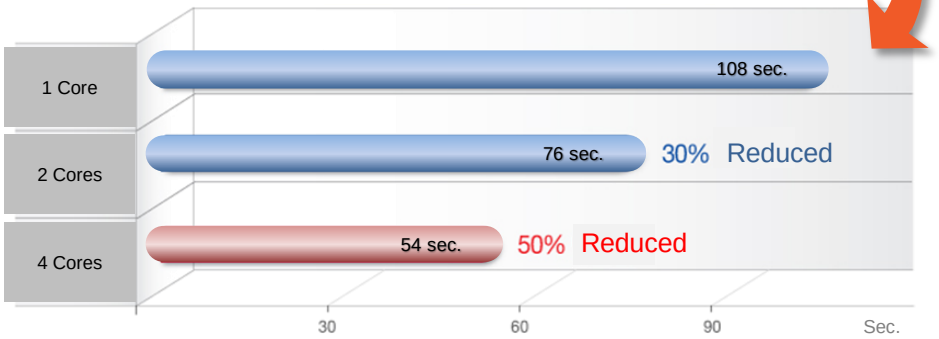
Dense mesh generated around holes



Uniform mesh generated using Pattern

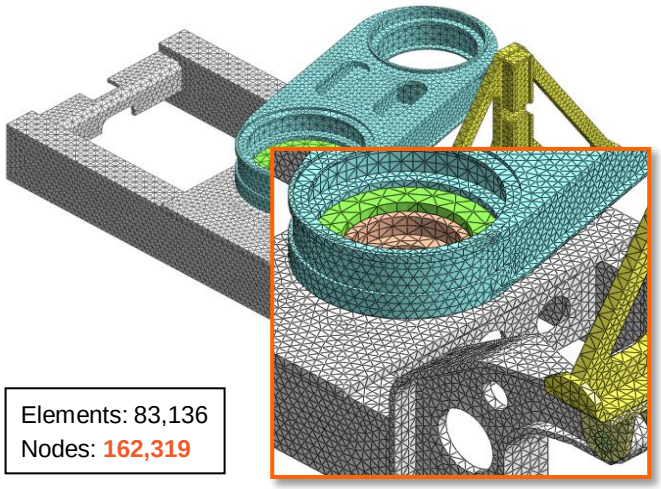


High quality mesh auto-generated with automatic control of density and shape

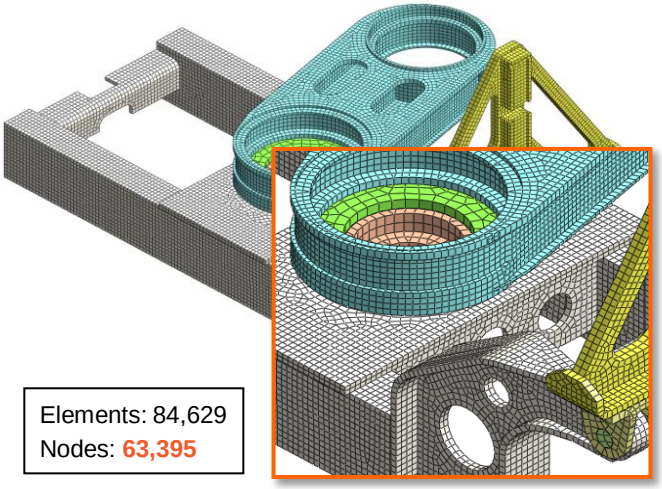


**With parallel processing on a normal PC,
640,000 elements in 113 parts generated within 1 minute**

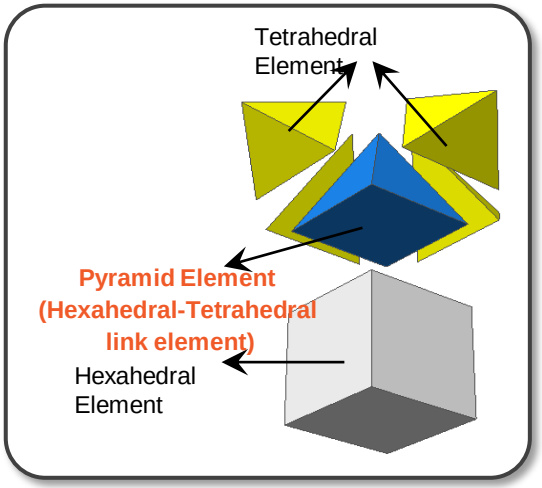
Latest hybrid element mesh generator leading to efficient analysis and superb results



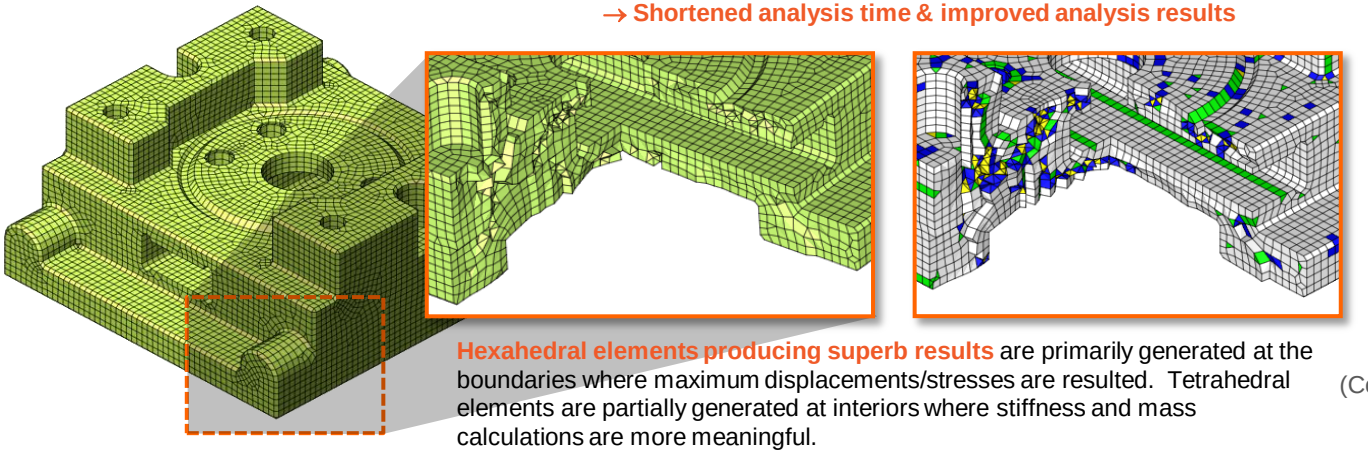
General higher order tetrahedral mesh



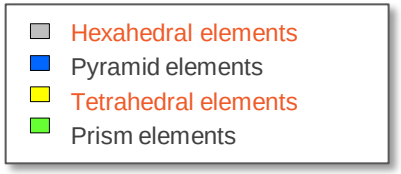
Hexahedral-Tetrahedral Hybrid Mesh
(same number of elements with approximately 1/3 of the nodes)
→ **Shortened analysis time & improved analysis results**



Composition of hybrid element mesh

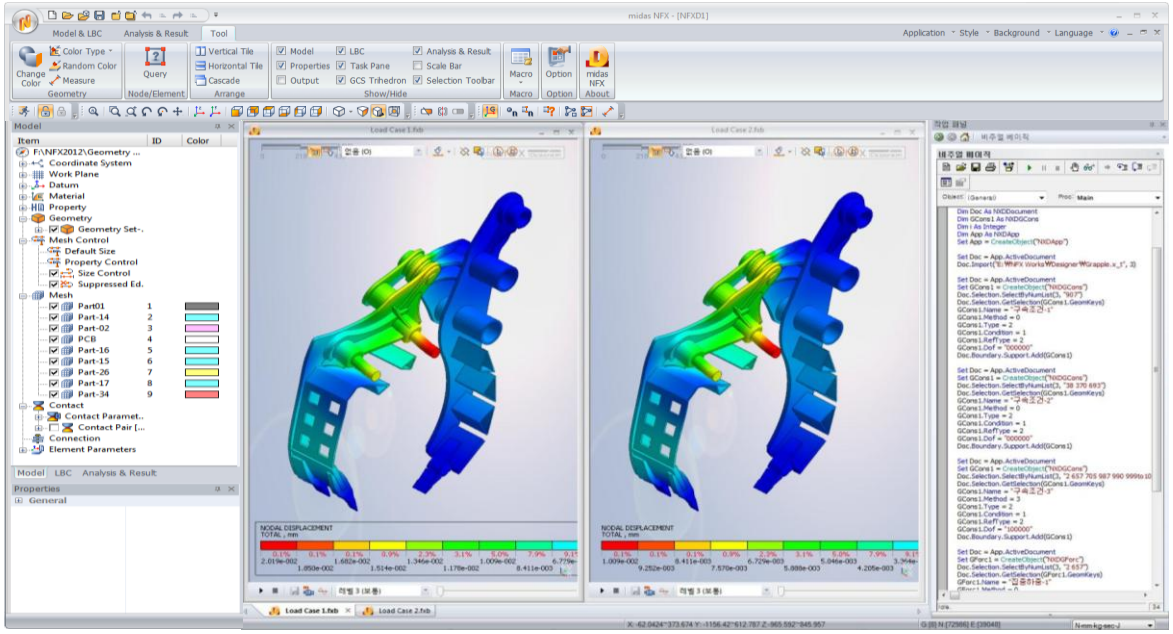


Hexahedral elements producing superb results are primarily generated at the boundaries where maximum displacements/stresses are resulted. Tetrahedral elements are partially generated at interiors where stiffness and mass calculations are more meaningful.

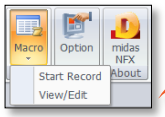


Element distribution of hybrid mesh
(Color representation of each element type)

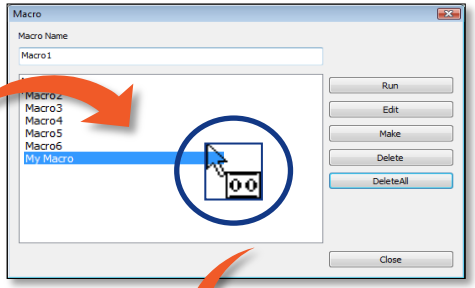
Automated processing of repetitive tasks using a variety of automatic recording of VBA macro

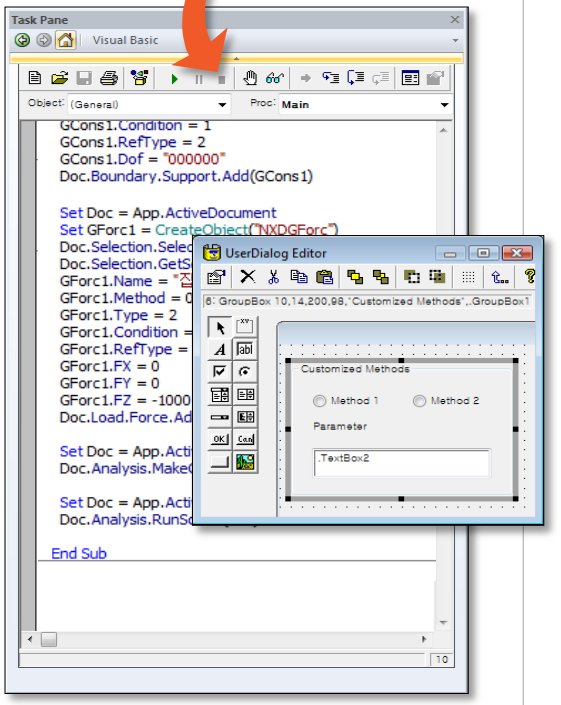


Example of comparing results after repeated analyses by changing the magnitude of loads in the auto-recorded macro



Macro
Auto-recording



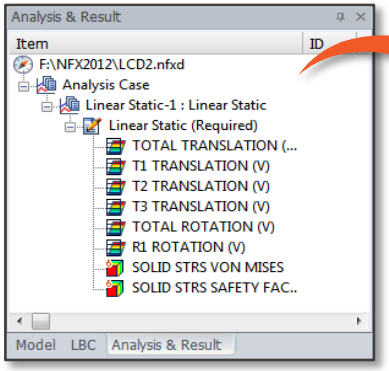


Macro of Standard VBA format

- Automatic macro recording enables the user to simply automate the work process without having any special knowledge in programming languages.
- Various applications include preparation of specific purpose dialog boxes and interface with other products in the standard VBA format.

Organization of results and auto-generation of practical report

Post-processing Result Tree

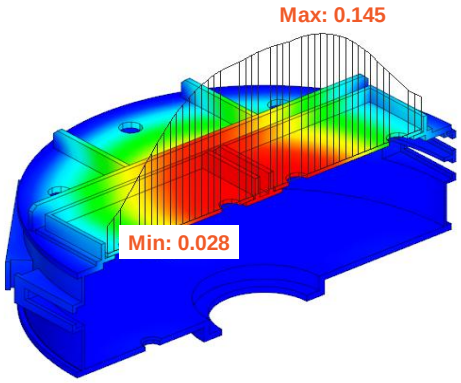


Numerical result table

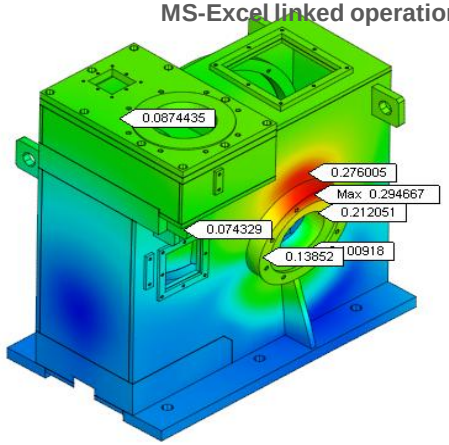
Node	TOTAL TRANSLATION(V)	T1 TRANSLATION(V)	T2 TRANSLATION(V)	T3 TRANSLATION(V)
1	1.058359e-003	0.000000e+000	0.000000e+000	-1.058359e-003
2	1.024430e-003	-1.398013e-011	-1.867688e-004	-1.007260e-003
3	9.674207e-004	-2.862719e-011	-1.621829e-004	-9.537293e-004
4	9.140114e-004	-3.524180e-011	-1.556210e-004	-9.006659e-004
5	8.626710e-004	-4.276236e-011	-1.603928e-004	-8.476293e-004
6	8.103070e-004	-4.494279e-011	-1.590659e-004	-7.945410e-004
7	7.583644e-004	-4.471968e-011	-1.591770e-004	-7.414710e-004
8	7.065779e-004	-4.251788e-011	-1.592490e-004	-6.883982e-
9	6.549701e-004	-3.896043e-011	-1.592106e-004	-6.353249e-004
10	6.036292e-004	-3.458692e-011	-1.592193e-004	-5.822520e-004
11	5.52			
12	5.02			
13	4.52			
14	4.02			
15	3.54			

	A	B	C	D	E
1	Node	Total Trans	X-Trans	Y-Trans	Z-Trans
2	1	1.05836E-03	0.00000E+00	0.00000E+00	-1.05836E-03
3	2	1.02443E-03	-1.39801E-11	-1.86769E-04	-1.00726E-03
4	3	9.67421E-04	-2.86272E-11	-1.62183E-04	-9.53729E-04
5	4	9.14011E-04	-3.52418E-11	-1.55621E-04	-9.00666E-04
6	5	8.62671E-04	-4.27624E-11	-1.60393E-04	-8.47629E-04
7	6	8.10307E-04	-4.49428E-11	-1.59066E-04	-7.94541E-04
8	7	7.58364E-04	-4.47197E-11	-1.59177E-04	-7.41471E-04
9	8	7.06578E-04	-4.25179E-11	-1.59249E-04	-6.88398E-04

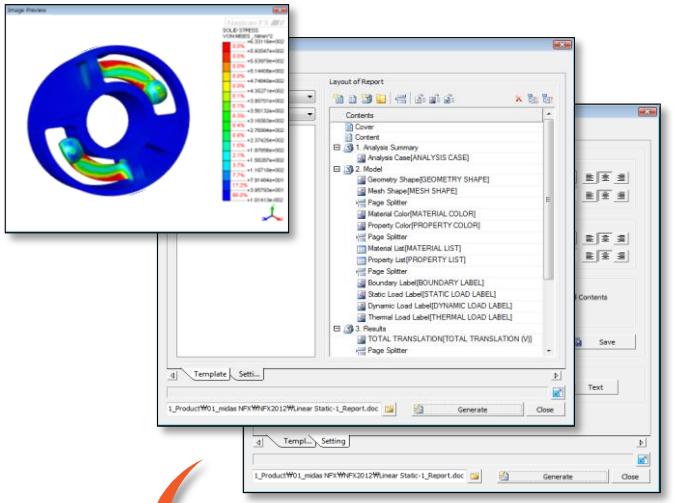
MS-Excel linked operations



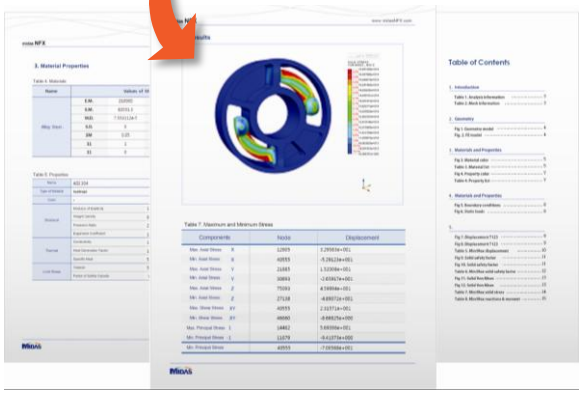
Result distribution graph along any specific line



Result inquiry at any location (tags displayed)

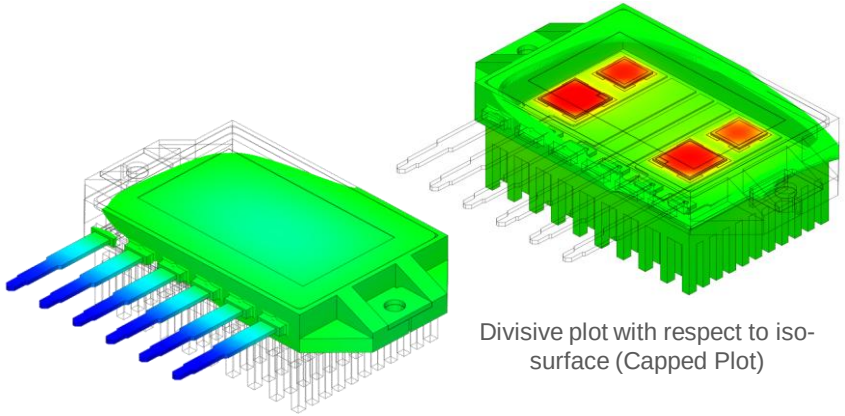


Auto Report Generation

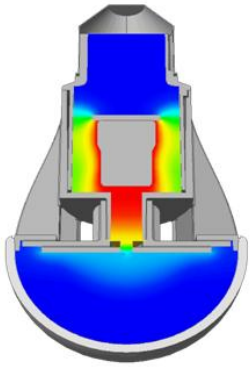


MS-Word Format Report
(specific customization possible)

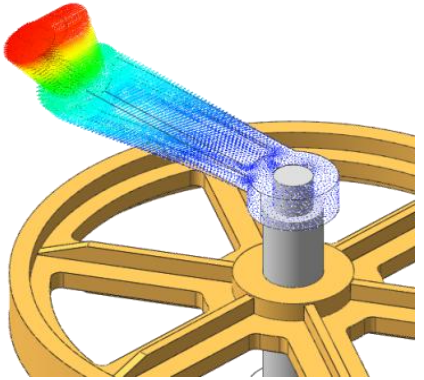
Diverse and sophisticated post-processing graphics



Divisive plot with respect to iso-surface (Capped Plot)



Slice plot of a specific plane

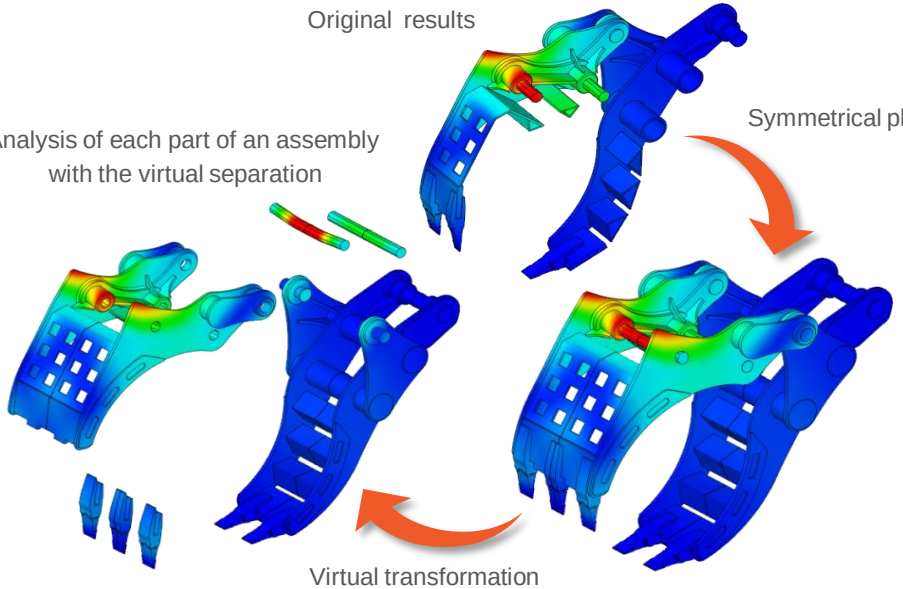


Vector plot

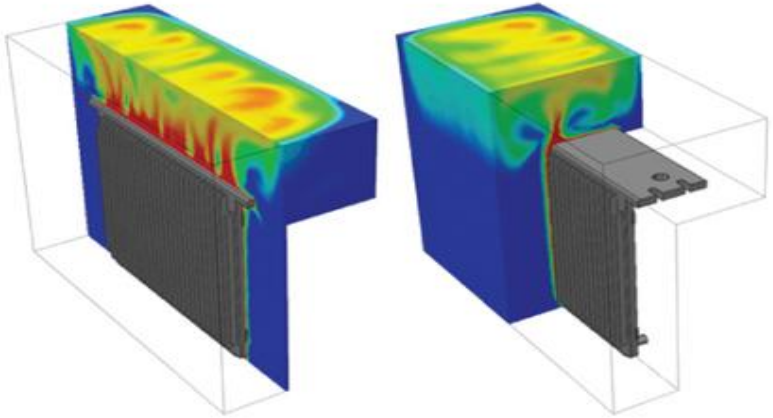
Original results

Analysis of each part of an assembly with the virtual separation

Symmetrical plot

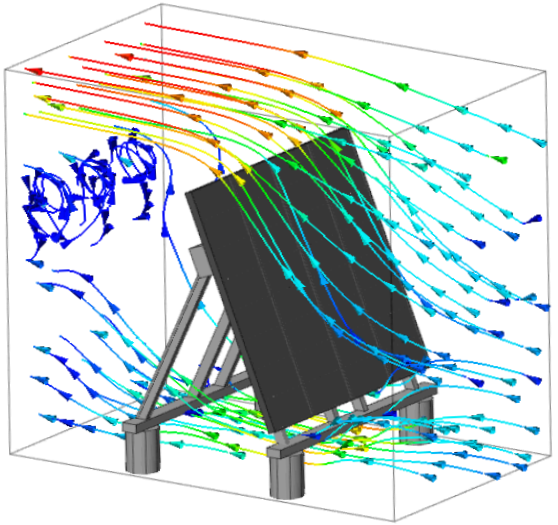


Virtual transformation

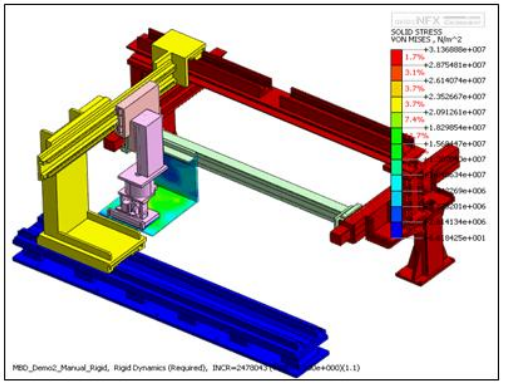


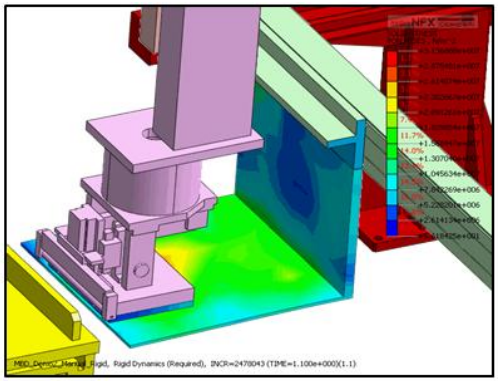
Checking changes in temperature using clipping at a specific plane

Special post-processing functions

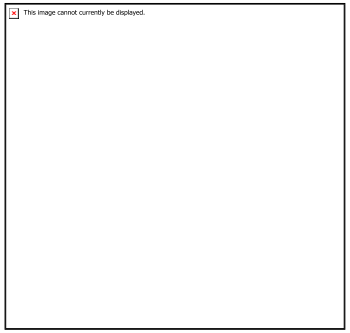


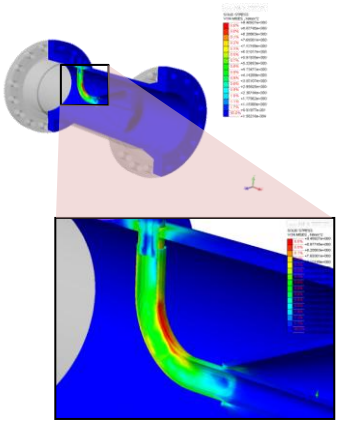
Stream line results to evaluate fluid flows



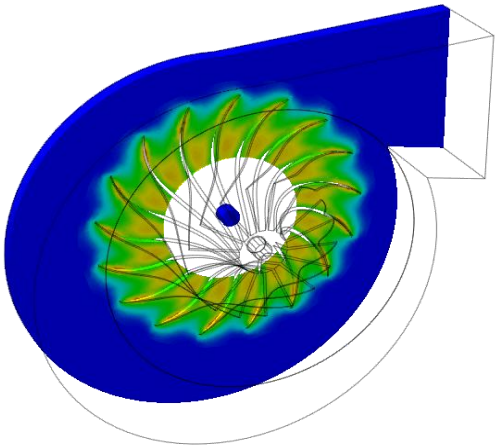


Post-processing example of simultaneous representation of geometrical shape and mesh based results (Multi-body dynamics)





Flux calculation at a specific pipe section



Mesh deformation function (Moving Mesh)

An insight into NFX 2012

Total Analysis Solutions for Multi-disciplinary Optimum Design

MIDAS