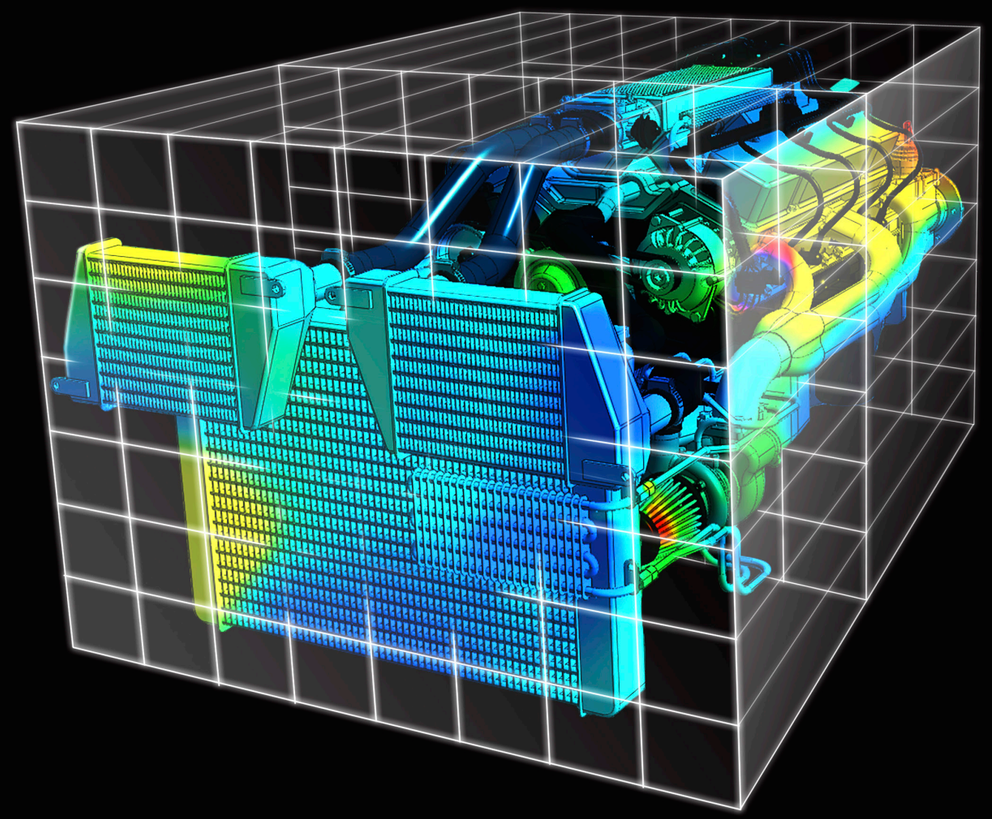
MIDAS MESHFREE

INNOVATIVE CAE SOLUTION, MIDAS MESHFREE

MIDAS MESHFREE

MIDAS

CONTENTS



Meet brand-new CAE

Only experts can use it?

“CAE is complicated, difficult, and time-consuming
because of the many tasks involved in the mesh generation.
It can only be used by a specialist or a person exclusively in charge.”
We broke this stereotypical image of CAE.

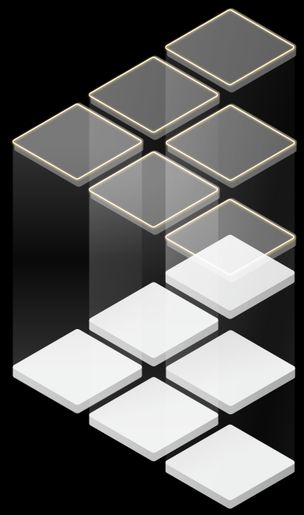
Everything starts from the origin.

In order to completely eliminate the limitations and
inefficiencies of traditional CAE, we have planned and
developed everything completely from the beginning.

MIDAS MESHFREE

New Design Tool

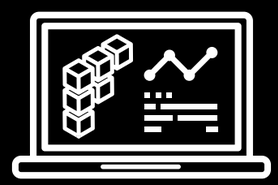
MESHFREE



MESHFREE is a cutting-edge design and analysis technology
developed through the collaboration with SAMSUNG Electronics.
It performs simulation of the original model you have designed without any modification.

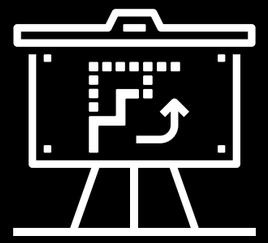
Until now, it took a lot of time and money to obtain the result.
However, MESHFREE will provide the opportunities to validate their
design through the analysis quickly, easily and accurately for everyone.
From now on, engineers can concentrate on more valuable tasks.

Now your technical know-how with MeshFree will be your competitive edge.



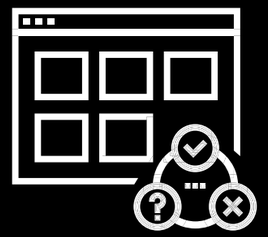
Analysis directly from 3D CAD Model

Conventional FEM required tedious and time consuming model idealization and cleanup process. However, MeshFree has the latest, efficient and effective numerical analysis algorithm that can simulate using the original 3D CAD model itself.



Simple 3-step Analysis Process

It is possible to start the simulation immediately after installation. 3-step intuitive workflow makes it easy for anyone to use. It will minimize the learning time and enable you to verify the working design model in a short time. Now, design engineers can quickly verify the performance of a model at the design stage.
It provides a revolutionary process over existing methods that required knowledge, manpower, and cost.



Powerful Design Modification

The ultimate goal of the simulation is to find out the possible flaw of the design product on the computer and to suggest a solution to the problem. MeshFree is able to use the 3D CAD model prototype as it is, and it has strong advantages in terms of the design modifications because it can generate the grid mesh independently of the complex shape. In addition, it automatically recognizes the changed part of the design, updates the model and automatically reflects the existing load and boundary conditions. Therefore it can be operated efficiently by eliminating unnecessary work that can occur repeatedly.

The ultimate goal of the simulation is **to find and solve problems at the design stage.**

Find problems in design products in advance on computer

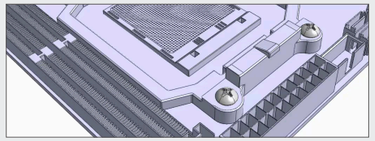
»

Predict, Analyze, Judge, Solve

Until a reasonable solution to the problem is obtained, design modifications and repetitive analysis are required.

»

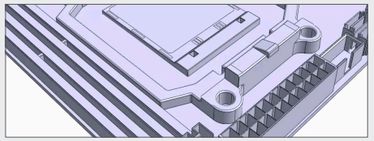
Suggest a solution to the problem



»»»

CAD model simplification and cleanup

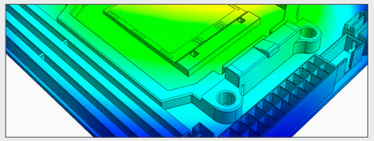
* Manual & time-consuming



»»»

Mesh generation and analysis

* Success rate of automatic mesh generation & quality of mesh



Idealization cleanup mesh generation takes up more than 70%

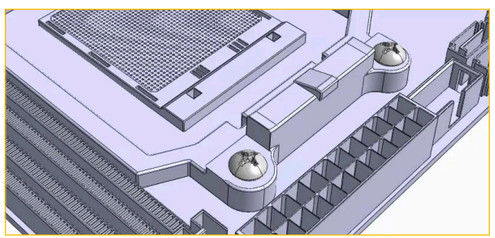
Analyze simulation result takes up less than 30%

Modeling	Idealization of Analysis Model	Cleanup	Material Properties	Define Mesh	Generate Mesh	Load & Boundary Conditions	Simulation	Result Verification

MESHFREE **3-STEP** and **AUTO UPDATE**

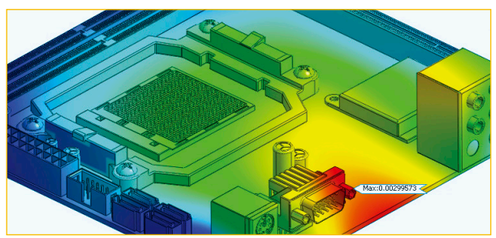
Modeling	Load & Boundary Conditions	Simulation	Result Verification

At least 60% shortening analysis process

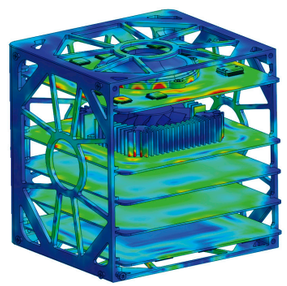


»»»

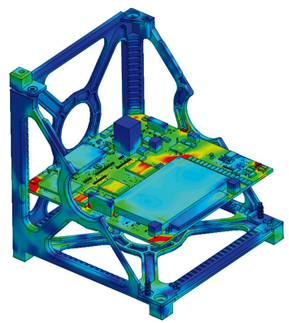
Perform analysis



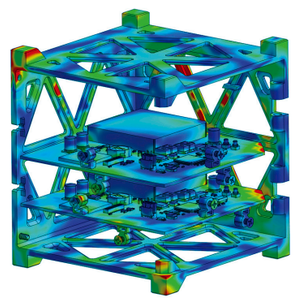
Various design proposals can be reviewed.
Derive the optimal design ensuring safety, usability and economical viability.



Alt #1



Alt #2



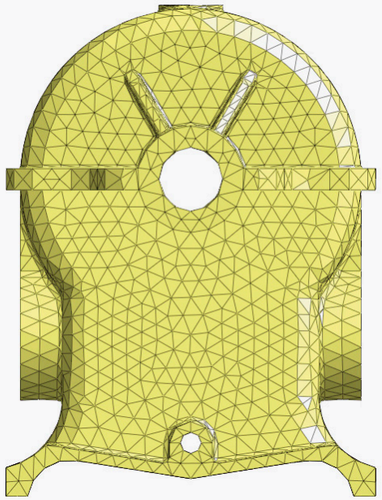
Alt #2

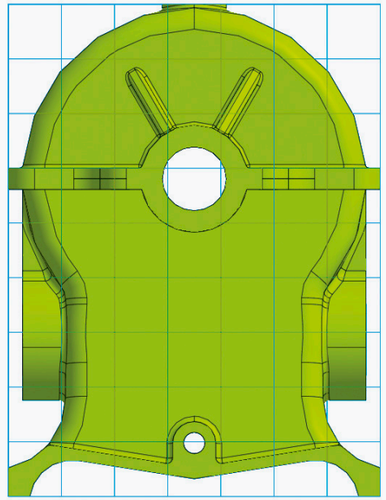
MESHFREE
MAIN FEATURES

To overcome the limitations of existing mesh,
we have successfully developed new and innovative
analysis technology.

midas MeshFree is an innovative CAE program
for engineers in mechanical field.

Mesh Free

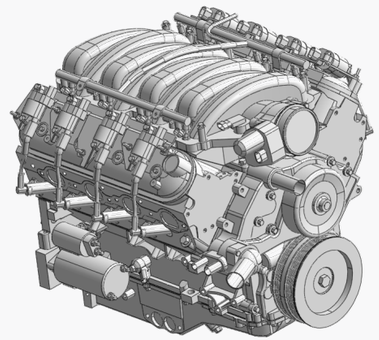


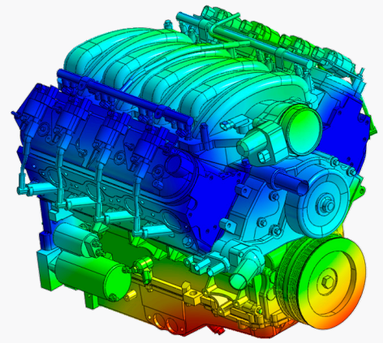


- Generation of mesh depending on the geometry
- Simplification & cleanup for high-quality mesh generation
- Mesh quality problems and limitations of automated processing

- Analysis using simple background grid
- Analysis regardless of the complexity of the geometry
- Elimination of inefficiencies
- Technology developed specifically for design phase analysis

Comprehensive Design Tool





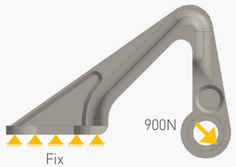
Any **complex geometry** can be analyzed.

Complex models can be simulated easily.

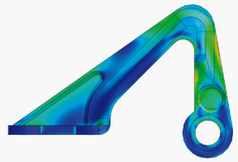
Models with complex geometry or models which were impossible to analyzed using conventional FEA can be analyzed.

Auto-update for Design Modification

Conduct performance verification by automatically considering design modifications that change continuously during the design process



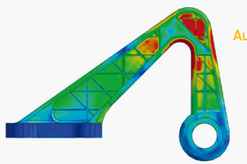
3 - STEP PROCESS



Max. Stress : 152MPa
Weight : 0.31kg

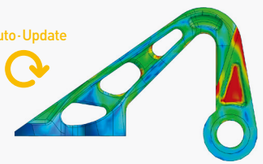
Auto-update

Apply the same analysis condition to the modified model



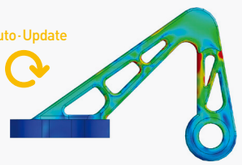
[Alt 1]
Max. Stress : 472MPa
Weight : 0.27kg

Auto-Update



[Alt 2]
Max. Stress : 334MPa
Weight : 0.26kg

Auto-Update

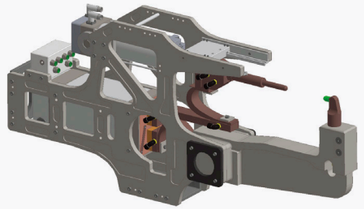


[Alt 3]
Max. Stress : 186MPa
Weight : 0.22kg

Highlights of Auto-update

Automatically reflect the modified model.
Apply the same load & boundary conditions as original design.
Import model and perform analysis at the same time.
3-step process is also omitted to perform analysis of modified model efficiently.

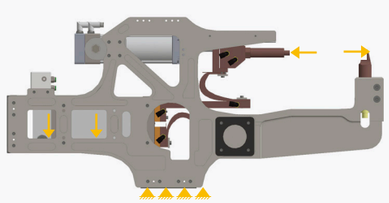
Simple and Intuitive **3-STEP Process**



STEP 1

Import 3D CAD Directly

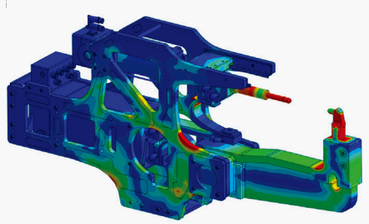
- Supports all commercial CAD programs (Solidworks, Inventor, Catia, NX, Solid Edge, Creo etc.)
- Automatic input of material information defined in CAD
- Define automatic contact between parts of assembly model



STEP 2

Input Load & Boundary Conditions

- Guide to constraint & load conditions according to analysis type
- Assign constraint and load condition to CAD model
- Provide various static, dynamic and thermal loads



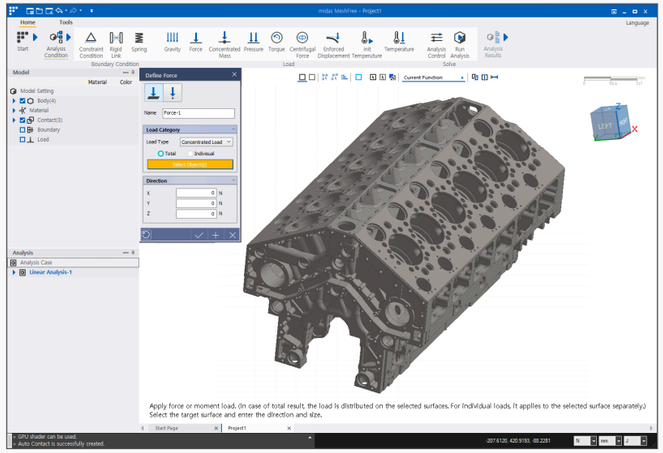
STEP 3

Analysis & Results Checking using the Latest Technology

- Direct analysis using CAD model with latest analysis technology
- Reduction of total analysis time and high success rate of analysis compared to the conventional FEM
- Various post processing function considering user's convenience

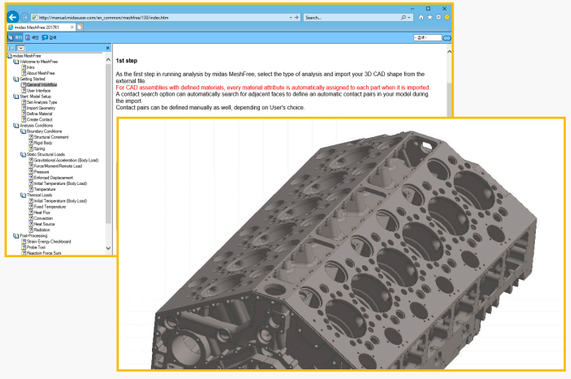
SMART **GUIDE**

Guide to make sure anyone can use it.



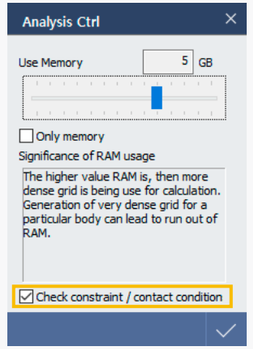
Provides important explanations and instructions of the functions at the bottom of the model window

Connect online help and video tutorials

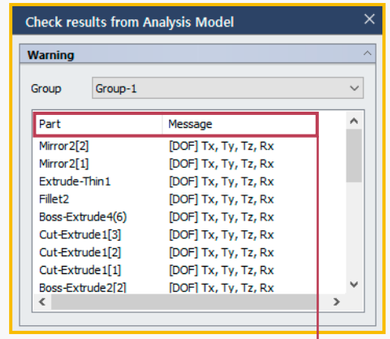


SMART **REVIEW**

Check errors and reliability in advance.



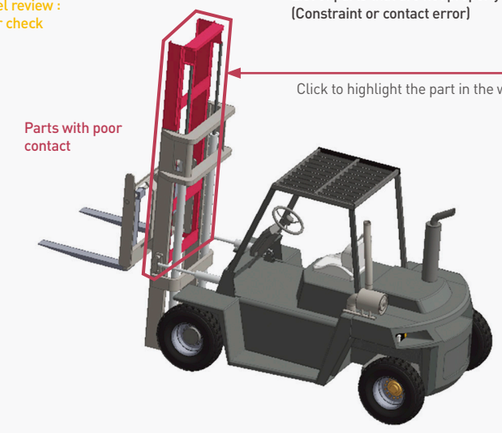
Analysis model review :
Singular error check



List of parts that are not properly constrained
(Constraint or contact error)

Click to highlight the part in the work screen

Parts with poor contact



- Check and guide appropriate constraints which is the most difficult and intrusive part for the beginners
- Useful check of constraints and contact conditions in large assembly models

MESHFREE ANALYSIS FUNCTIONS

MeshFree provides the powerful analysis features required during the design phase and optimal design techniques for efficient design work.

MIDAS

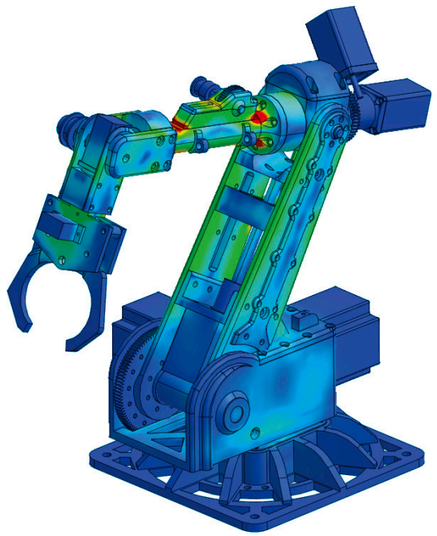
Linear Static Analysis

Analyze large and sophisticated models quickly and accurately.

- Displacement / Stress / Safety Factor Results
- Thermal Deformation / Stress due to Temperature Difference
- Possible to Consider Prestress
- Result Combination using Sub-cases by Load Conditions
- Linear Contact: Welded Behavior, Sliding
- Practical Load / Boundary Conditions

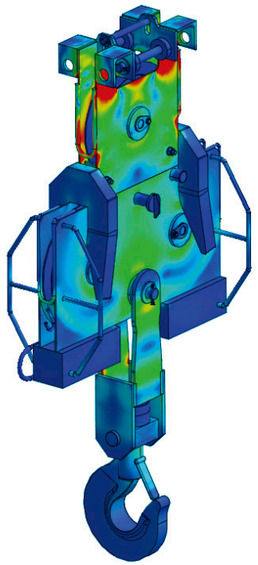
Crane Strength Safety Evaluation

- Structural safety review of applied load at crane operation
- Perform analysis by welded contact and sliding contact to 232 parts



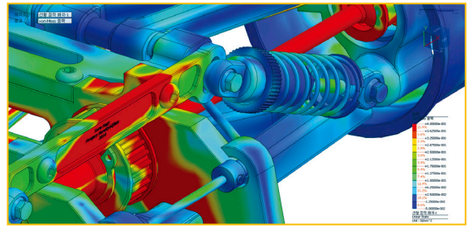
Stiffness Evaluation during the Operation of Robot Arm

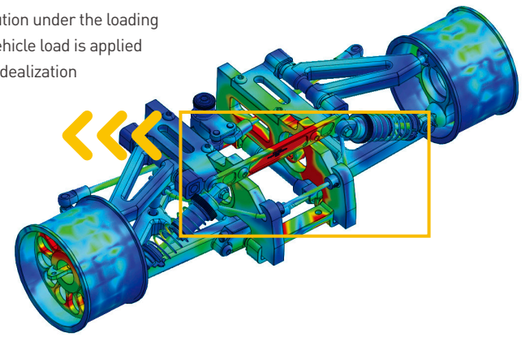
- Structural safety evaluation of robot arm considering various load conditions
- Strength review of major parts of structure

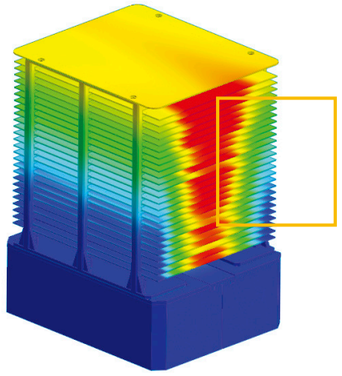


Structural Safety Analysis of Suspension of Vehicle

- Structural safety review through suspension displacement and stress distribution under the loading
- Analysis of maximum displacement and stress component occurring when vehicle load is applied
- Analysis using original CAD model prototype without model simplification or idealization

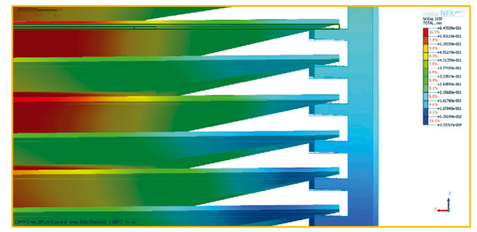






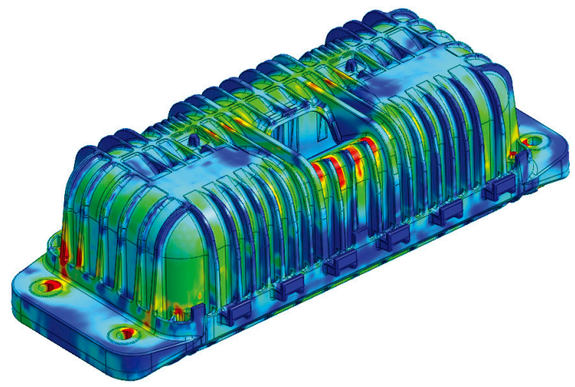
Examination of Glass Deflection by Temperature Load

- Examination of deflection of glass when applying temperature load
- Linear general contact conditions to create the identical environment as the working environment



Evaluation of Housing Stiffness

- Models with complex NURBS patches and sliver faces
- Perform stiffness evaluation using CAD model prototype without simplification process
- Traditional FEM system: 3 days including cleanup and mesh generation
- MeshFree: 35 minutes



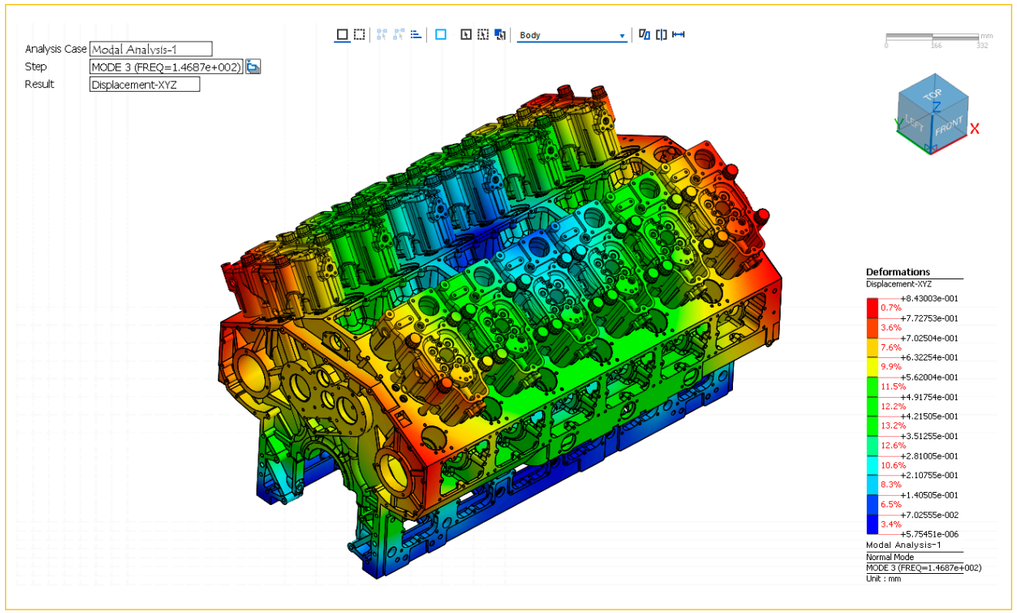
Modal Analysis

Fast eigenvalue analysis of large assembly model is possible.

- Natural Frequencies and Mode Shapes
- Calculation of Modal Participation Rate, Effective Mass and Calculation Error Check
- Strum Sequence Check within the Specified Eigenvalue Range (Check for Missing Eigenvalues)
- Possible to Consider Prestress
- Linear Contact: Welded Behavior, Sliding

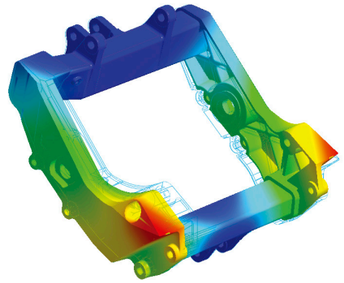
Natural Frequency Analysis of Engine Block

- Modal analysis of engine block consisting of 140 parts
- Perform natural vibration analysis using the design model without idealization of mass and spring elements



Comparison of Natural Frequency of Subframe

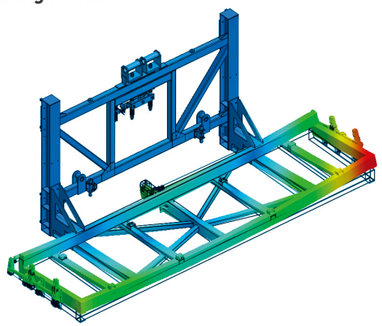
- Review of resonance possibility during the operation of lifting frame through natural frequency analysis

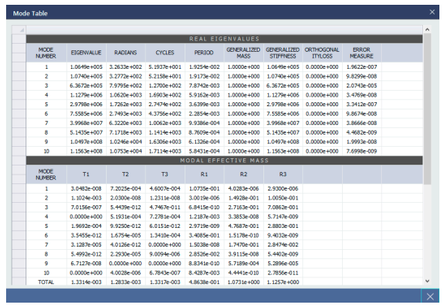


Mode	MeshFree	FEM	ERROR
1st Mode	466.06 Hz	460.41 Hz	1 %
2nd Mode	593.59 Hz	589.25 Hz	1 %
3rd Mode	647.70 Hz	637.27 Hz	2 %
4th Mode	777.91 Hz	765.28 Hz	2 %
5th Mode	1064.02 Hz	1052.14 Hz	1 %
6th Mode	1289.17 Hz	1265.91 Hz	2 %

Dynamic Analysis of Lifting Frame

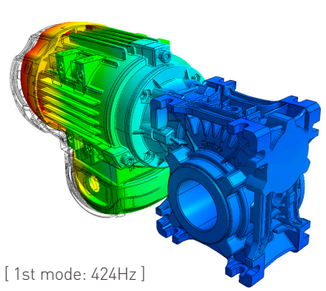
- Review of resonance possibility during the operation of lifting frame through natural frequency analysis



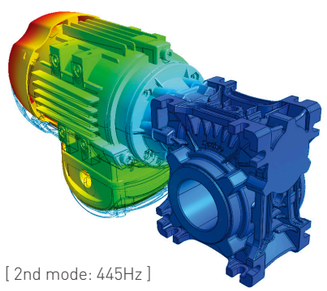


Dynamic Analysis of Gearbox

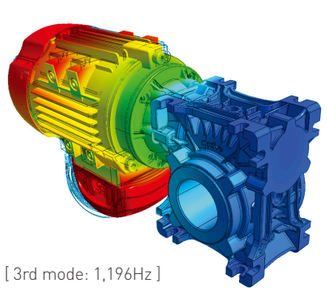
- Generation of vibration in gearbox by motor
- Review and elimination of resonance effect on gearbox due to vibration



[1st mode: 424Hz]



[2nd mode: 445Hz]



[3rd mode: 1,196Hz]

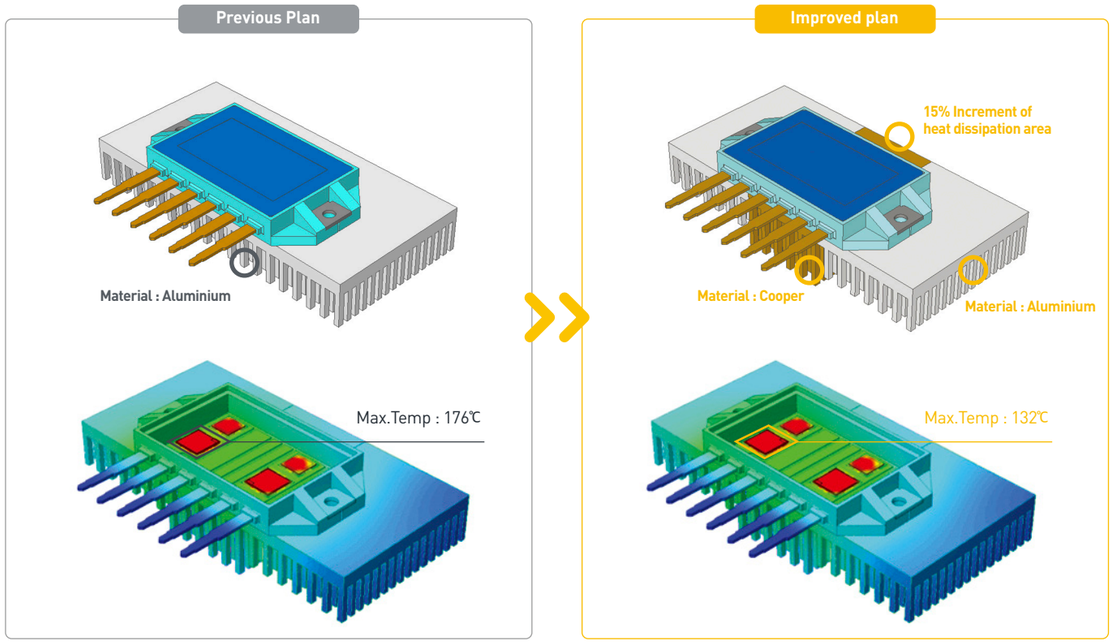
Heat Transfer / Thermal Stress Analysis

Convenient and effective heat transfer / thermal stress analysis is possible by supporting practical conditions.

- Heat Transfer Analysis
- Supports Various Loading Conditions such as Heat Generation, Conduction, Convection, Radiation and Heat Flux
- Thermal Contact for Conduction between Discontinuous Parts

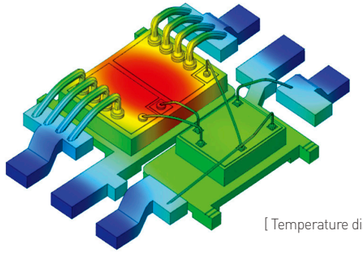
Improvement of Cooling Performance on MOSFET Heatsink

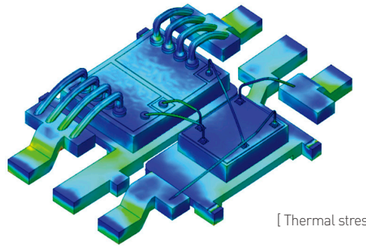
- Improvement of cooling performance by increasing area of heat dissipation
- Improvement of cooling performance by material modification



Safety Verification of Chip Through Displacement / Stress Analysis due to Heat Generation

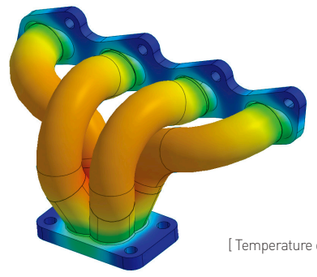
- Review of chip temperature distribution by heat generation
- Verification of design safety through thermal stress analysis at maximum temperature

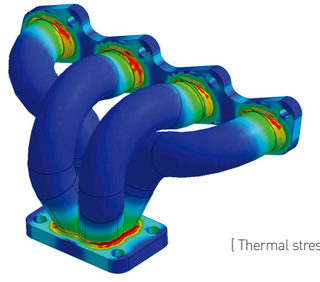

[Temperature distribution]


[Thermal stress]

Analysis of Temperature Distribution and Thermal Stress of Manifold for Automobile

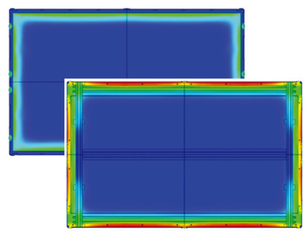
- Examination of temperature distribution of manifold by gas emission
- Safety verification through analysis of manifold deformation and stress by thermal expansion

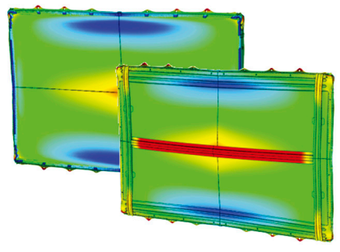

[Temperature distribution]

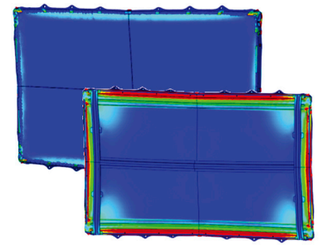

[Thermal stress]

Heat Transfer / Thermal Stress Analysis of LCD TV

- Examination of temperature distribution of whole BLU caused by heat of PKG
- Examination of product safety through thermal stress analysis


[Temperature results of heat transfer analysis]


[Displacement result]


[Stress result]

Fatigue Analysis

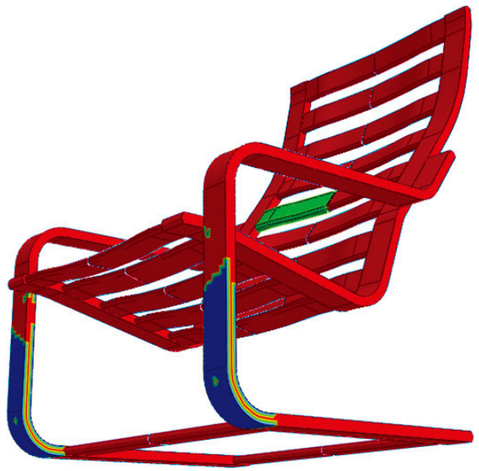
Durability check is available as independent function.

- Damage, Fatigue Life Results
- Rainflow Counting, Mean Stress Correction Options
- Select Evaluation Stress (Signed Von-mises, Absolute Maximum Principal Stress)
- Linear S-N Curve Function

Review Fatigue Life of Chair

- Examination of stress distribution when the chair is occupied
- Examination of endurance life expectancy using S-N function

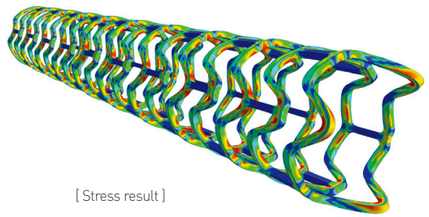
[Stress result]



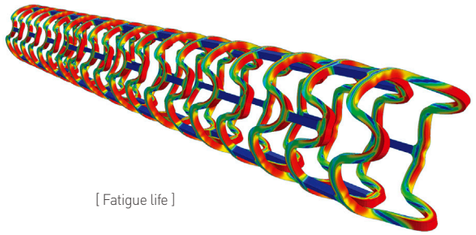
[Endurance life]

Endurance Life Analysis of Stent for Cardiovascular

- Review on stress distribution for pressure loads repeatedly acting on the stent
- Examination of stent life span using S-N function



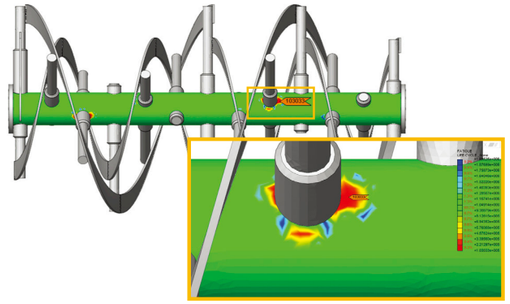
[Stress result]



[Fatigue life]

Fatigue Durability of Blender Structures

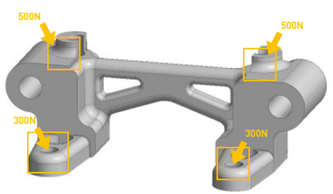
- Fatigue safety examination of blender in operation
- Check fatigue durability after reviewing Static Analysis for complex geometry

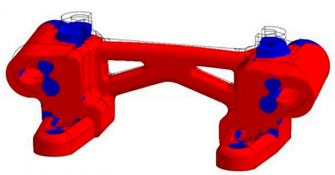


Mean Stress Correction	Model-1		Model-2		Model-3	
	No. of Life Cycle		No. of Life Cycle		No. of Life Cycle	
	Max	Min	Max	Min	Max	Min
Case-1	1.00E+6	1.00E+6	1.00E+6	9.97E+5	1.00E+6	1.00E+6
Case-2	1.83E+6	8.38E+5	1.88E+6	2.99E+5	1.95E+6	3.80E+5
Case-3	1.96E+6	2.01E+5	1.99E+6	1.03E+5	1.97E+6	1.32E+5

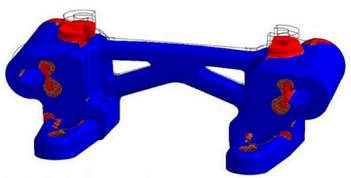
Fatigue Life Prediction of Bracket

- Stress and fatigue life analysis due to load applied to each part of bracket
- Stress analysis of bracket contact area according to load
- Review on fatigue life according to load





[Fatigue life result due to load]



[Damage result due to load]

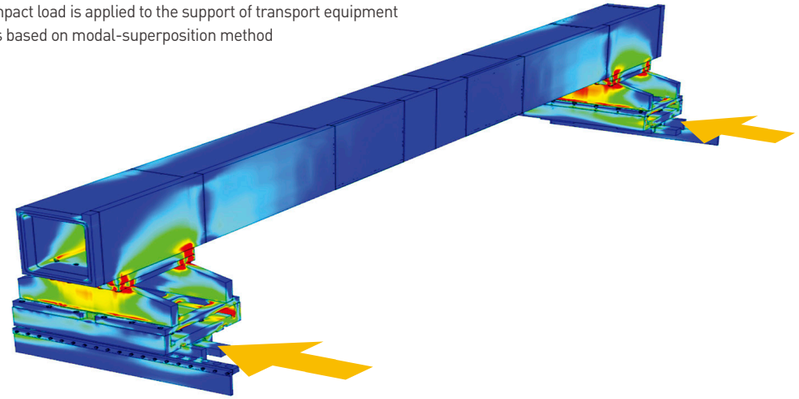
Linear Dynamic Analysis

Using the direct method and modal method, it is possible to perform the analysis considering reliability and efficiency.

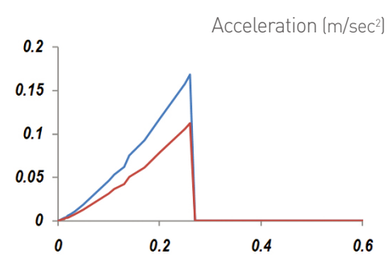
- Direct Method and Modal Method Analysis
- Transient Response Analysis
- Frequency Response Analysis
- Random Vibration Analysis
- Response Spectrum Analysis
- Various Damping Effects (Modal / Structure, Frequency Dependent)
- Design Spectrum Database for International Standards

Impact Response Analysis of Transport Equipment

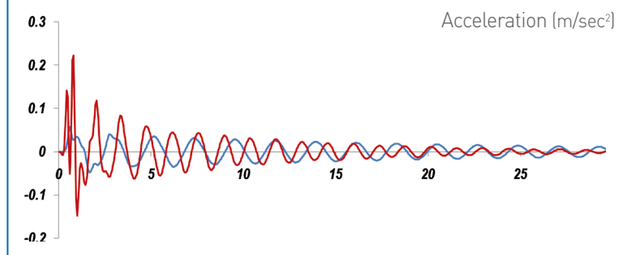
- Safety examination when impact load is applied to the support of transport equipment
- Transient response analysis based on modal-superposition method



Impact load

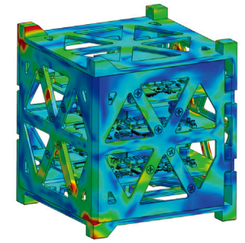


Impact response

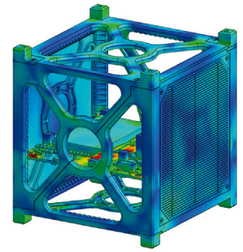


Cube-type Satellite Random Vibration Analysis

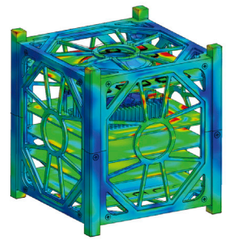
- Random vibration safety check for various types of cube-type satellites
- 3-sigma RMS stress review for each direction



[Type - 1]



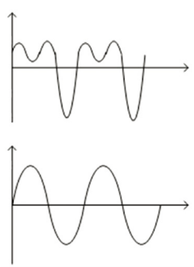
[Type - 2]



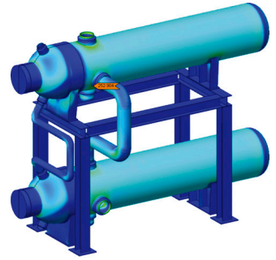
[Type - 3]

Safety Evaluation of Plant Structure by Vibration Load

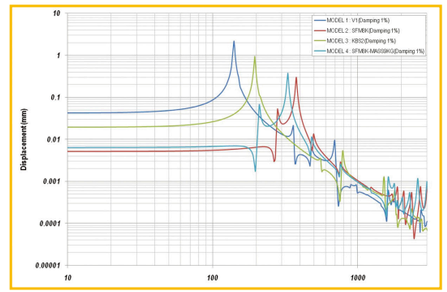
- Input the harmonic load by analyzing the modal component
- Analyze frequency response to check whether there is crack in the structure and piping



[input the harmonic load]

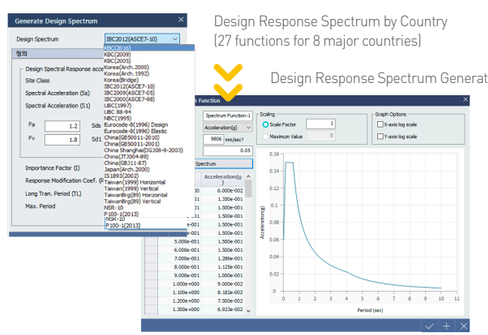


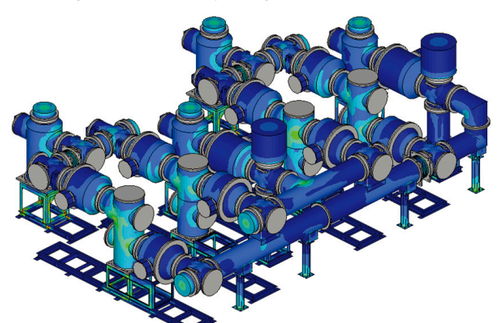
[Stress distribution in case of resonance]



Seismic Analysis of Gas Insulated Switchgear

- Seismic verification of gas insulated switchgear installed in earthquake-prone area
- Safety verification considering seismic load and operating load





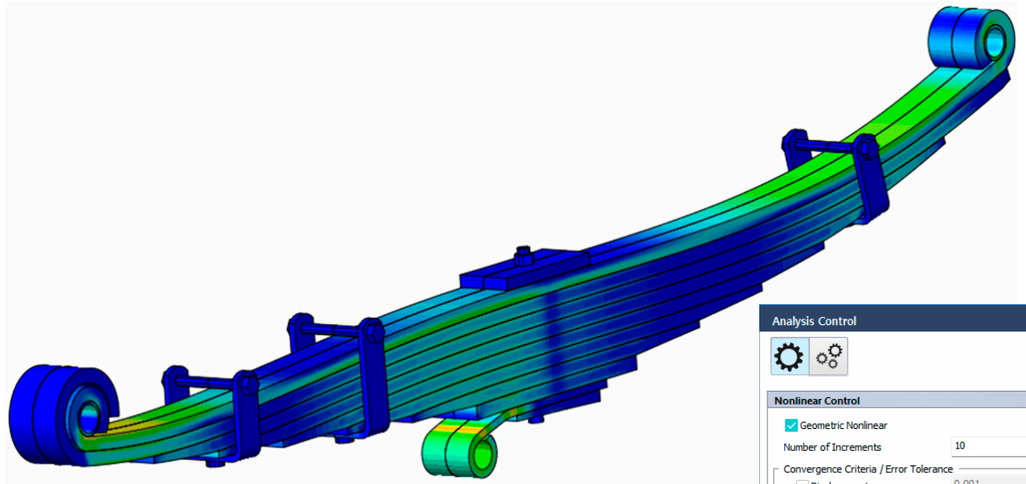
Nonlinear Static Analysis

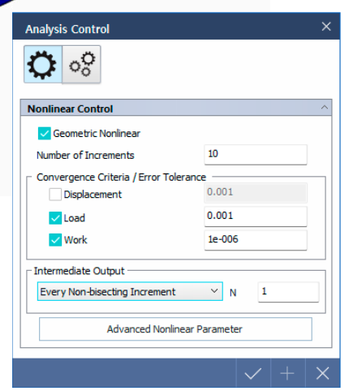
Support various iterative methods, stiffness update method and convergence criterion method.

- **Material nonlinearity**
  - Elasto plastic model
  - Hyper elastic model
- **Geometric nonlinearity**
  - Large deformation, Large rotation support
  - Following force support
- **Contact nonlinearity**
  - General contact support

Analysis of plate spring strength and rigidity

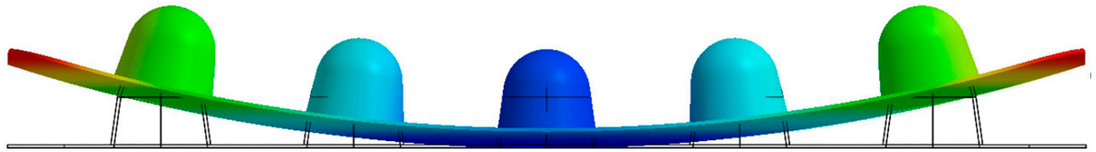
- Geometric nonlinear analysis considering bi-directional sliding contact





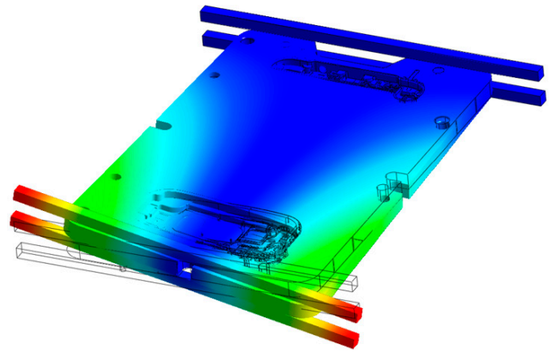
Medicine case in heat chamber

- Thermal deformation according to temperature distribution
- Deformation analysis considering general contact condition and large deformation



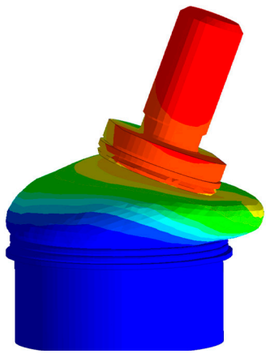
Torsion analysis of electronic components

- Definition of general contact considering deformation due to torsion
- Applied geometric nonlinear considering large deformation and rotation



Performance analysis of rubber materials

- Application of material nonlinearity using hyper elastic materials
- Application of geometric nonlinearity considering large deformation
- Linear contact condition applied



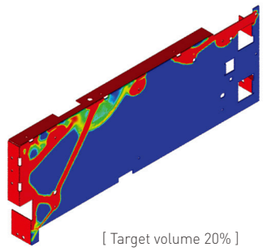
Topology Optimization

Static / mode analysis and phase optimization analysis considering manufacturing process is possible.

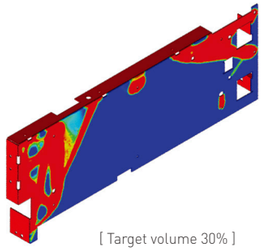
- **Optimized Analysis Function with Static Analysis and Dynamic Analysis**
  - Linear Static Analysis
  - Modal Analysis
- **Analysis Function Considering Manufacturing Process Conditions**
  -Design Limitation / Constraint Settings such as Stress, Displacement, Volume, Draw Direction and Symmetric Condition
- **Simultaneous Optimization Analysis considering Various Operating Conditions / Load Conditions**

Optimization of Bracket Geometry to Maintain Rigidity

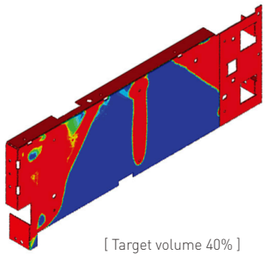
- Optimal design that can maintain the stiffness of the bracket applied to the product keeping it as light as possible
- Optimize the desired volume to be used (up to 40% reduction in target volume)



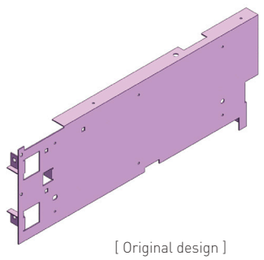
[Target volume 20%]



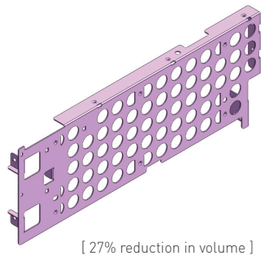
[Target volume 30%]



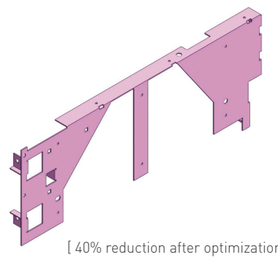
[Target volume 40%]



[Original design]



[27% reduction in volume]

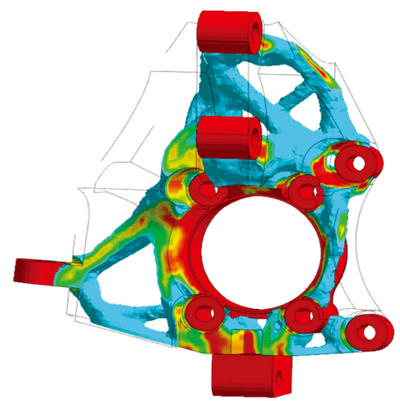


[40% reduction after optimization]

Optimized Shape of Automobile Knuckle with Multiple Loads

- Optimal design of vehicle knuckle with multiple loading conditions
- The volume change is small, but the performance is improved by 39% compared to the existing one.

[Original model]

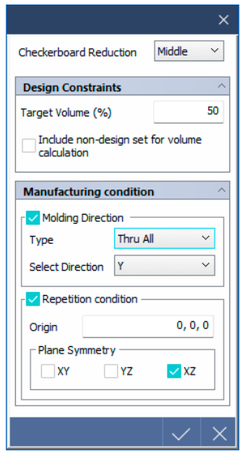


[Topology optimization design result]

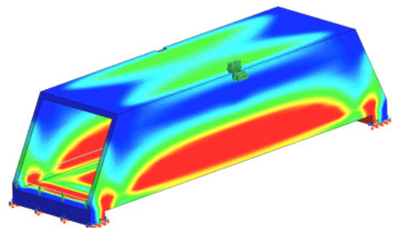
[Optimized model]

23% weight reduction

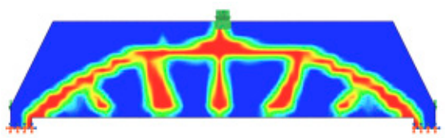
Example of optimized shape output considering product manufacturing process



- Shape layout output using manufacturing conditions (symmetry)



[Initial optimization analysis result]



[Optimization analysis result considering manufacturing conditions]

Reliability of Results

NAFEMS reference results for unit model, comparison of FEM analysis results for practical model

NAFEMS
Theoretical Value and Verification

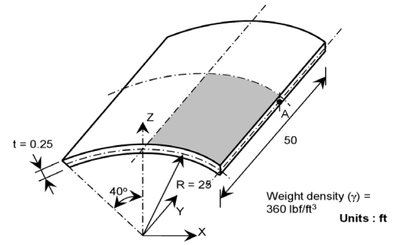
Comparison with **FEM Results**

Comparison with **Practical Model**

NAFEMS
Theoretical Value and Verification

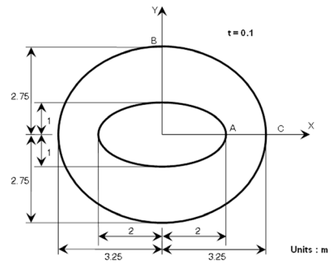
Scordellis-Lo barrel vault (gravity load)

	Vertical displacement at point A [ft]
Reference	- 0.3024
MeshFree	- 0.3025
% Difference (MeshFree/Theory)	0.04%



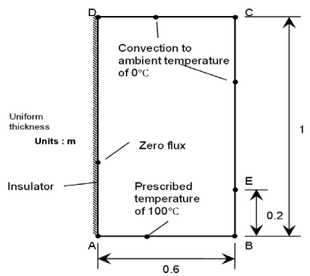
Elliptic membrane under uniform outward pressure

	Stress_yy at point A
Reference	92.7 MPa
MeshFree	92.6 MPa
% Difference (MeshFree/Theory)	0.11%

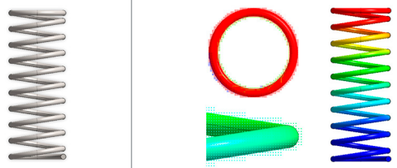
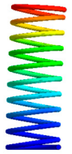


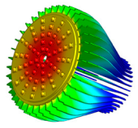
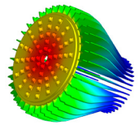
Two-dimensional heat transfer with convection

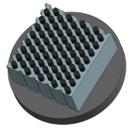
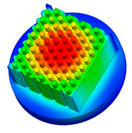
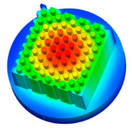
	Temperature at point E
Reference	18.3
MeshFree	18.1
% Difference (MeshFree/Theory)	1.10%



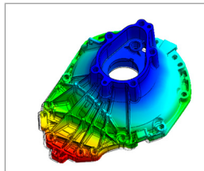
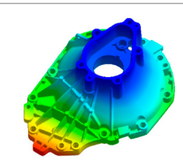
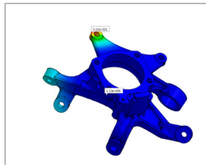
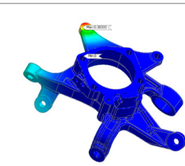
Comparison with **FEM Results**

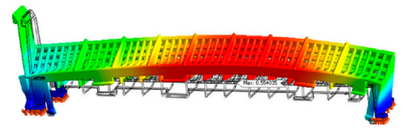
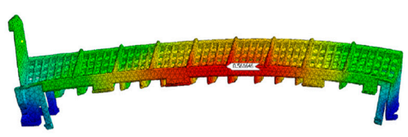
Model	MeshFree		FEM	
				
Results	1st Mode	24.8 Hz	1st Mode	24.8 Hz
	2nd Mode	24.9 Hz	2nd Mode	25.0 Hz
	Max. displacement	0.0733 mm	Max. displacement	0.0730 mm
	Max. stress	2.86 MPa	Max. stress	2.90 MPa

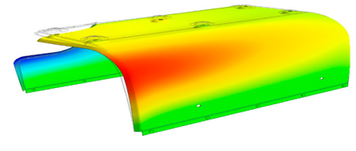
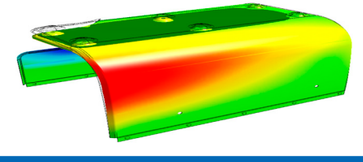
Model	MeshFree		FEM	
				
Results	Max. temperature	144 (°C)	Max. temperature	142 (°C)

Model	MeshFree		FEM	
				
Results	Max. temperature	42.7 (°C)	Max. temperature	43.7 (°C)

Comparison with **Practical Model**

MeshFree	FEM	MeshFree	FEM
			
0.197 mm	0.197 mm	0.3016 mm	0.3033 mm

MeshFree	FEM
	
0.55 mm	0.56 mm

MeshFree	FEM
	
- 20.0 mm	- 21.0 mm

MESH FREE Specification

Details	
Linear Static Analysis	Linear Static Analysis
	Modal Analysis
	Prestressed Modal Analysis
Heat Analysis	Steady Heat Analysis
	Heat Stress Analysis
Optimization	Topology Optimization Analysis
Fatigue Analysis	S-N curve (Stress-life Method)
Transient Heat Analysis	Temperature-dependent Material
Linear Dynamic Analysis	Transient Response Analysis
	Response Spectrum Analysis
	Frequency Response Analysis
	Random Vibration Analysis
Nonlinear Static Analysis	Material Nonlinear Analysis
	Geometry Nonlinear Analysis
	Contact Nonlinear Analysis

MIDAS
MESH FREE

MIDAS

nfx.midasuser.com