MIDAS NFX CFD

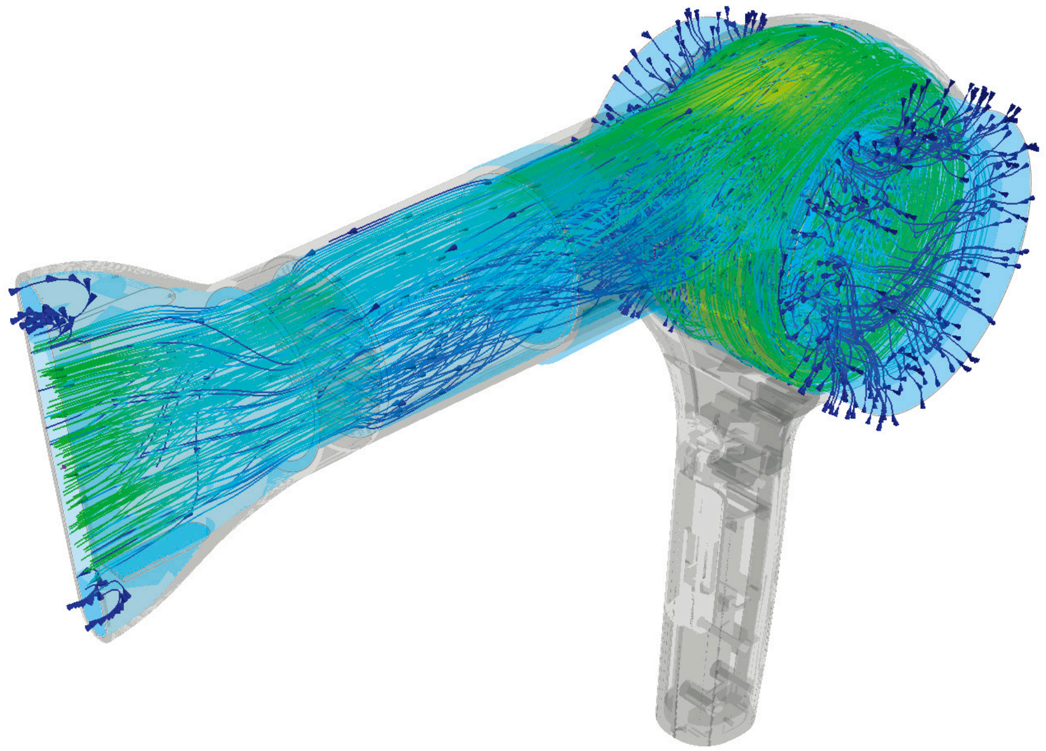
TOTAL SOLUTIONS FOR TRUE ANALYSIS-DRIVEN DESIGN

MIDAS NFX CFD

MIDAS

MIDAS NFX CFD

TOTAL ANALYSIS SOLUTIONS FOR OPTIMUM DESIGN IN MULTI-DISCIPLINES



midas NFX provides a finite element based CFD analysis function, which allows all fluid analyses in the flow velocity domain, various heat transfer analyses and multi-phase analysis.

A single work environment combines both structural and fluid analyses in the same geometric analysis model.

MIDAS

Tailored Work Environment
for Design Practitioners

MIDAS NFX CFD

PRE-PROCESS

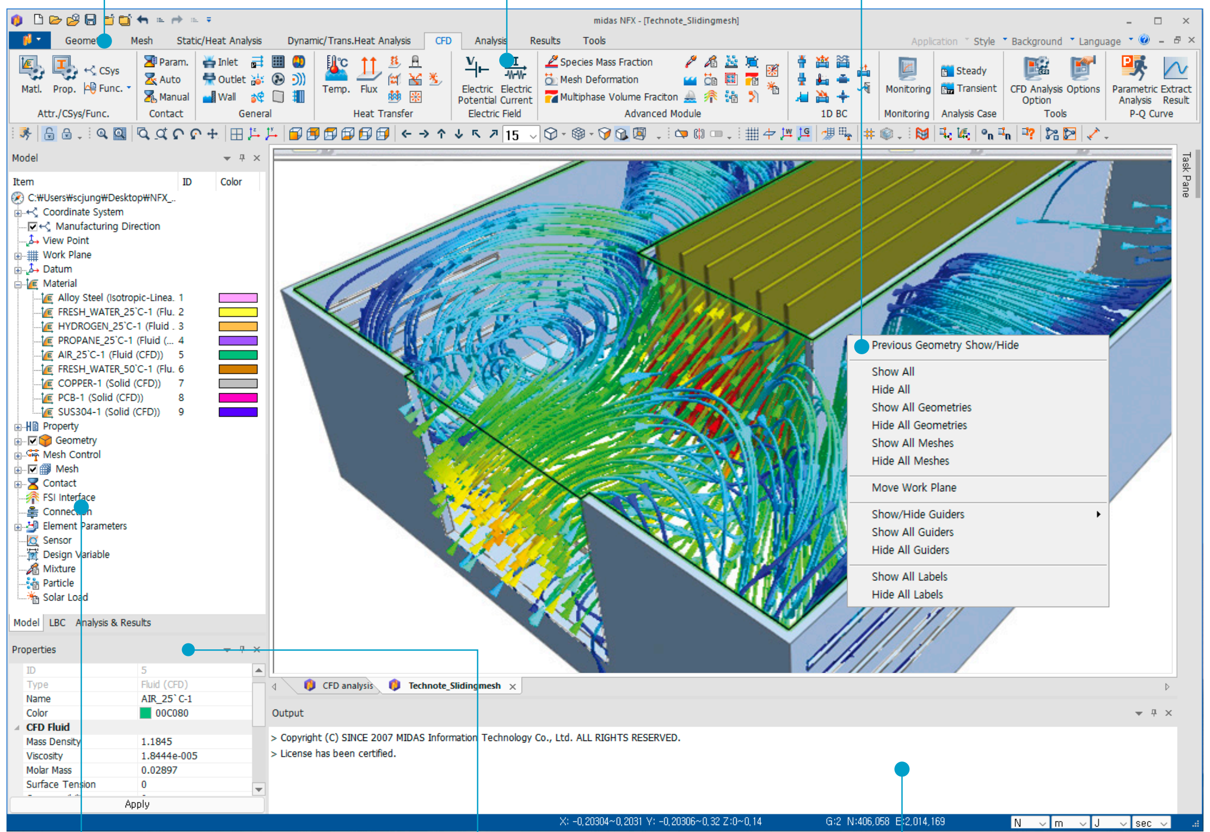
Effective and practical Work Environment

Overview : Graphic User interface

Ribbon Menu
Related functions are grouped together in one panel

Tabbed Toolbar
Provides categorized toolbar icons

Context Menu
Provides frequently used functions depending on the selected entities.



Work Tree
Presents model data in an intuitive way. Data can be directly managed from the tree menu.

Properties Window
Review and edit values of the selected item

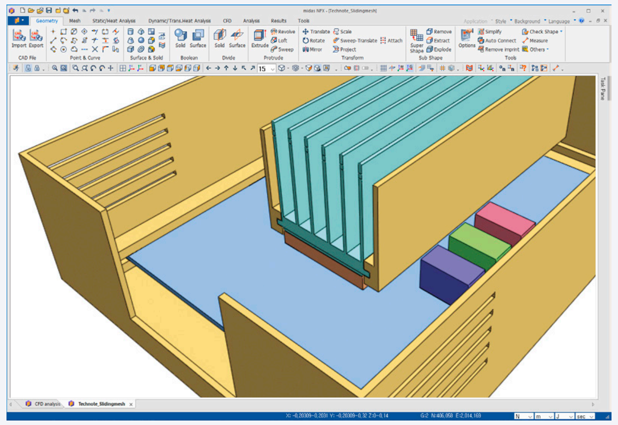
Message Window
Provides useful feedback information during work.

PRE-PROCESS

Integrated work environment for high efficiency

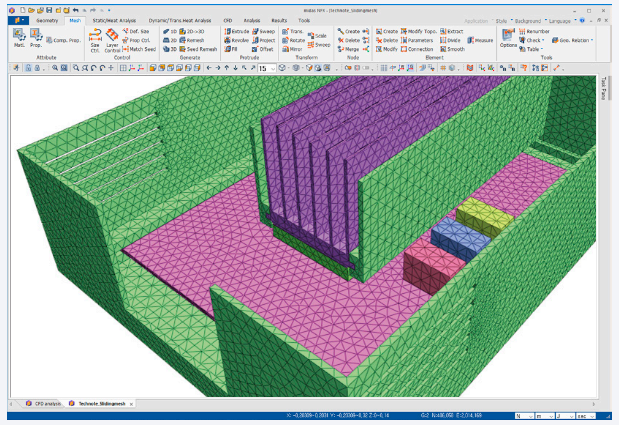
Pre-process CAD Modeling

Provide basic CAD functions for model preparation : CAD creation, modification, auto-simplification



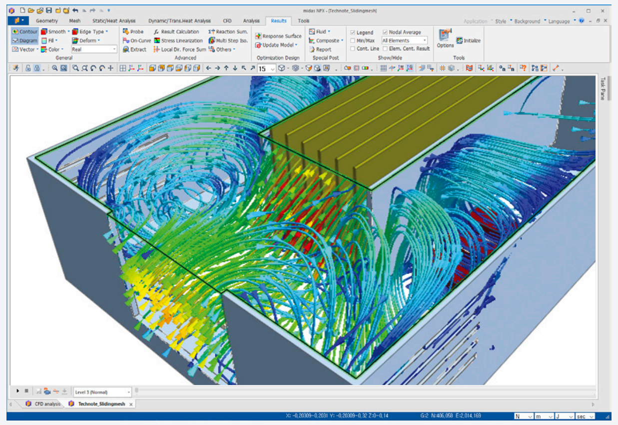
Mesh generation

Fast mesh generation and easy mesh checking in one unique interface.



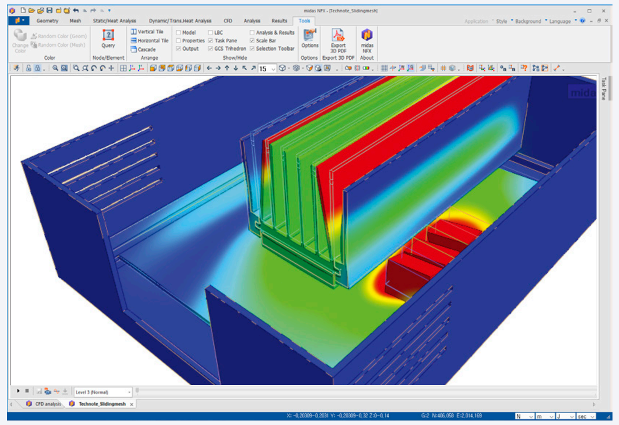
CFD Analysis

From CFD calculation to result output, analysis in one unique program.



Coupled with thermal, structural analysis

Various analysis types supported in one unique operation environment, including static, dynamic, thermal stress, optimization, fatigue analysis, etc.



Intuitive Modeling
Automation Feature

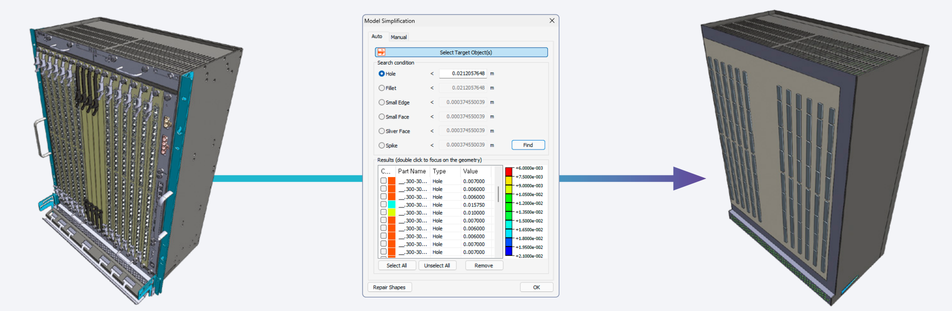
MIDAS NFX CFD

GRAPHIC USER INTERFACE (GUI)

CAD modeling capacities for CFD analysis

CAD modeling capacities for CFD analysis

Automatically delete unnecessary parts and clean up the model for CFD analysis.



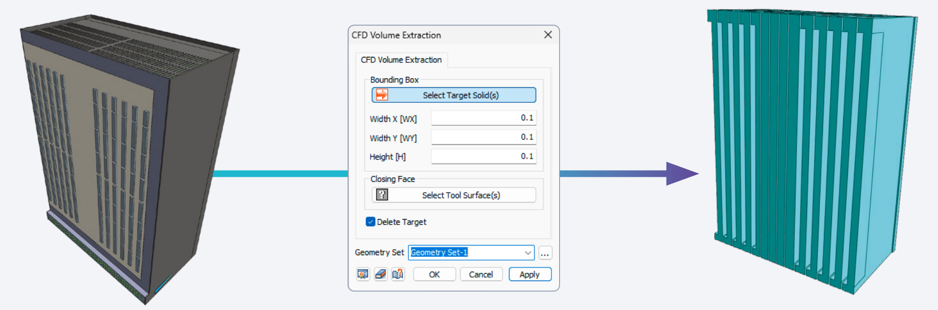
Original design CAD for mobile control gateway Automatic

Automatic simplification window

Simplified CAD model for CFD analysis

Automatic fluid volume extraction

Necessary fluid volume for CFD analysis can be automatically extracted from design CAD model.



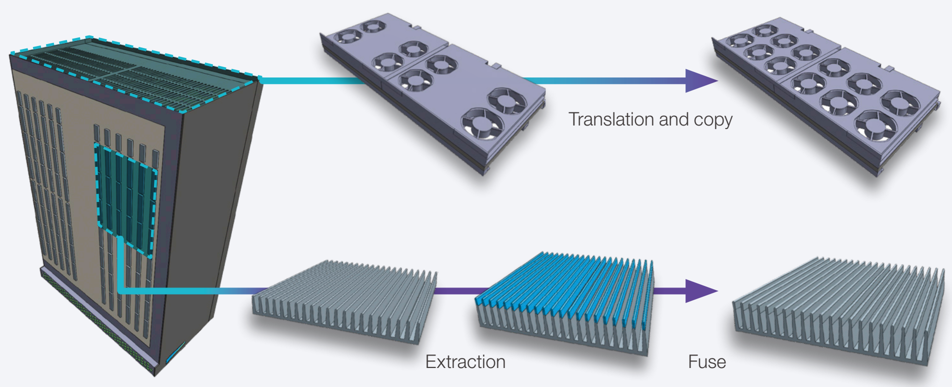
Structure part CAD model including outer shell

CFD area automatic extraction window

Fluid area inside outer shell

Basic CAD modeling for CFD analysis

Design can be modified directly using integrated CAD operations.

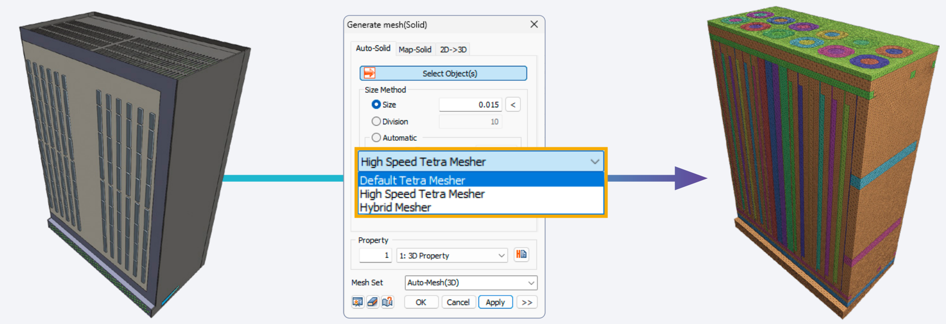


GRAPHIC USER INTERFACE (GUI)

Automatic Mesh Generation

Automatic mesh generation

Auto-meshing function is provided to generate high quality meshes with minimum effort.



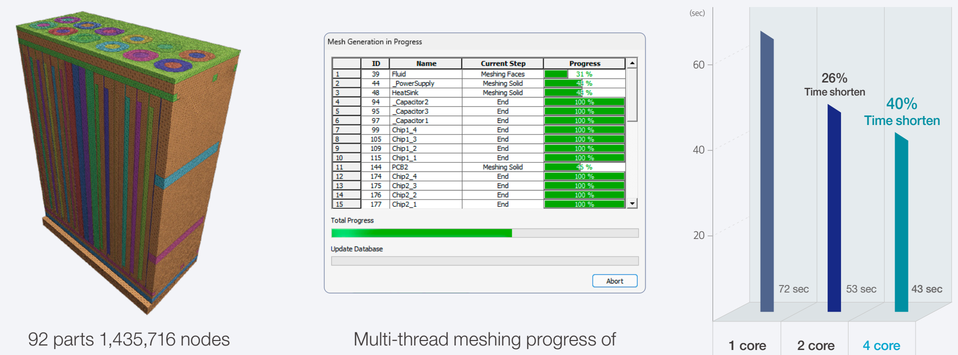
CAD model

High performance auto-meshing

Meshed model

Speed-up mesh generation by parallel processing

When meshing complicated geometries, multi-cores can be used to save mesh generation time.

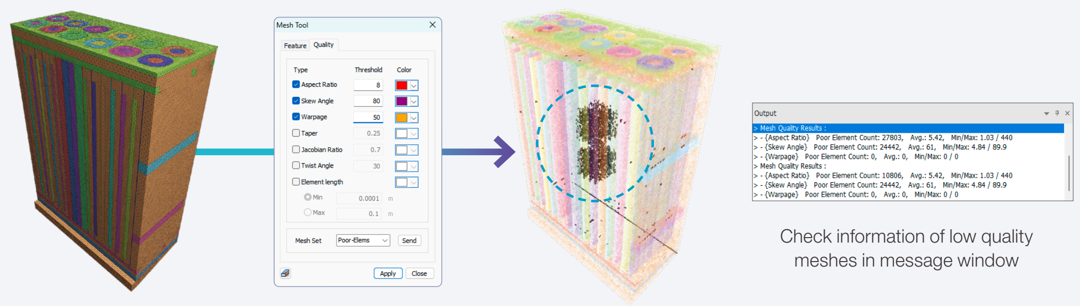


92 parts 1,435,716 nodes
8,264,493 elements

Multi-thread meshing progress of multi-core CPU can be checked

Automatic mesh quality check function

Automatical quality check of generated meshes.



Generated meshes

Mesh quality auto-check types

Visualization of low quality meshes

Check information of low quality meshes in message window

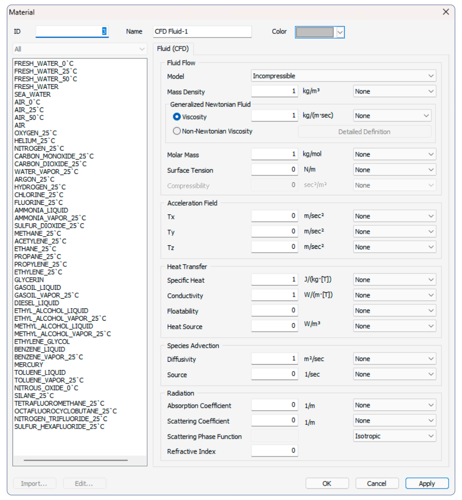
Database and Repetition Automation Features
for Minimizing Manual Work

MIDAS NFX CFD

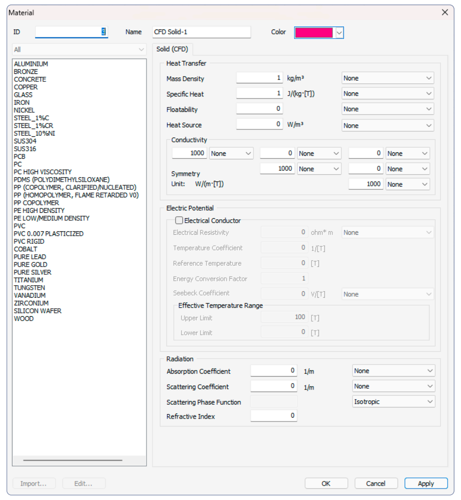
GRAPHIC USER INTERFACE (GUI)

Material properties database

Automatic input for necessary material properties such as density, viscosity, conductivity and specific heat.



Fluid materials



Structure materials

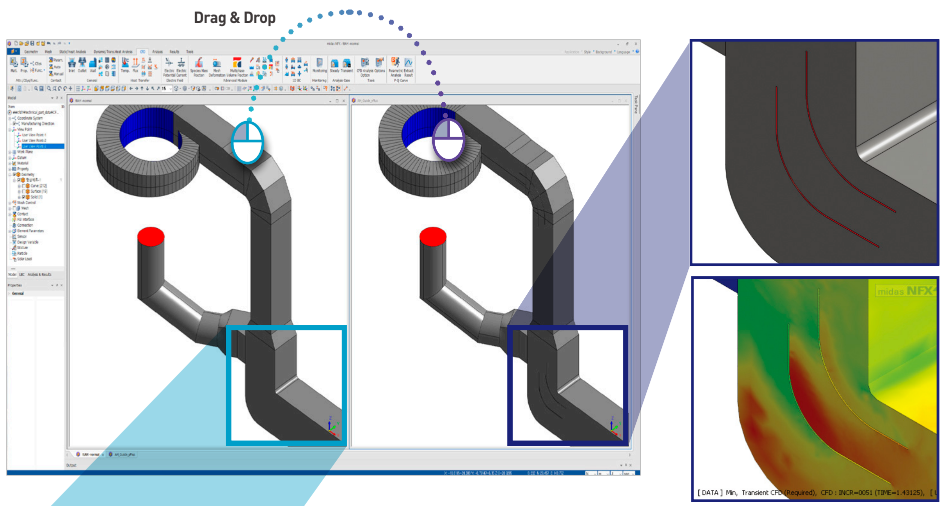
PRE-PROCESS

Automatic copy function for analysis conditions

Repetitive condition input can be done automatically

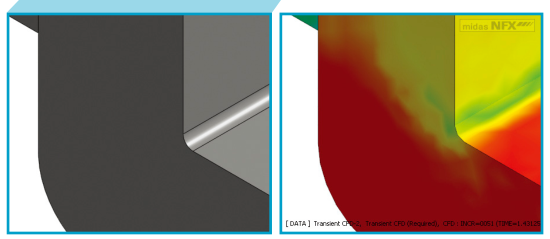
Analysis condition can be copied according to geometry colors

- Blue surface | Speed condition
- Red surface | Pressure condition



Case2 | With guide vanes

Copy the same boundary condition to different geometrical models in 2 analysis cases



Case1 | Without guide vanes

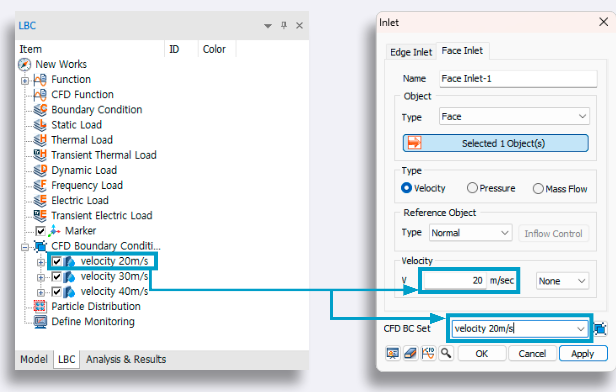
GRAPHIC USER INTERFACE (GUI)

Analysis case management: duplicate and reservation

Analysis cases can be easily duplicated, modified and calculated at the same time.

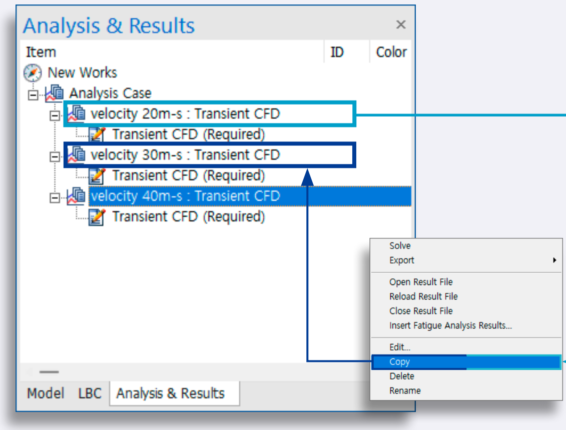
STEP 1

Boundary condition input



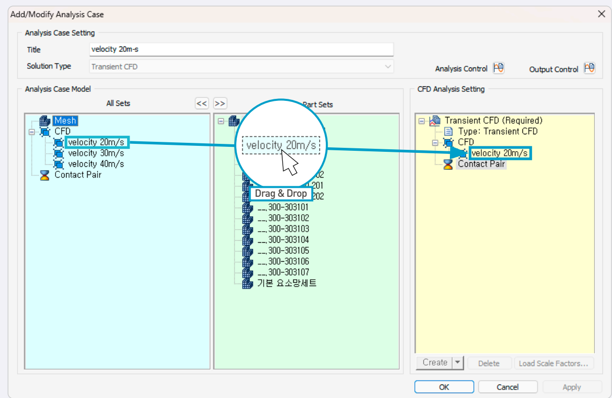
STEP 2

Duplicate existing analysis case



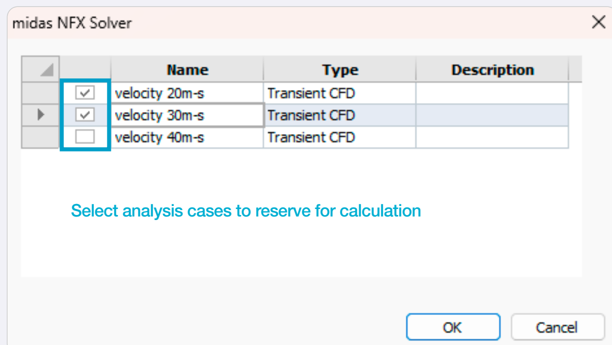
STEP 3

Select a combination of boundary conditions for new analysis case



STEP 4

Reservation for analysis case calculation



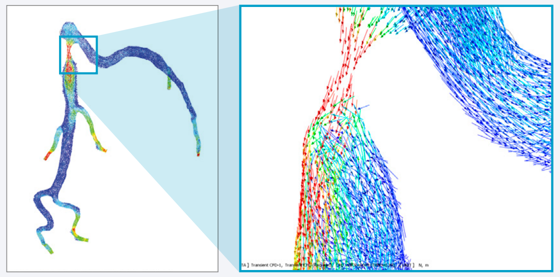
Providing analysis Scalability through Various Turbulence Models and Customizable Functions

SOLVER

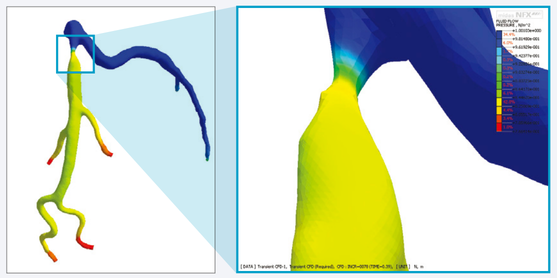
Accurate velocity analysis through 14 types of turbulent models

Variety of turbulent models are provided such as k-ε, k-ωSST, LES and DNS.
Combining results from different models is possible.

Analysis of intravascular blood flow through k-ε composite model

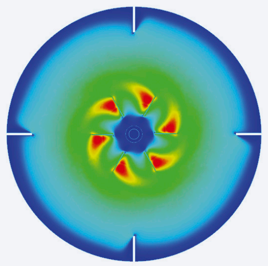


Evaluation of flow lines
at stenostomia place of the vessel

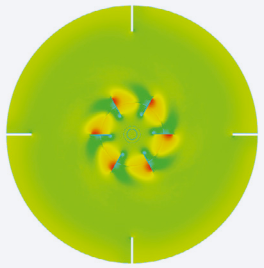


Evaluation of hydraulic pressure
at stenostomia place of the vessel

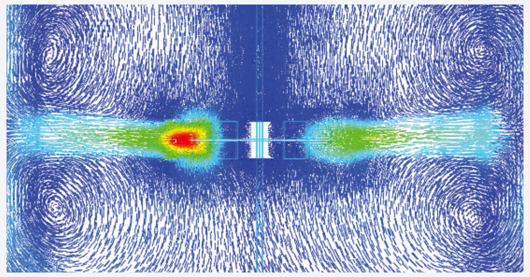
Analysis of agitator´s rotation through k-ωSST model



Velocity distribution
at cross section



Pressure distribution
at cross section



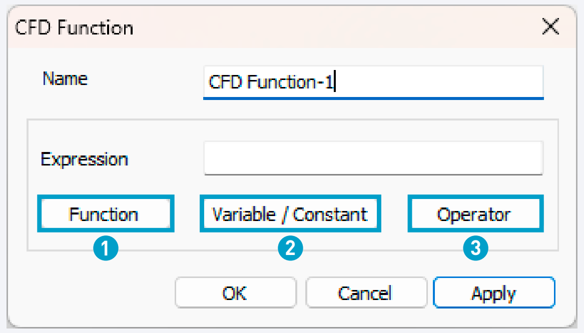
Side velocity vectors

SOLVER

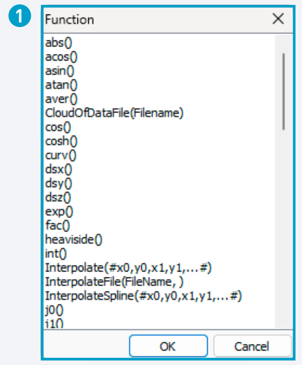
Easy application of variety of boundary conditions through CFD functions

You can input a numerical expression instead of final value into the input box,
the value can be calculated automatically by the input box.

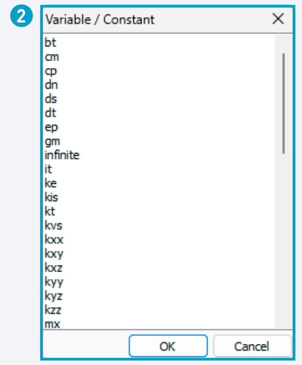
Function input window



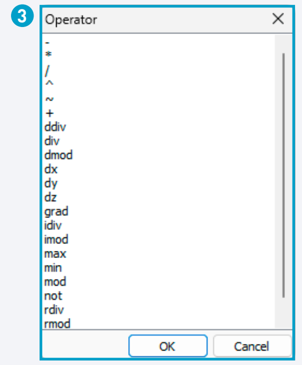
Function input window
(similar to function input method in excel)



Select mathematic function

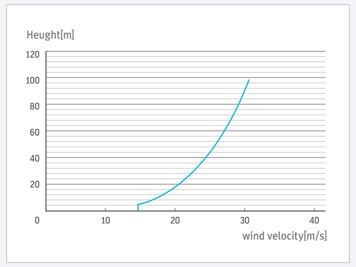


Select parameters such as
velocity, pressure, temperature

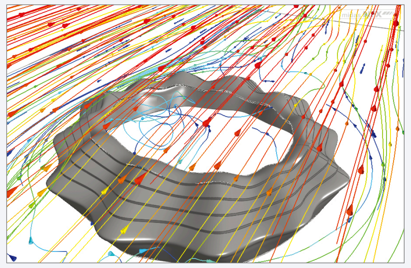


Basic Arithmetic Operators

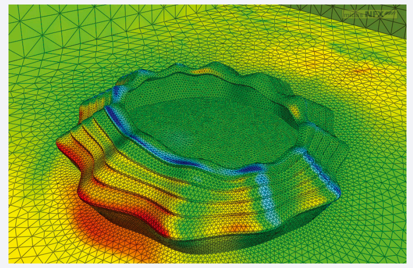
Wind speed input according to height



Wind speed data according to height



Visualization of flow lines



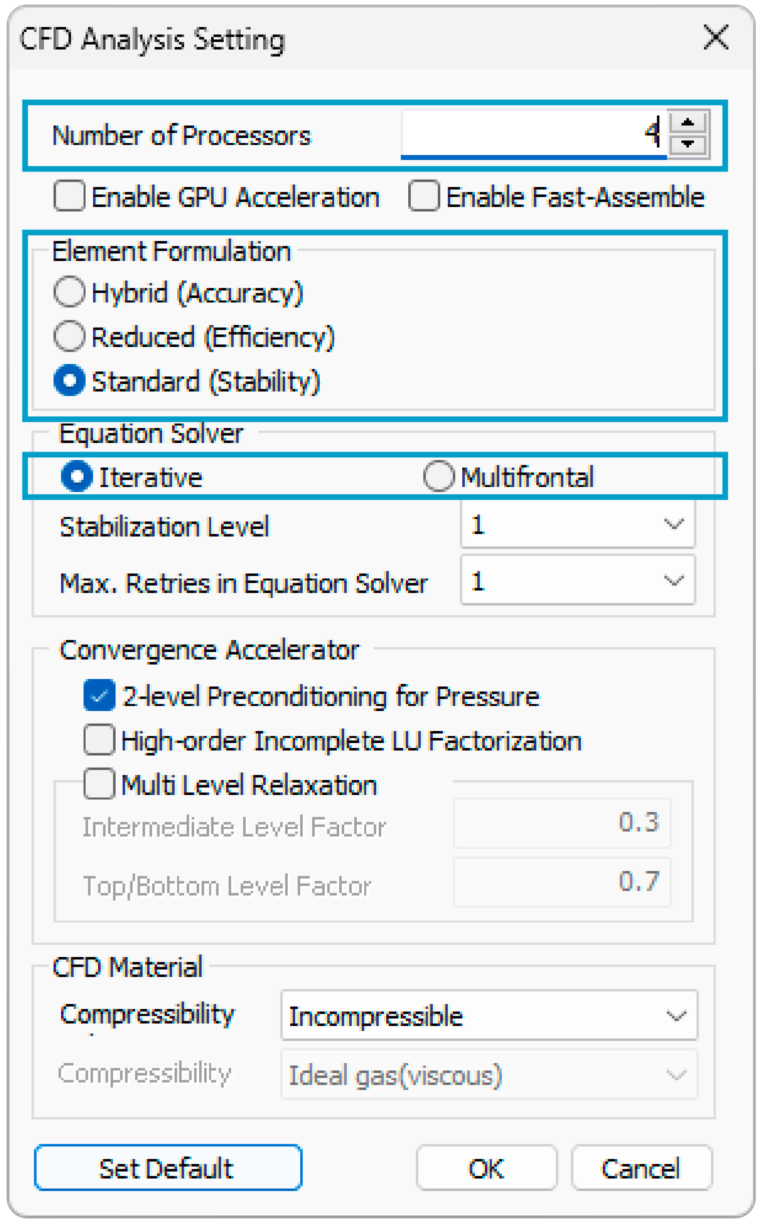
Pressure distribution

Optimal CFD with Parallel Computing and Stabilization Algorithms Applied

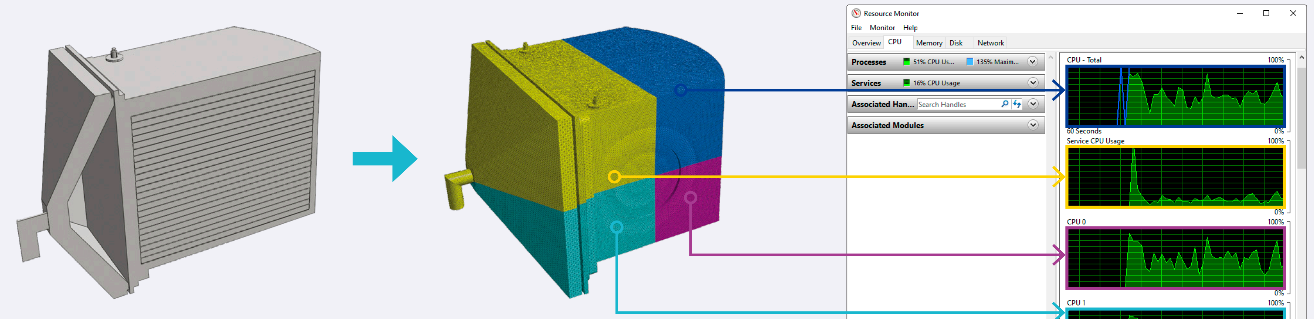
SOLVER

Best CFD Solver

NFX multi-thread Solver is optimized with maximum stability to obtain accurate results for complex design model even in case of rough modeling



1 High speed calculation by using multiple CPUs in parallel



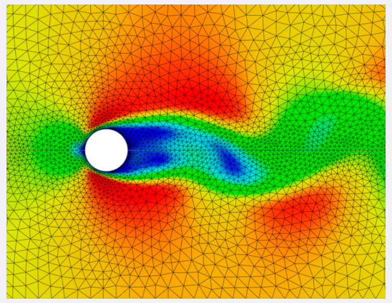
Heat flow analysis model of semiconductor part

Analysis area division based on number of CPUs

2 Automatic setting of meshing algorithm

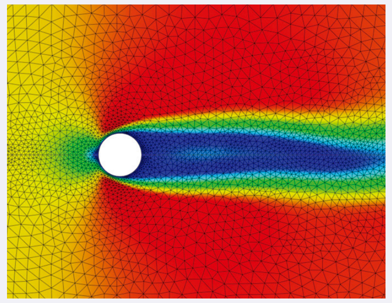
1. Hybrid option

Accuracy based algorithm is used when result accuracy is more important than computation speed.



2. Standard option

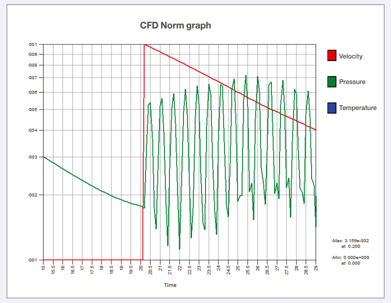
Stability algorithm is used when computing speed is more important than result accuracy.



3 Automatic solver setting for beginner level users

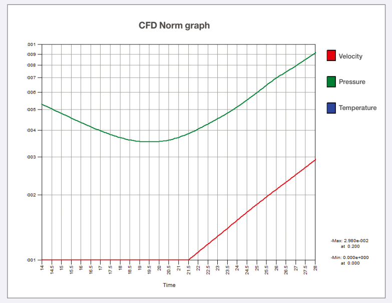
1. Iterative option

Calculation is fast in the usual way, but if user modeling is not appropriate, the calculation diverges.



2. Multifrontal option

Iterative methods provide stability for hard-to-calculate models, but increase computational time.

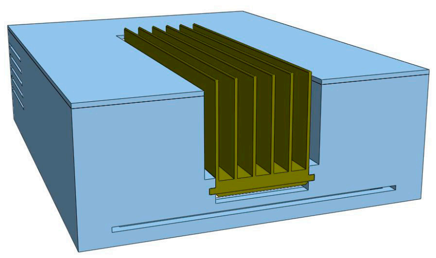


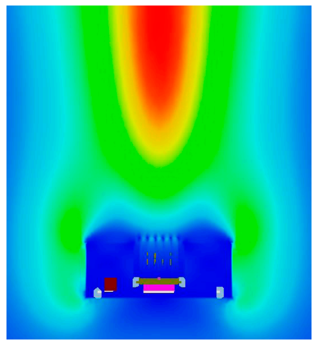
Specialized Thermal-Fluid Functionality
Reflecting Diverse Practical Demands

SOLVER

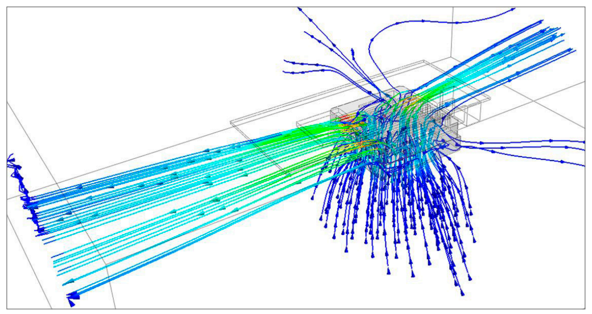
Conjugate heat transfer

Fluid and solid heat transfer, as well as fluid-solid inter-fluid heat transfer analysis, enables water cooling, air cooling, and heating analysis of heating elements.





An increase in ambient air temperature due to heat generation in the heat sink



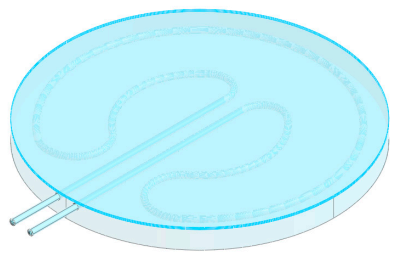
Forced convection analysis by fan

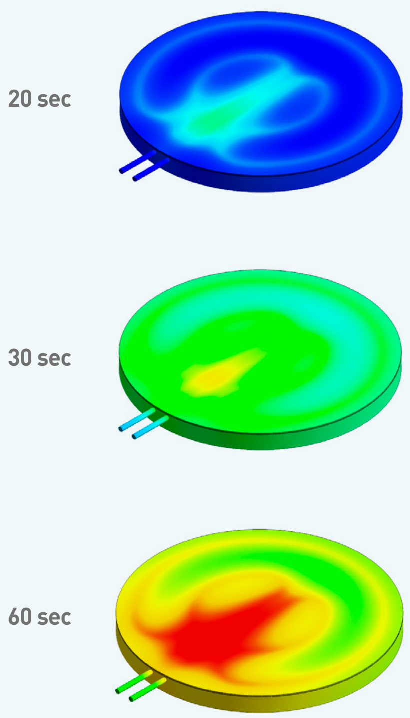
Applications : Natural Convection by Heat Sink, Forced convection analysis by fan

SOLVER

Joule heating

Estimate temperature changes by calculating the amount of heat generated by the electric potential difference inside the conductor.





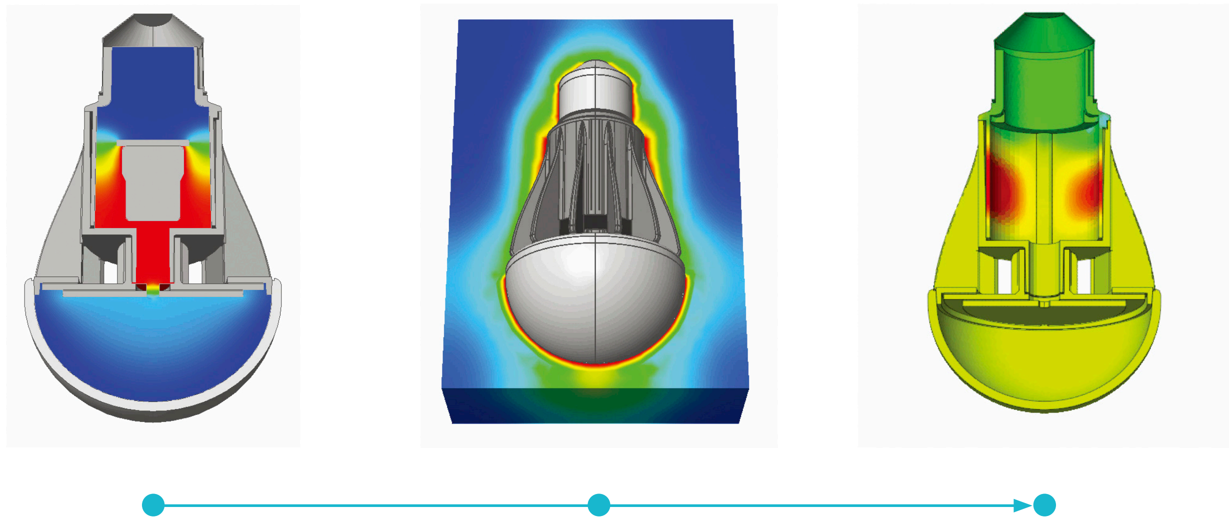
Heating of wafer chuck due to electric potential difference

Applications : Semiconductor wafer chuck, electric heater, Distribution panel

SOLVER

Radiation heat transfer analysis

When radiation phenomena of the analysed structure need to be considered, radiation heat transfer analysis can be used.

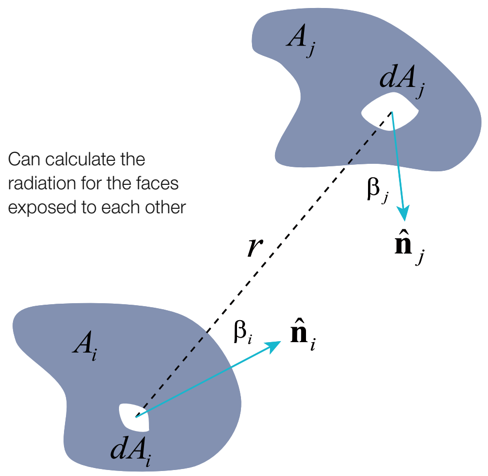


Inner radiation heat transfer

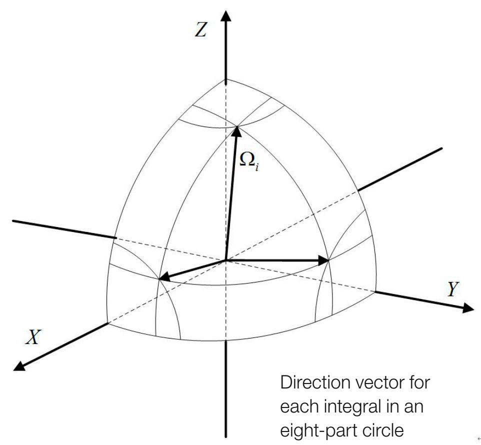
Outer radiation heat transfer

Final temperature distribution of LED lighting parts

Cavity Radiation



Discrete Ordinate Method



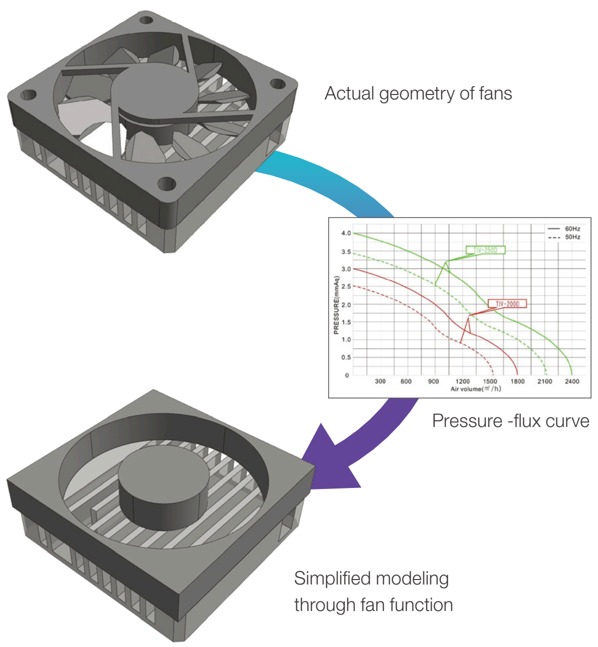
Applications : Heat treatment equipment for semiconductors and displays, furnaces, combustion engines, automotive under-hoods, headlines.

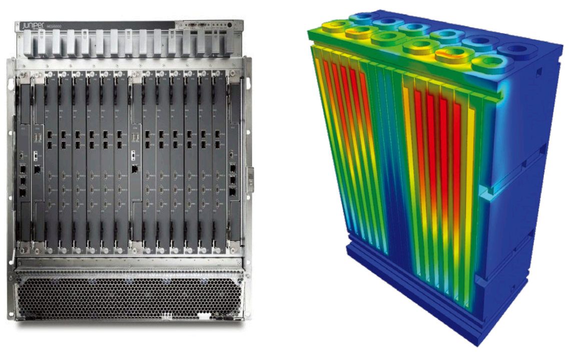
Dedicated Fluid Analysis Features
for Effective Practical Design

SOLVER

Auto-generation of fans

Instead of modeling the rotation of fans,
you can easily model fans by inputting pressure-flux curve.





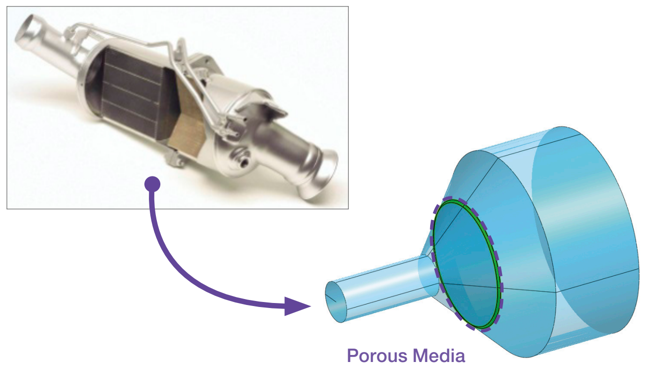
Temperature distribution inside the equipment

Applications : cooling fans for electronic equipment, clean room FFU, fans for production equipments

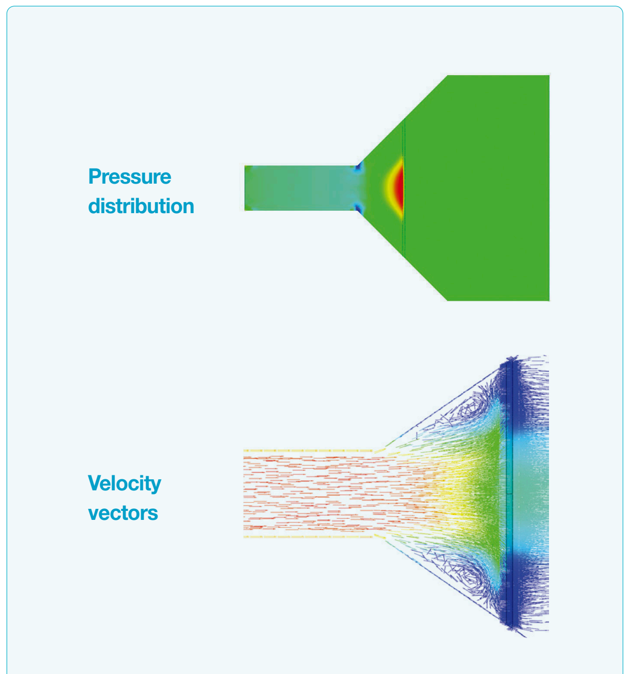
SOLVER

Porous model

Use porous media function to analyze a tube expansion system in which microfiber mediums are used to expend flow paths.



Application example of porous model



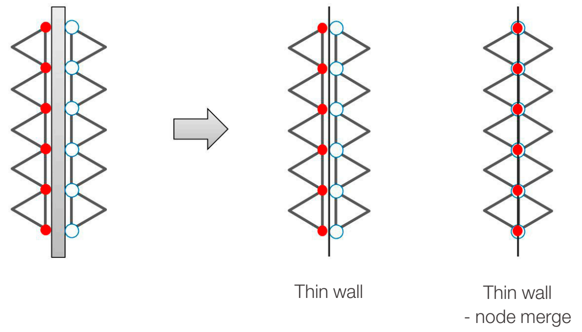
Analyze a tube expansion system

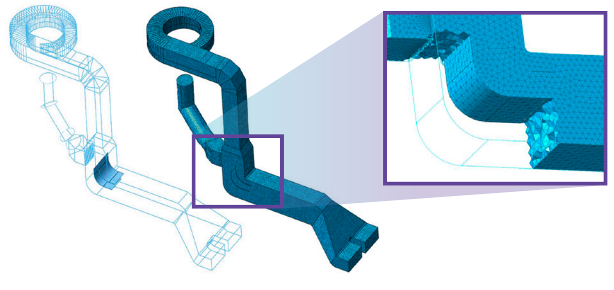
Applications : filters, perforated plates, grills, dust collectors, laminated materials, car exhaust systems

SOLVER

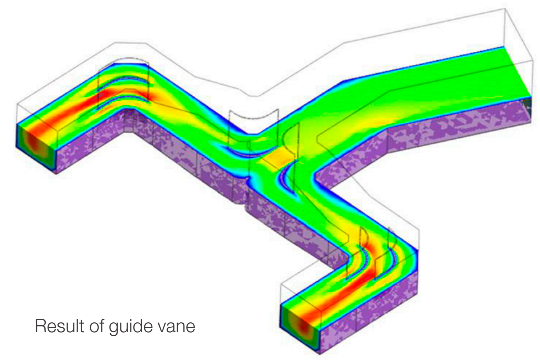
Thin wall model

Thin wall is modeled as a face helps create efficient meshing.





Generation mesh of guide vane
using thin wall modeling



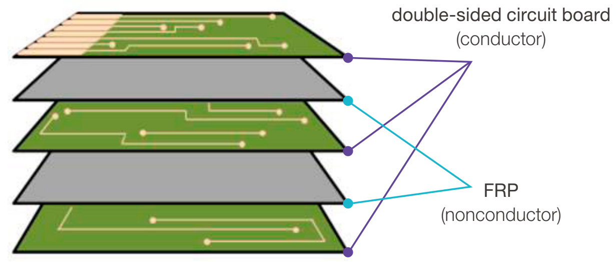
Result of guide vane

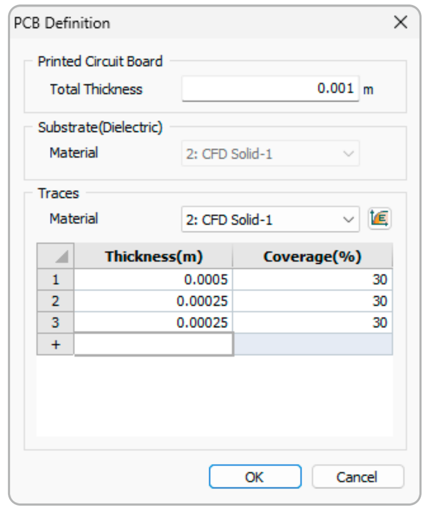
Applications : electronics baffle, duct guide vane

SOLVER

Thermal resistance · PCB model

Effective heat transfer analysis of board level system by reflecting thermal resistance characteristics of semiconductor package and PCB.





$$k_{inplane} = \frac{\sum k_i t_i}{\sum t_i} \qquad k_{normal} = \frac{\sum t_i}{\sum (t_i / k_i)}$$

$$k_i = f_i k_{cu} \quad \text{or} \quad k_i = k_{\mathrm{FR4}}$$

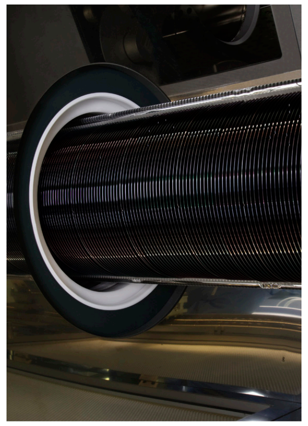
Applications : semiconductor package, Board level system, PCB system

Functionality for
Specialized Design Issues

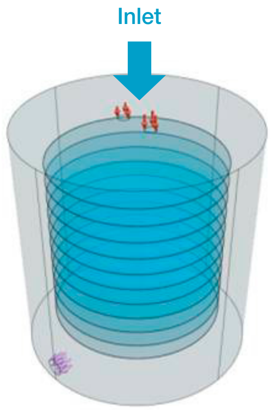
SOLVER

Mixture

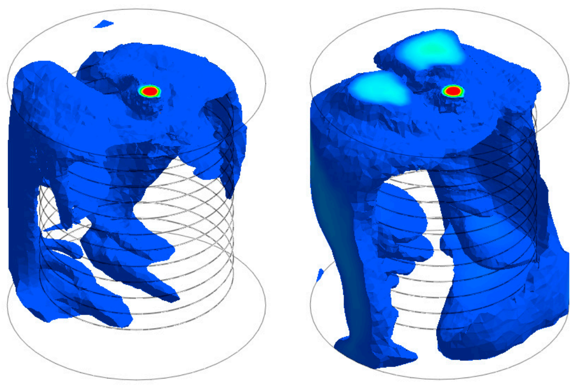
Calculate the fraction of a particular substance by mixing two or more types of fluids into one area using diffusion properties.



CVD SiC chamber



Mixture model



20sec   30sec

Results of mass fraction

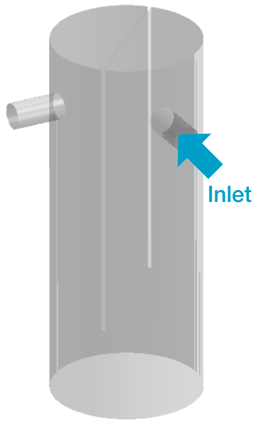
Applications : diffusion of pollutants, mixed gas valve, CVD chamber, agitator's rotation

SOLVER

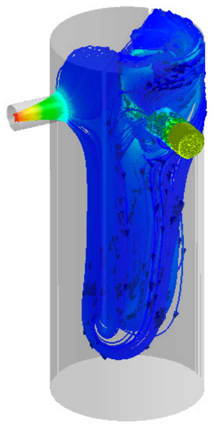
Particle

Analysis including small particles such as aerosols can be performed and the speed and travel path of particles can be predicted.

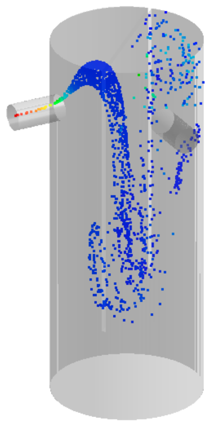
Oil catch can



Particle model



Particle path



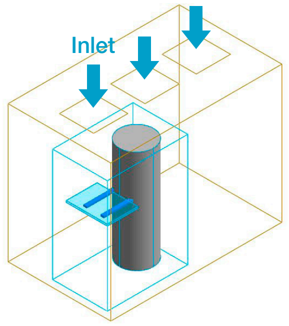
Particle spread

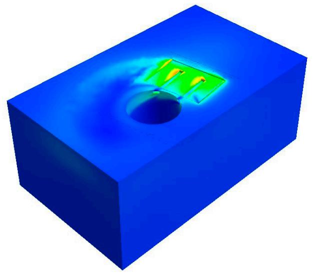
Applications : Oil catch can, oil injection, cyclone, clean room equipment, collection efficiency

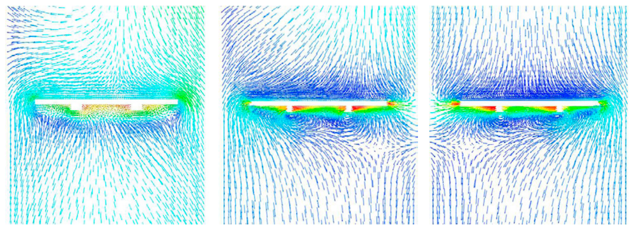
SOLVER

Overset Mesh

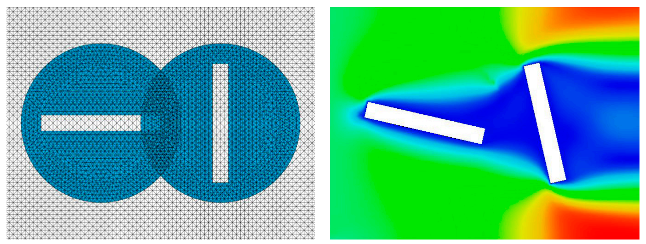
Can easily calculate by setting the surroundings of objects with complex behavior as Overset Mesh.







Analysis of clean room equipment using Overset Mesh



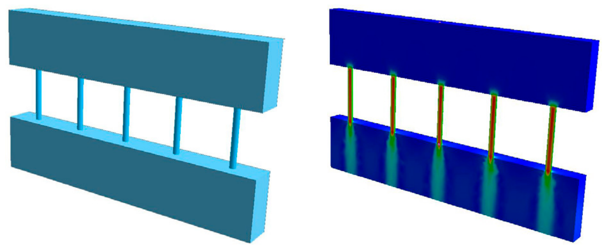
Analysis of mixer rotation

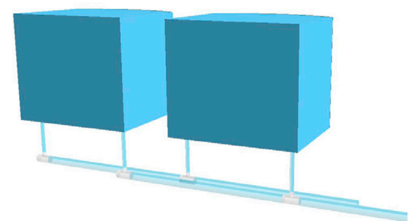
Applications : agitator's rotation, clean room equipment, analysis considering cross-movement, analysis of switchgear

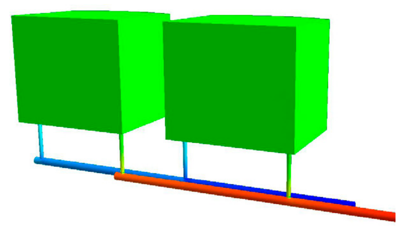
SOLVER

1- Dimention modeling

The plant's large piping system is set up as a one-dimensional element, dramatically reducing analysis time and enabling efficient calculation.







Analysis of tank-piping linkage system using one-dimensional flow analysis function

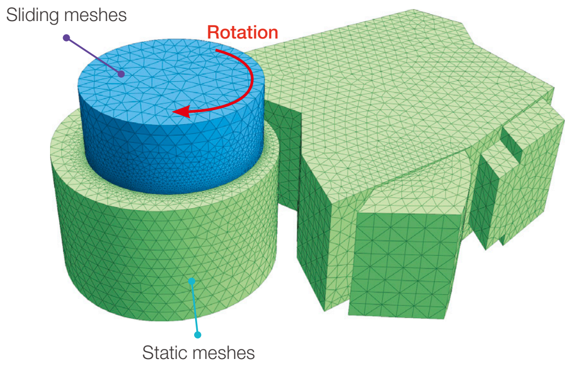
Applications : Tank-to-pipe linkage, plumbing system

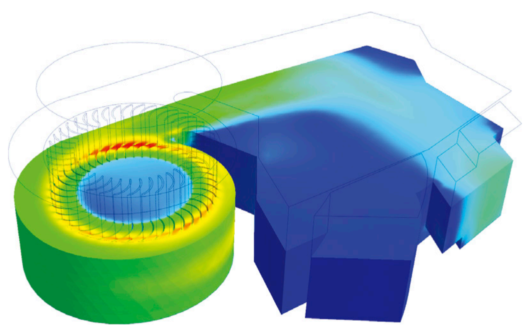
Utilization of Various Practical Analyses
through Specialized Functionality

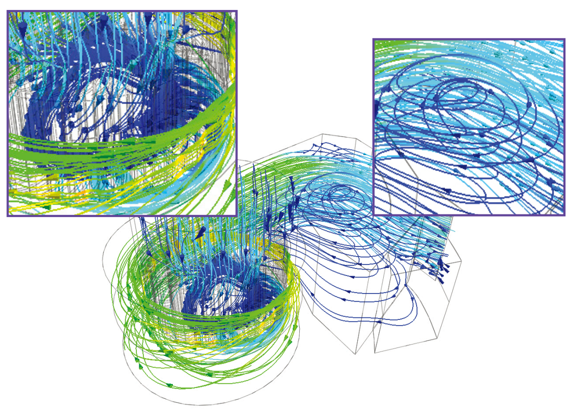
SOLVER

Mesh deformation module

When product perform rotation or linear motion together with hydraulic machine, this module can be used.





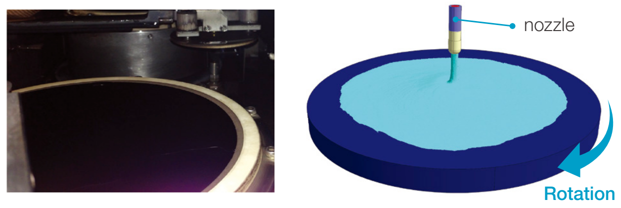


Performance analysis of rotation machine

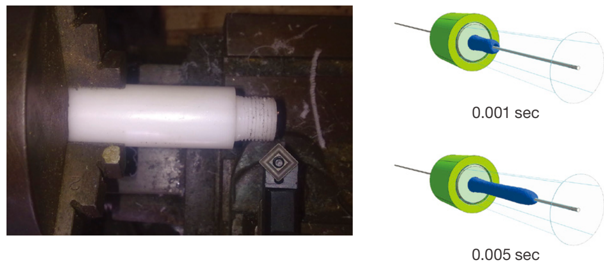
SOLVER

Multi-phase flow module

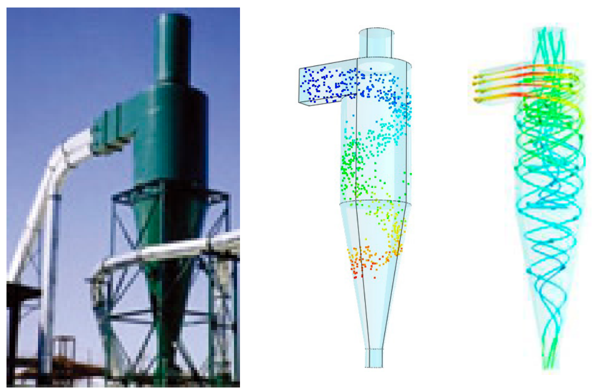
When fluid and gas of the free water surface need to be analyzed at the same time, this module can be used.



Fluid analysis of a rotating wafer



Textile guideline fluid nozzle



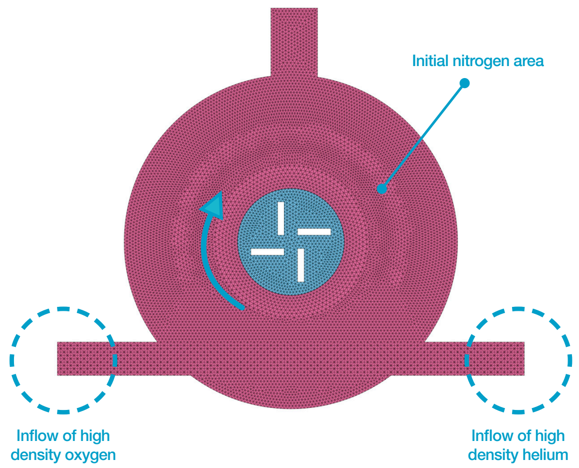
behavior of particles introduced into the cyclone

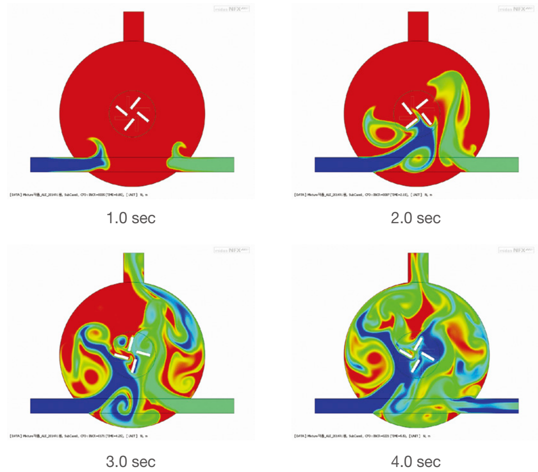
SOLVER

Species advection module

This module can be used to observe the diffusion phenomena of mixed materials defined by concentration fraction.

Mixed gas valve

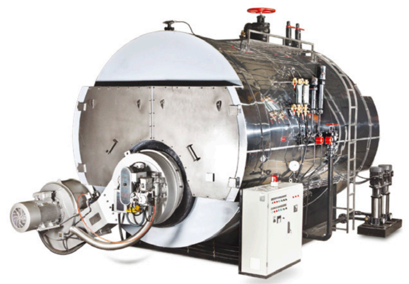




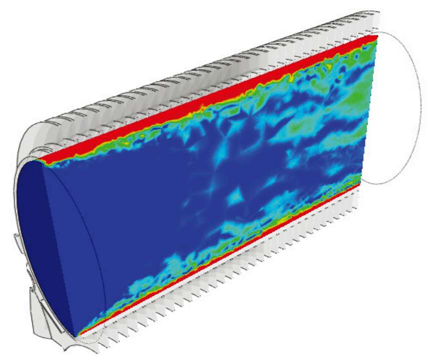
SOLVER

Fluid - structure coupled analysis module

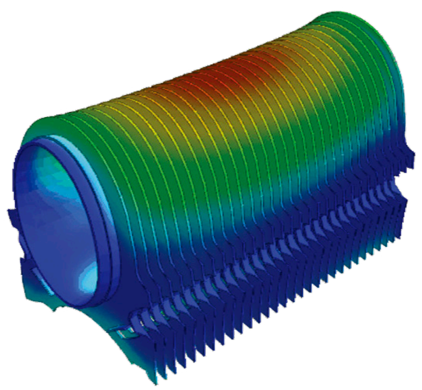
The analysis results of CFD analysis are used in structural analysis to calculate deformations and stresses.



Heat exchanger of a boiler



Heat flow analysis (CFD)



Thermal contraction analysis
(structural heat transfer analysis)

MIDAS NFX Specification

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

MIDAS NFX CFD

MIDAS

MIDAS
NFX CFD

MIDAS

nfx.midasuser.com