PROJECT APPLICATIONS

midas NFX

Total Solutions for True Analysis-driven Design

MIDAS

midas NFX

Total Solutions for True Analysis-driven Design

PROJECT APPLICATIONS

midas NFX

Total Solutions for True Analysis-driven Design

Machinery

Deflection measurement of robot arm

Overview

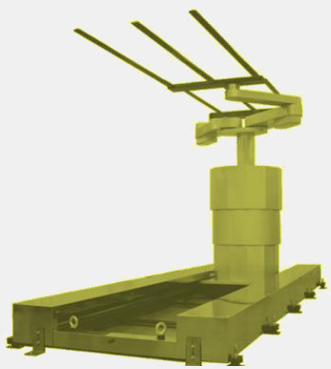
Objective

To identify structure's maximum displacement during the loading of the glass parts and during the movement of the loaded arm.

To propose optimum design according to deformation allowance.

Analysis Type

- Geometry nonlinearity
- Nonlinear Quasi-Static Analysis

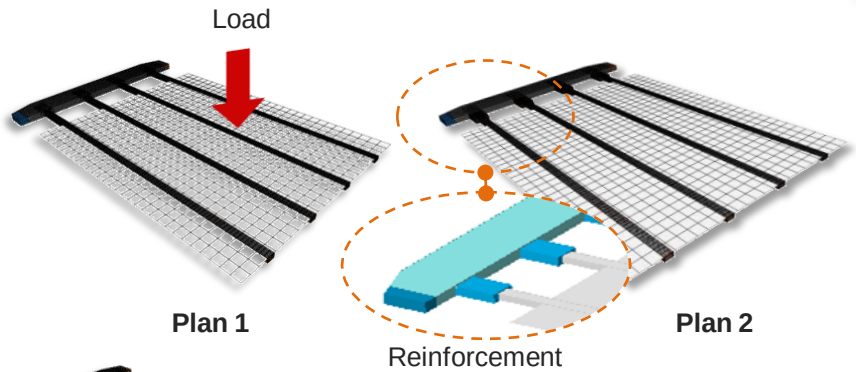


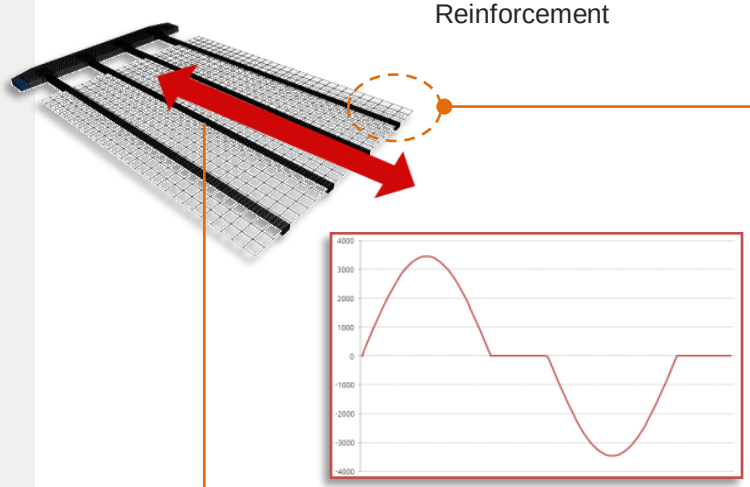
Pre Processing

Modeling : 3D Solid Element/ 2D Shell Element (Glass size >2000 mm)

Boundary Condition: Glass is attached to the robot arm using contact

Loading Condition: Acceleration values have been defined as the loading condition to identify robot arm's deformation when it accelerates, moves uniformly and decelerates.



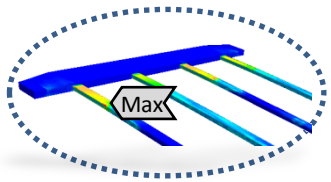


Acceleration loading input according to time

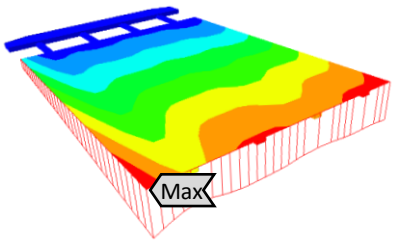
Post Processing

For steady state, both plan 1 and plan 2 are within the allowance deformation

Max Stress: 15.82 MPa



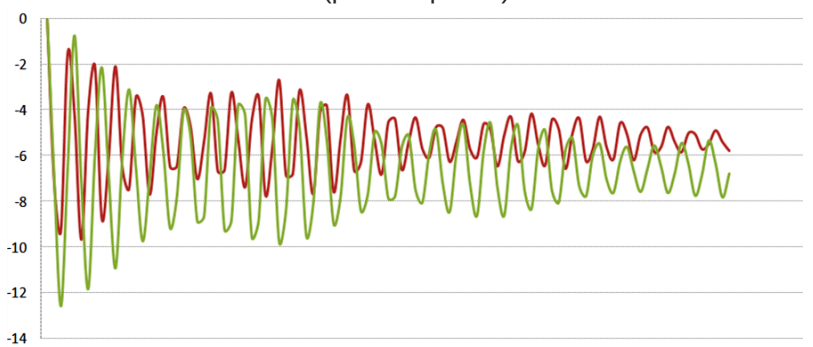
Max Displacement: 6.5 mm



When the weight of plan 2 model is increased by 3.2 kg (2% of total weight)

- maximum displacement decreases from 13 mm to 7.7 mm (deflection decreases 40%)

Deformation results at arm end nodes during glass transfer (pan 1 & plan 2)



Safety evaluation of semiconductor CVD chamber under vacuum load

Overview

Objective

Evaluation of semiconductor CVD chambers structural safety under self weight and repetitive vacuum load.

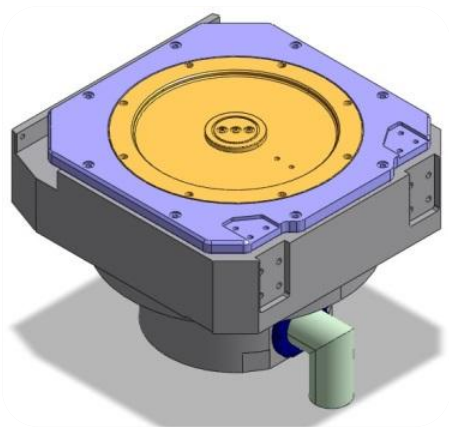
Analysis Type

- Linear static analysis

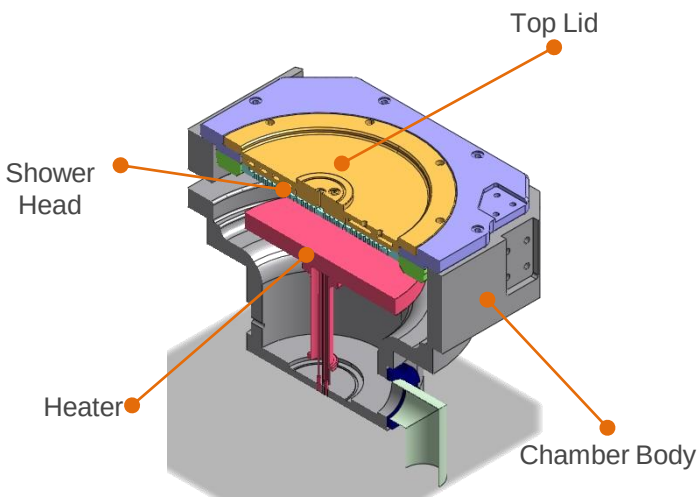
Pre Processing

Modeling : 3D Solid Element

Loading Condition : self weight+ vacuum load



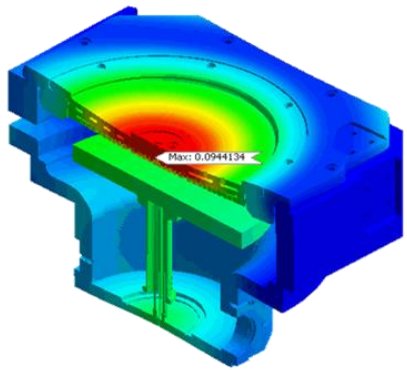
Analysis model



Post Processing

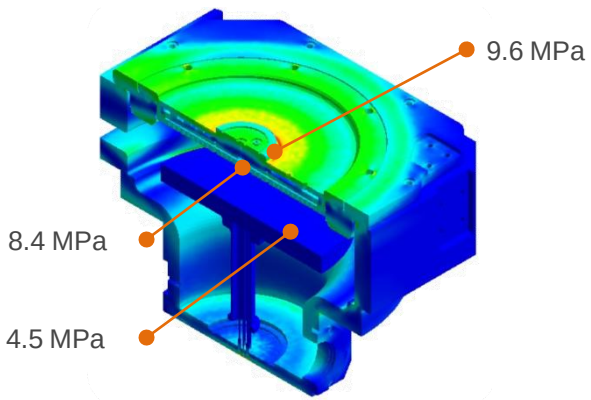
Current design has an excessive safety factor. Design modifications are required to decrease the costs of production.

- compared to allowance stress, calculated stress is very small



Displacement distribution

Maximum displacement: 0.1 mm (at the top lid)



Stress distribution

Maximum stress: 9.6 MPa (at top lid)

Design of semiconductor CVD chamber to improve flow uniformity

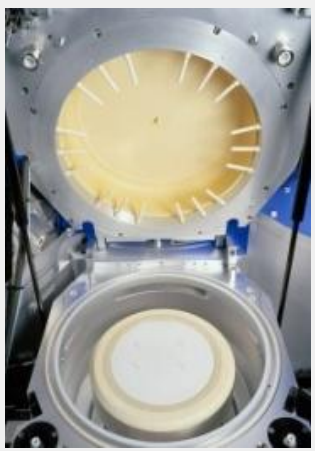
Overview

Objective

Evaluation of shower nozzle's injection uniformity of a semiconductor CVD chamber

Analysis Type

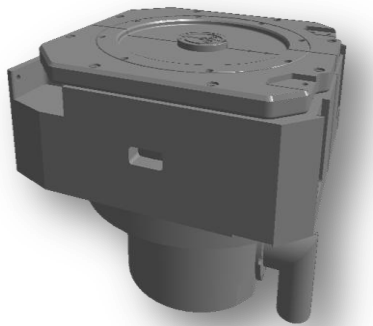
- Transient flow analysis
- Fluid flow module



Pre Processing

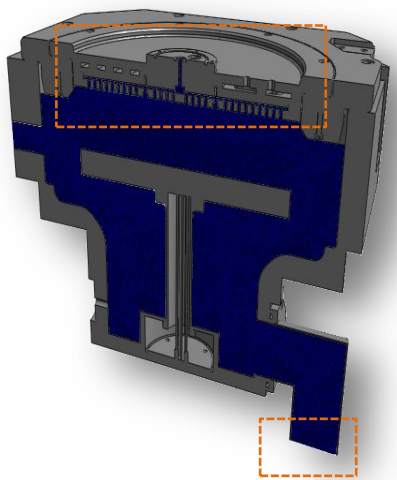
Analysis condition:

- k-ε turbulence model, initial stabilization step applied
- Inlet condition (velocity
- Outlet condition(pressure)
- Wall condition (no slip)



MOCVD chamber

Inlet: Shower nozzle 1140 sccm injection

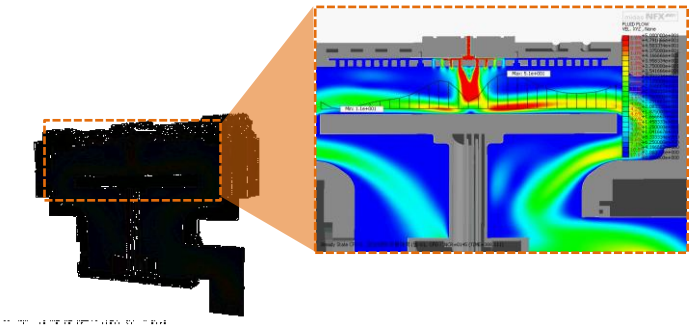


Outlet

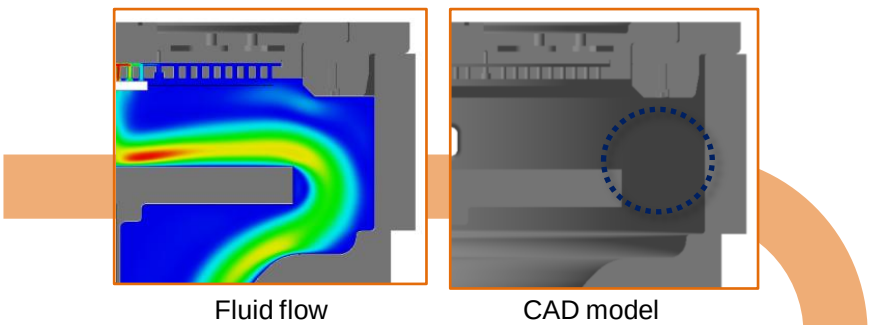
Post Processing

Graph generation on the model to evaluate flow uniformity

velocity and turbulence intensity

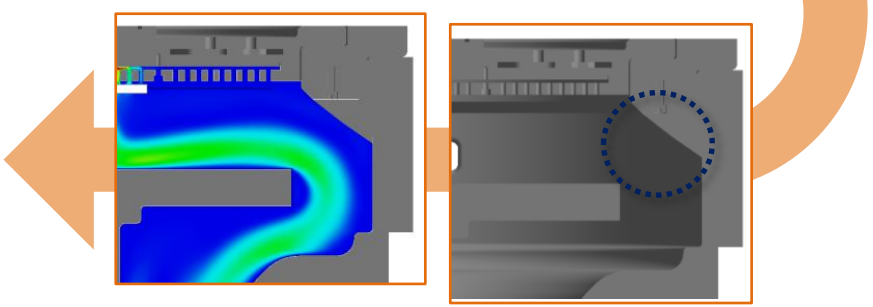


Visualization of fluid flow and gas uniformity



Fluid flow

CAD model



Performance improvement of fluid distribution

CAD model for modified design

Junction temperature evaluation of semiconductor system in a telecommunication

Overview

Objective

Prediction of junction temperature of the system under the condition of forced air cooling by fans.

Improvement of system design according to analysis results.

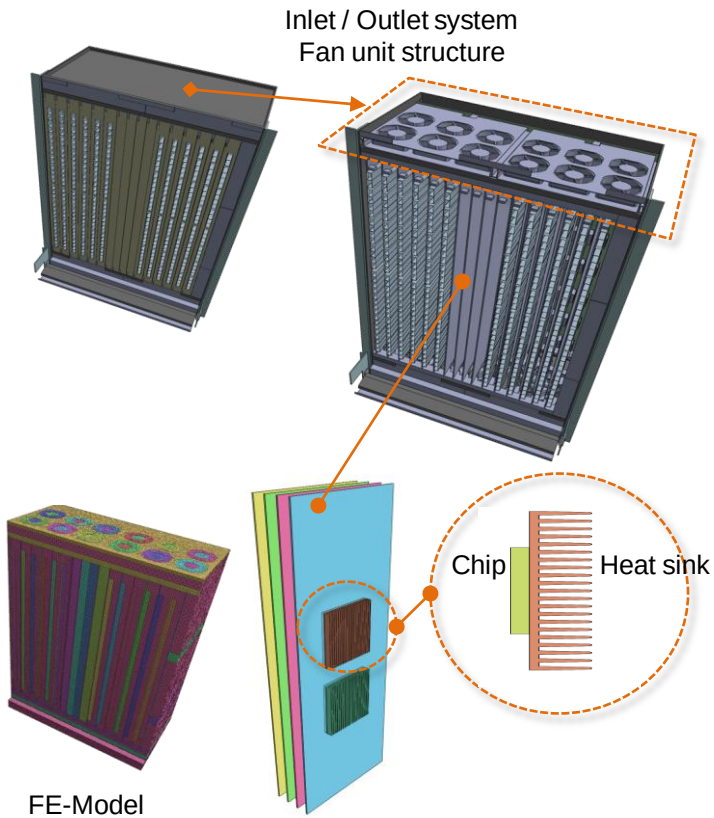
Analysis Type

- Fluid flow analysis
- Fluid-solid coupled heat transfer CFD analysis

Pre Processing

Modeling: 3D Solid Element

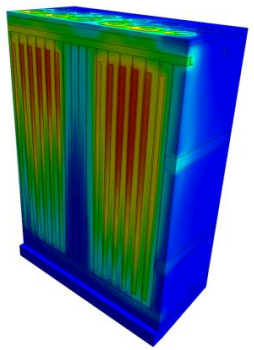
Analysis condition:
- Inlet: fan (pressure-flux curve)
- Outlet: fan (pressure-flux curve)
- Surface: non-dimensional wall distance, chip heat generation



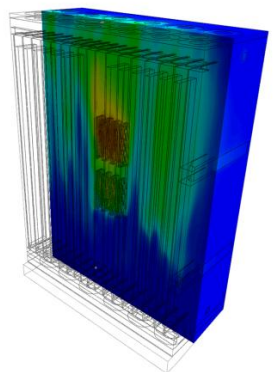
PCB – chip – heat sink system

Post Processing

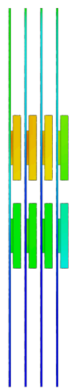
Investigation of overheating problem of upper chip according to air floatability is performed to evaluate the temperature distribution on whole system and air flow of fans.



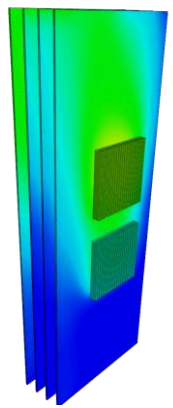
Temperature distribution on whole system



Temperature distribution on cross section



Temperature evaluation of PCB - chip - heat sink system



Temperature evaluation

Internal resistance analysis of air jetting nozzle in a semi-conductor equipment

Overview

Objective

Velocity and pressure analysis inside nozzle system

Verification of the flow stagnation area and the pressure drop area

Analysis Type

- Fluid flow analysis

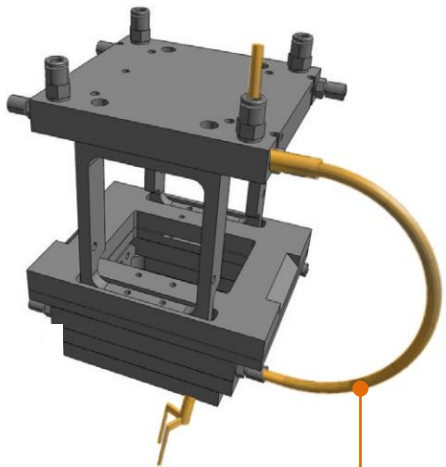
Pre Processing

Modeling: 3D Solid Element

Analysis condition:
- Inlet: velocity (flux)
- Outlet: pressure(pressure function by flux)
  k-ε turbulence model
- Surface: wall distance

Clean room system

Jetting nozzle



Inlet

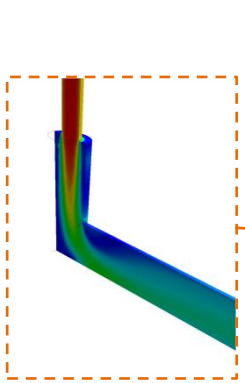
Modeling of fluid

Outlet

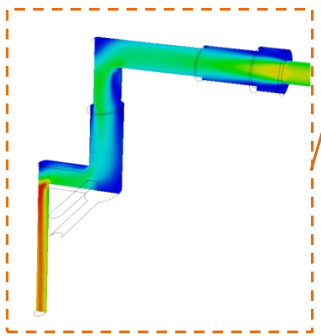
Post Processing

Fluid resistance evaluation in the curved tube

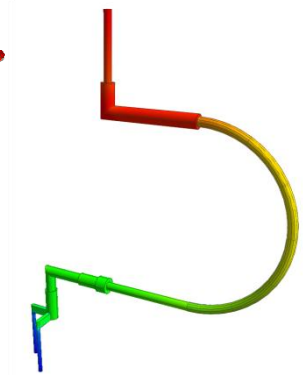
Pressure and flow line analysis



Velocity distribution



Velocity distribution



Differential pressure

Performance evaluation of a mobile speaker through sound pressure level (SPL) analysis

Overview

Objective

Investigation of dynamic characteristics of the speaker, displacements / pressures according to frequency spectrum.

Evaluation of performance of the speaker through sound pressure level

Analysis Type

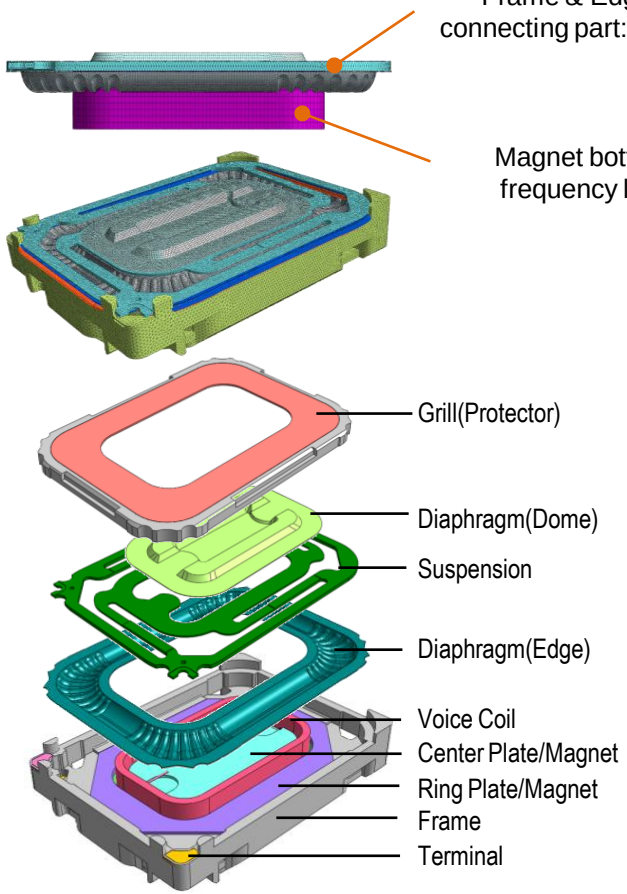
- Mode analysis
- Frequency response analysis

Pre Processing

Modeling: 3D Solid Element / 2D Shell Element

Loading condition:
- Frequency load: up-down movement of magnet
- Response result within audible frequency range 200~20,000 Hz

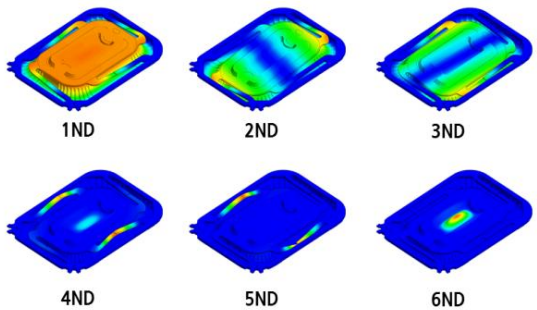
Contact condition:
- welded contact between all parts



Post Processing

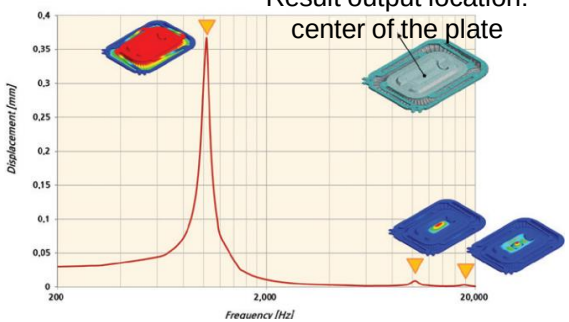
Highest mass participation at 1st mode

(maximum displacement occurs around 1000 Hz)

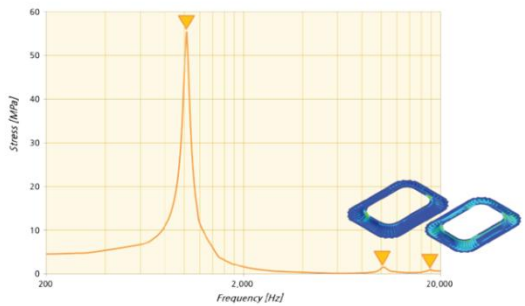


Mode	Frequency
1st	1 046 Hz
2nd	1 778 Hz
3rd	1 801 Hz
4th	9 457 Hz
5th	9 679 Hz
6th	10 217 Hz

Result output location:
center of the plate



Displacement



Stress

Air cooling analysis of semi-conductor package

Overview

Objective

Performance evaluation of heat generation according to size, shape and location of heat sink in the system

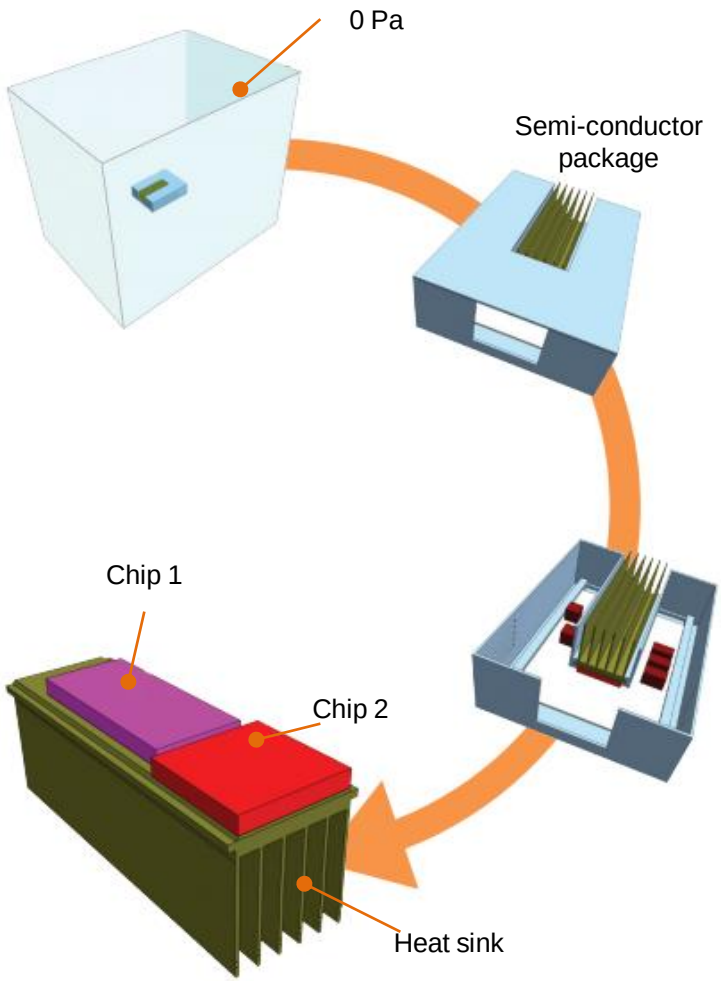
Prediction of the maximum temperature on chip in steady state

Analysis Type

- Fluid flow analysis
- Fluid-solid coupled heat transfer CFD analysis

Pre Processing

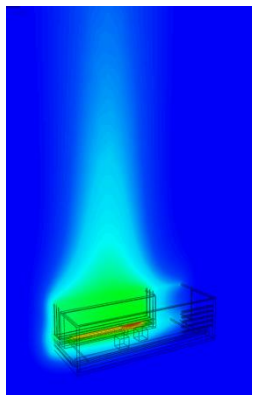
Modeling: 3D Solid Element
Incompressible ideal gas model for natural convection
After modeling external air, a 0 Pa outlet condition have been applied.
Air temperature 25 ℃



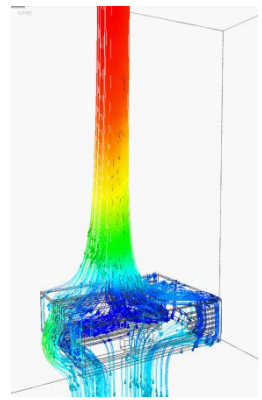
Post Processing

Analyze cooling performance inside the system under condition of natural convection outside the system

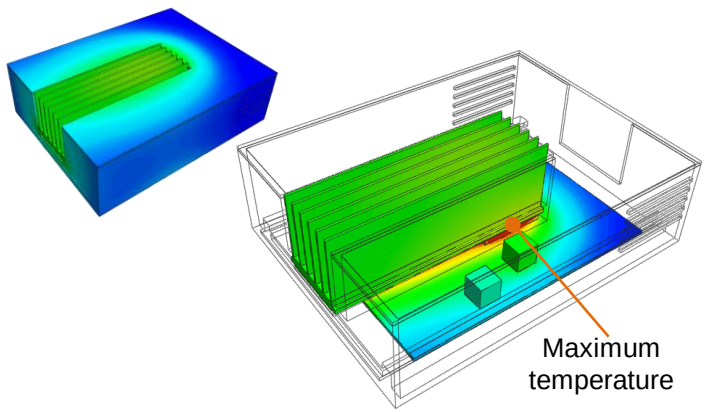
Heat transfer analysis on chip and heat sink



Heat airflow distribution



Heat airflow path distribution



Temperature distribution inside the system

Service life prediction of bellows weld zone

Overview

Objective

To predict the service life of bellows under repetitive tension and compression

Analysis Type

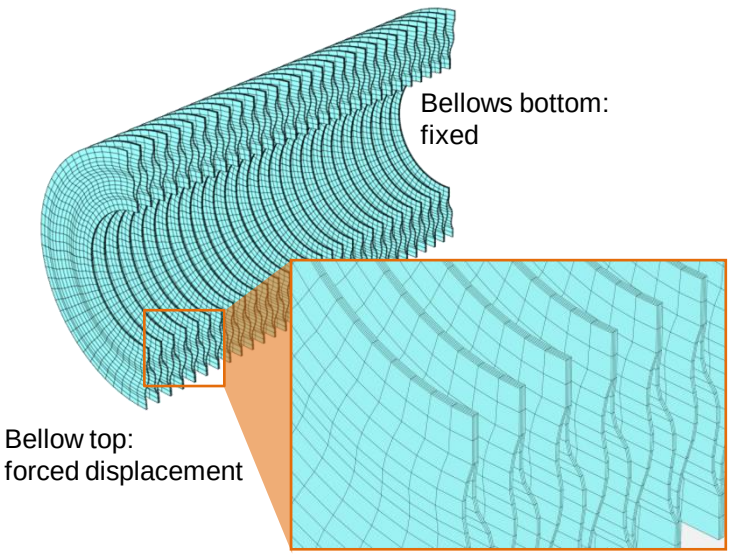
- Material nonlinearity
- Geometric nonlinearity
- Fatigue analysis

Pre Processing

Modeling : 3D Hexahedral Solid Element

Loading Condition

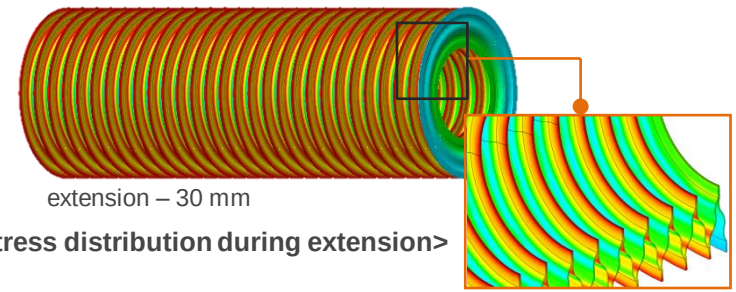
- Forced displacement is used to realize tension and compression of bellows
- Geometric nonlinearity is used to consider large deformation.
- Fatigue analysis using S-N curve method is used to predict the service life of bellows



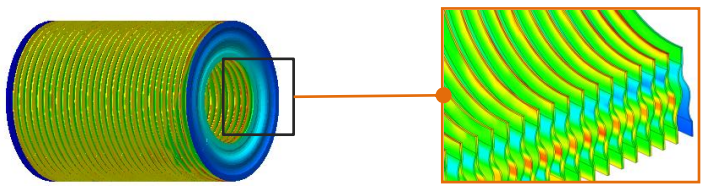
<Hexahedral mapped meshes>

Post Processing

Calculate stress at weld zone through nonlinear analysis
- extension : Maximum 240 MPa
- compression : Maximum 220 MPa

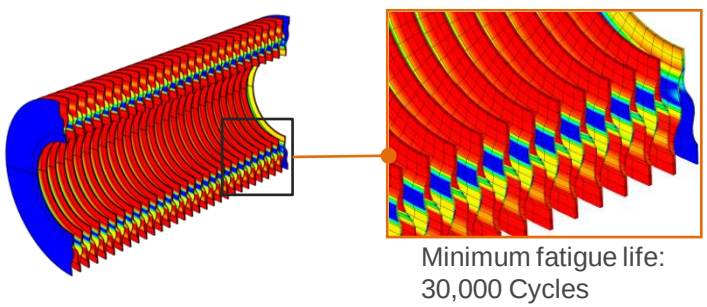


<Stress distribution during extension>



<Stress distribution during compression>

Calculate fatigue life by applying fatigue analysis
- minimum fatigue life: 30,000 Cycles



<Service life>

Heat transfer and thermal deformation analysis of boiler heat exchanger

Overview

Objective

Heat exchanging performance assessment of the heat exchanger inside boiler.

Structural safety assessment by analyzing heat exchanger's thermal expansion / contraction

Analysis Type

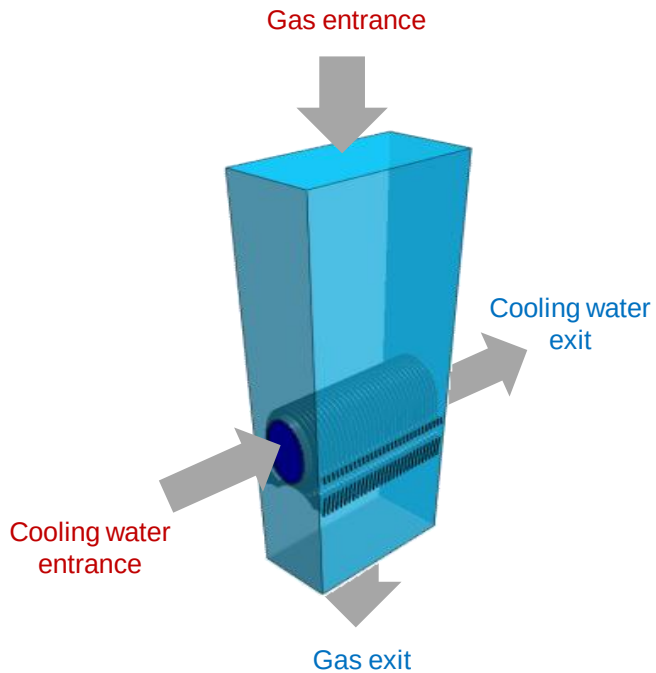
- Transient flow analysis
- Fluid flow module
- Heat transfer module
- Solid heat transfer module
- Linear static analysis

Pre Processing

Analysis Condition:

- Inlet condition (velocity) for cooling water inside and air flow outside
- Outlet condition (pressure) for cooling water inside and air flow outside
- Wall condition (no slip) for heat exchange between solid outside and inside

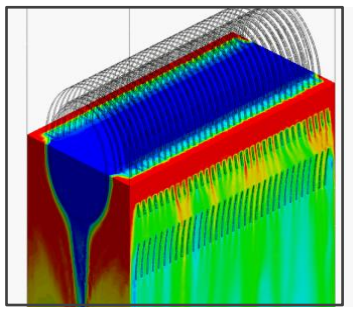
Temperature result from flow analysis coupled with thermal structural analysis

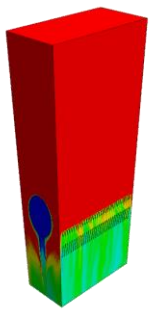


Post Processing

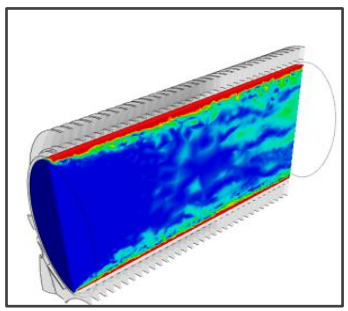
Assess temperature and flow of high temperature gas outside and cooling water inside

Analyze heat exchanger's thermal contraction by coupling with thermal analysis

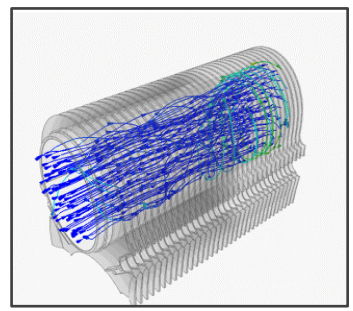




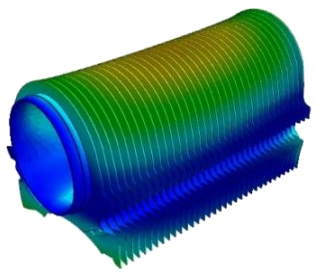
High temperature gas outside



Temperature



Flow line



Thermal contraction state (stress)

Material reaction performance analysis inside mixing tank

Overview

Objective

Evaluation of material mixing performance of a mixing tank through CFD analysis and mixing performance by shape modification, material concentration assessment

Analysis Type

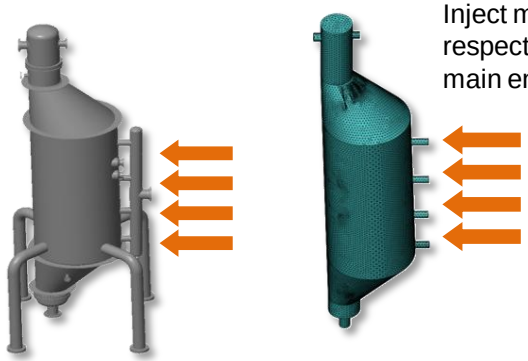
- Transient flow analysis
- Fluid flow module
- Material diffusing module

Pre Processing

Boundary Condition:

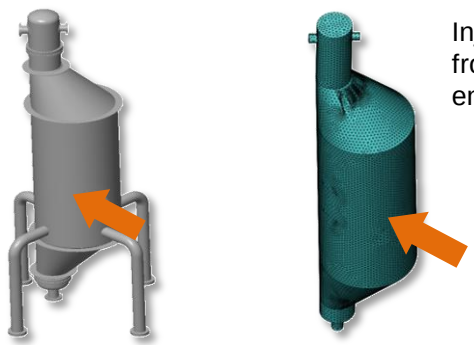
- Entrance: inlet boundary (speed), material density
- Exit: outlet boundary (pressure)
- Surface: wall boundary (no slip)
- Definition of materials and diffusion coefficients

CASE 1



Inject materials respectively from 4 main entrances

CASE 2



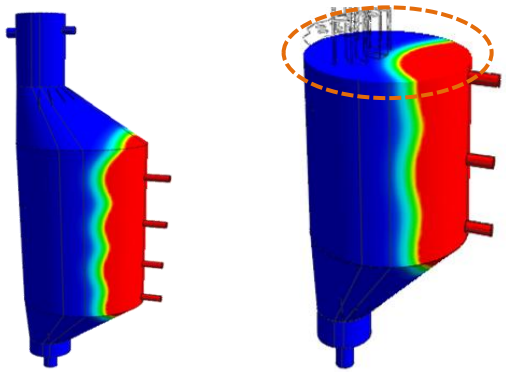
Inject materials from 1 main entrance

Post Processing

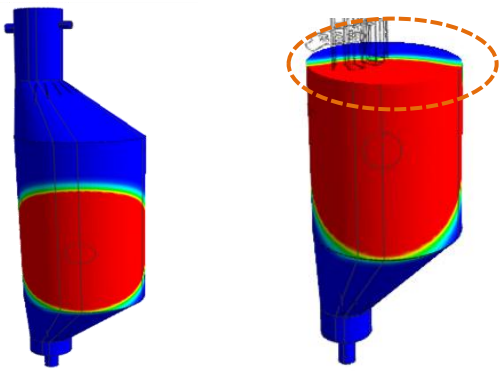
Concentration is measured at target location according to the modification of the inlet ‘s shape.

When 4 small inlets are replaced by 1 large inlet, concentration at the target location is increased.

CASE 1



CASE 2



Material reaction performance analysis inside mixing tank

Overview

Objective

Design cooling fins to ensure effective cooling performance of a pump, which operates under high temperature and high pressure environment

Analysis Type

- Steady state heat transfer analysis

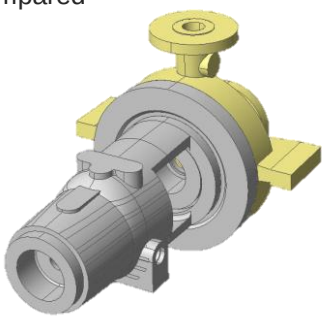
Pre Processing

Modeling : 3D Solid Element

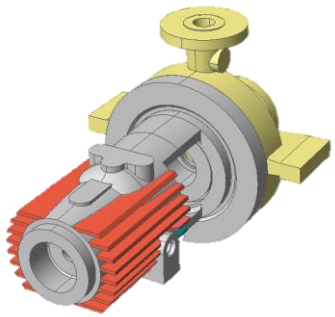
Main loading condition:

- Temperature load input measured from experiment
- Heat flux, fixed temperature, convection condition

Performance between original design and improved design is compared

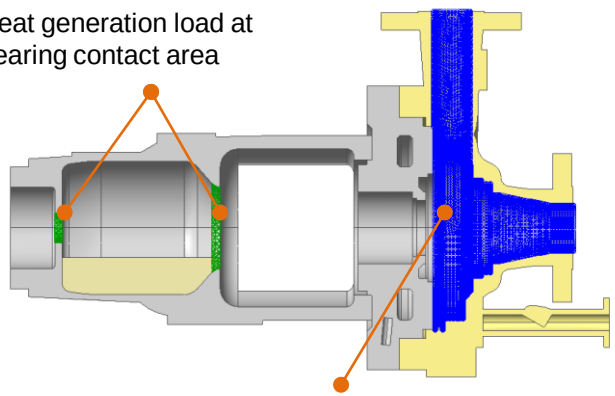


<Original Design>



<Improved Design>
Add cooling fins

Heat generation load at bearing contact area



Fluid temperature load inside the housing

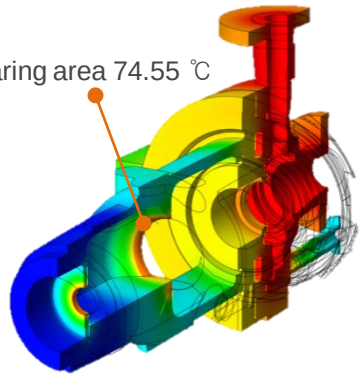
Post Processing

Steady state heat transfer analysis
- Original design : bearing area 74.55 ℃
- Improved design : bearing area 65.53 ℃

Through design of cooling fins, temperature is reduced by 9 ℃, 13.7%.

Through design of cooling fins, water head of bearing increases and pump stability is improved

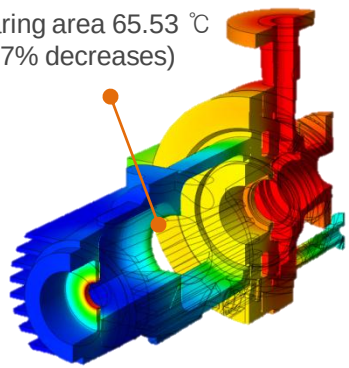
Bearing area 74.55 ℃



<Original Design>

Position	Temperature
7310 bearing	95.95 ℃
6309 bearing	74.55 ℃
Housing center	53.80 ℃
Lubricant	69.27 ℃

Bearing area 65.53 ℃
(13.7% decreases)



<Improved Design>

Position	Temperature
7310 bearing	95.38 ℃
6309 bearing	65.53 ℃
Housing center	45.04 ℃
Lubricant	62.16 ℃

Cone meter fluid force analysis and structural stability evaluation

Overview

Objective

Evaluation of structural stability inside the cone meter which is used to measure flow rate.

Calculation of fluid force at the pipe area through CFD method coupled with structural analysis

Analysis Type

- Transient flow analysis
- Fluid flow module
- Linear static analysis

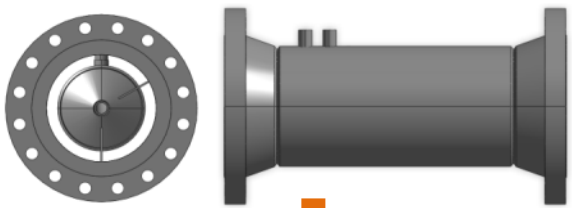
Pre Processing

Modeling : 3D Solid Element

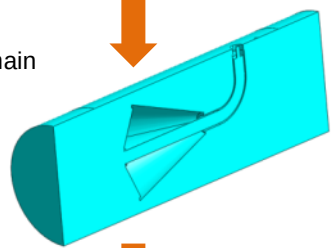
Main loading condition:

- Temperature load input measured from experiment
- Heat flux, fixed temperature, convection condition

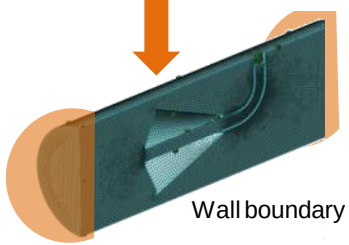
Compare performance between original design and improved design



Automatic fluid domain extraction



Automatic mesh generation



Inlet
91.6m³/h
(0.5m/s)

Wall boundary

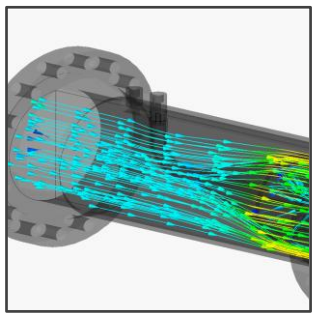
Outlet
Atmosphere pressure condition hypothesis

Post Processing

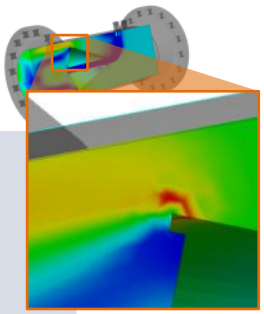
Verify fluid speed and pressure distributions

Verify high velocity areas in the fluid

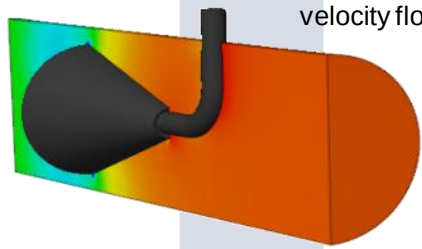
Verify stress and displacement distribution



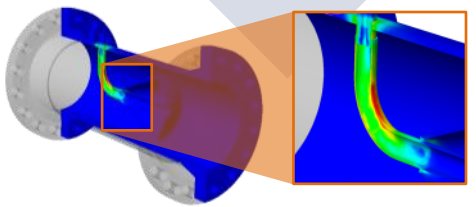
Pipe – cone meter flow lines



Observation of high-velocity flow



Fluid pressure around pipe area



Structural analysis coupled with fluid pressure results

Deformation analysis of rotating turbine blade during collision

Overview

Objective

Calculation of deformation of a rotating turbine blade during the collision of a small projectile.

Analysis Type

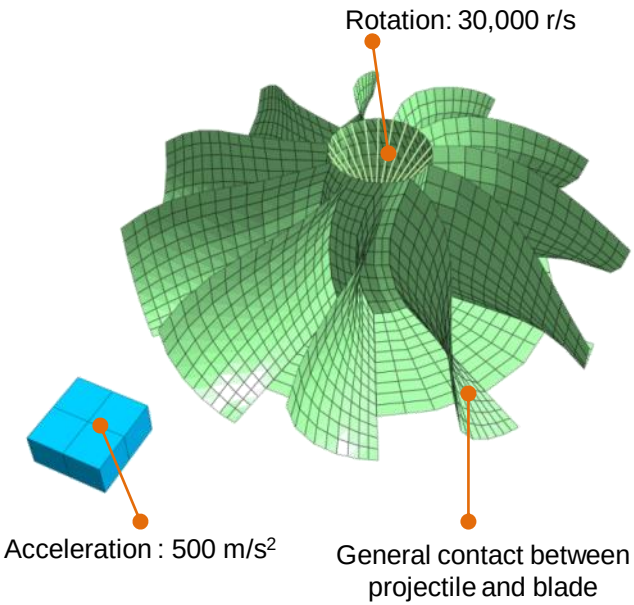
- Explicit nonlinear dynamic analysis

Pre Processing

Modeling : 2D Quadrilateral Element s & 3D Hexahedral Elements

Analysis Condition:

- Rotation speed : 30,000 rad/sec
- Define rotation of the turbine blade via initial speed
- Acceleration of rigid projectile: 500 m/s^2

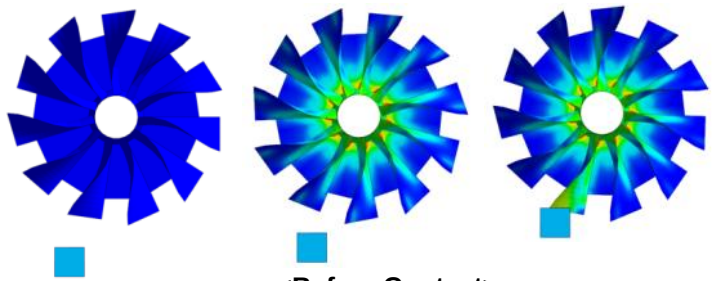


Post Processing

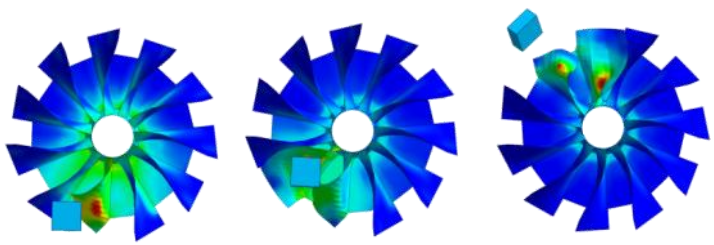
The impeller deforms to absorb the collision energy

Backlashing happens under excessive collision load

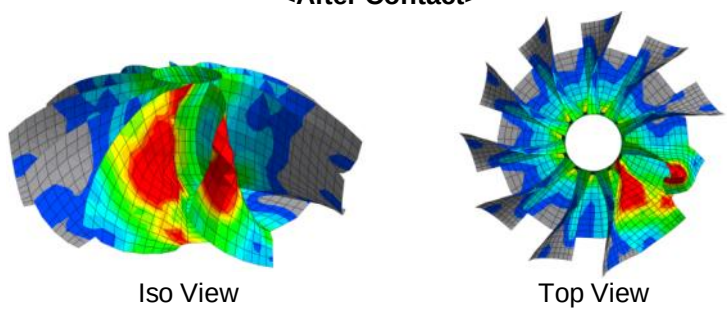
Snap-through buckling happens on the impeller during collision



<Before Contact>



<After Contact>



Snap-Through Buckling

Valve system performance evaluation at specific opening

Overview

Objective

Evaluation of valve performance at specific opening and analysis of differential pressure and flux to determine valve performance.

Checking of the stability of the disk system

Analysis Type

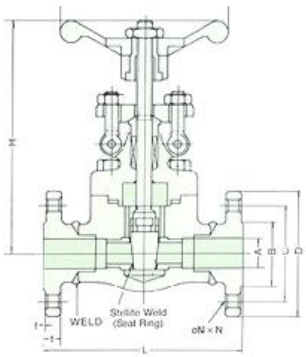
- Transient flow analysis
- Fluid flow module
- Linear static analysis

Pre Processing

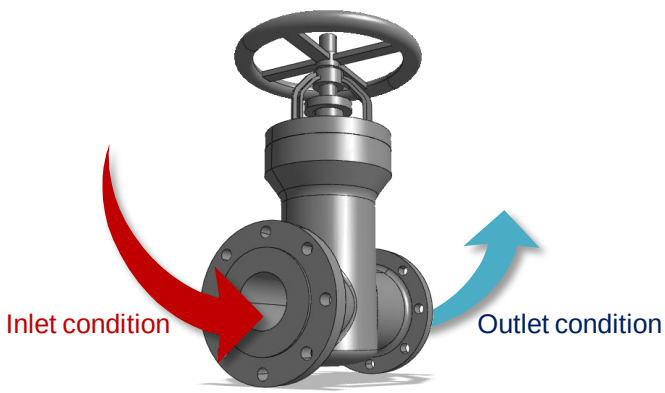
Analysis Condition:

- k-ε turbulence model, apply initial stabilization step
- Inlet condition(speed)
- Outlet condition(pressure)
- Wall condition(no slip)

Linear static analysis: pressure distribution input by coupling with CFD analysis results



Wall condition

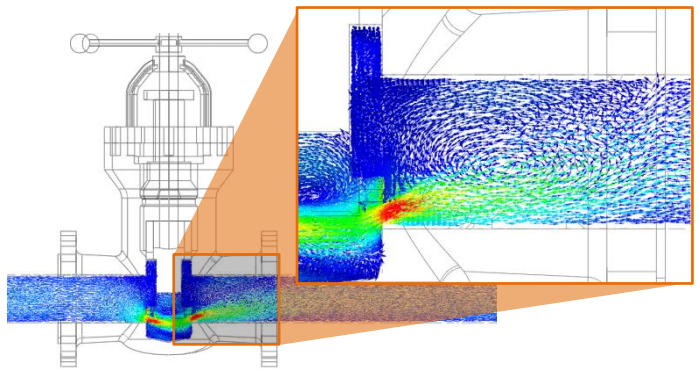


Post Processing

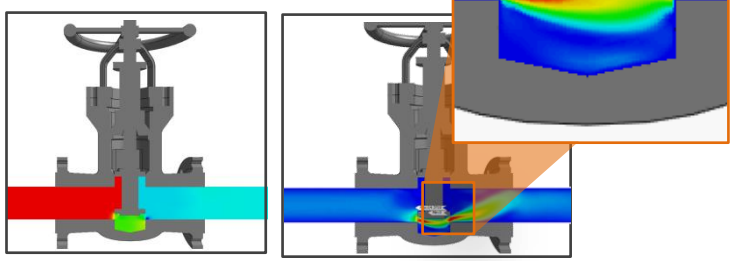
Evaluation of :

- differential pressure by measuring pressure results at entrance and exit.

- flux by velocity distribution and flux visualization and structural stability at disk area



Flow and vortex evaluation



Differential pressure analysis

Maximum velocity and dynamic pressure evaluation

Stability review of solar panel under wind load

Overview

Objective

Review of solar panel stability under complex wind load for which standard wind load formula doesn't apply

Perform structural analysis by coupling with CFD analysis fluid pressure result

Analysis Type

- Transient flow analysis
- Fluid flow module
- Linear static analysis

Pre Processing

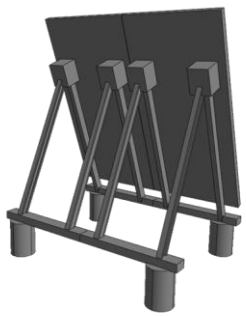
Analysis Condition:

- k-ε turbulence model, apply initial stabilization step
- Inlet condition(velocity)
- Outlet condition (pressure),
- Wall condition (no slip)
- Free surface condition (Vertical velocity= 0)

Linear static analysis: Input pressure distribution load through result conversion function.

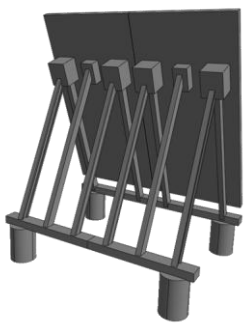
- Stress and deformation review of supporting parts

CASE 1



Initial design

CASE 2



Support reinforced design

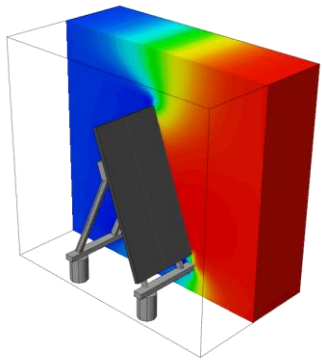
Post Processing

Wind load calculation through pressure data

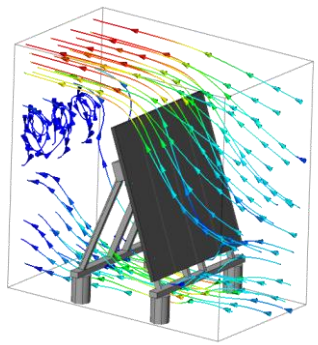
Stress calculation according to design modifications

Stress before design modification : 2.88×10^7 Pa

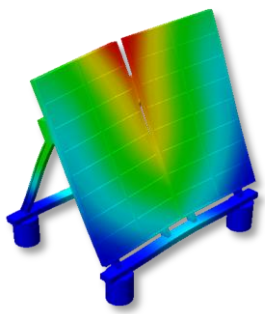
Stress after design modification : 2.47×10^7 Pa



Wind pressure review (solar panel front)

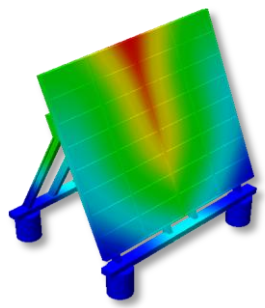


Wind flow lines and turbulence analysis



CASE1 Deformation shape

Maximum stress $2.88X10^7$ Pa



CASE 2 Deformation shape

Maximum stress $2.47X10^7$ Pa

Heat treatment furnace design to improve air conditioning performance

Overview

Objective

Redesigning circulation path inside the heat treatment furnace to improve its cooling efficiency

Evaluation and comparison of temperatures on the solid part inside the furnace.

Analysis Type

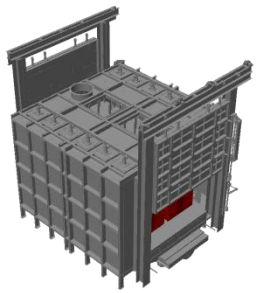
- Transient flow analysis
- Fluid flow module
- Heat transfer module
- Solid heat transfer module

Pre Processing

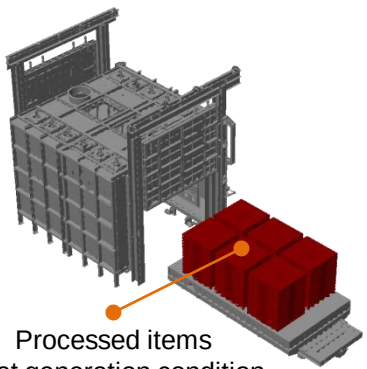
Analysis Condition:

- Material heat generation condition (heat source)
- Entrance: inlet (velocity) and temperature conditions
- Exit: outlet (pressure) condition
- Fluid around the contact surface of the solid structure: wall condition (no slip)

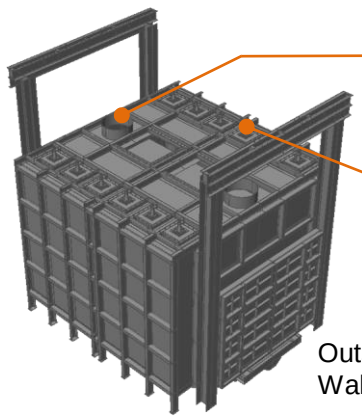
Prediction of temperature change according to different designs



Heat treatment furnace model



Processed items
Heat generation condition



2 exhaust ports
Outlet condition

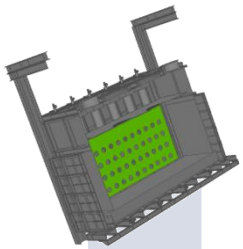
6 intake ports
Inlet condition

Outside the solid
Wall condition

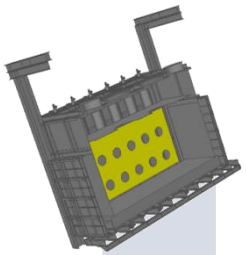
Post Processing

There is a wall between the exhaust ports and the intake port. Nozzles are set on the wall.

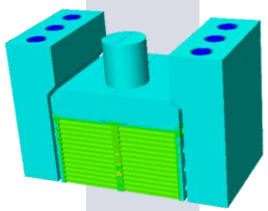
For furnaces of different nozzle designs, evaluate and compare temperatures on the processed items.

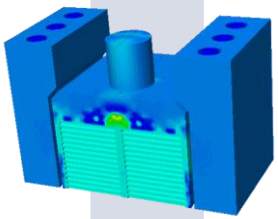


Original Design
60 small nozzles

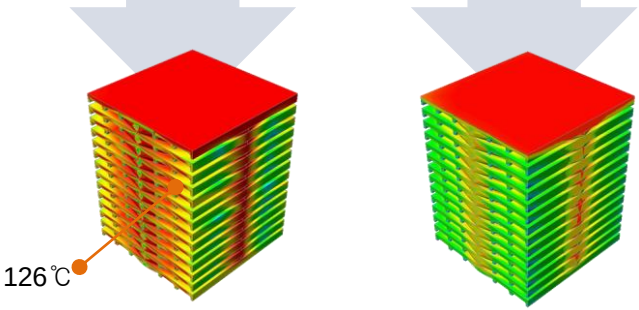


Improved Design
14 large nozzles





Temperature distribution on inner air flow



126℃

Temperatures on processed item

Resonance avoidance design of ultra large AC servo robot

Overview

Objective

Evaluation of results to control if resonance happens during the operation of servo robot.

Safety evaluation of the robot structure when earthquake happens.

Analysis Type

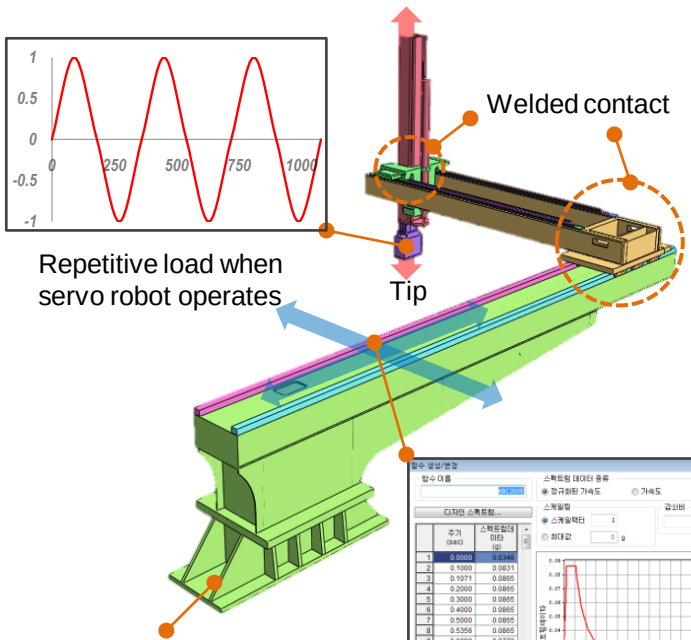
- Linear static analysis
- Mode analysis
- Frequency response analysis
- Response spectrum analysis

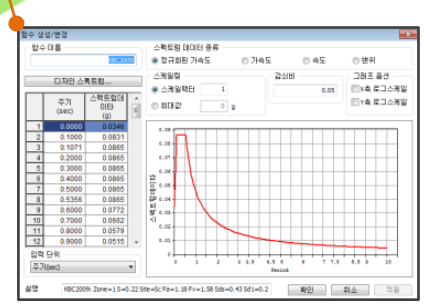
Pre Processing

Modeling: 3D Solid Element & Mass Element

Main loads:

- Self weight: safety evaluation under self load
- Repetitive load: resonance review under harmonic loads
- Seismic load: response spectrum according to KBC2009



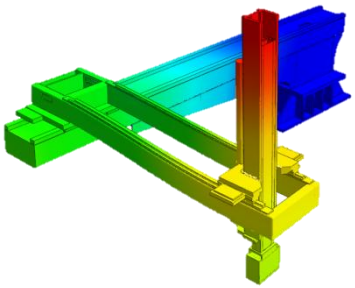


X,Y vibration according to KBC2009 design response spectrum

Post Processing

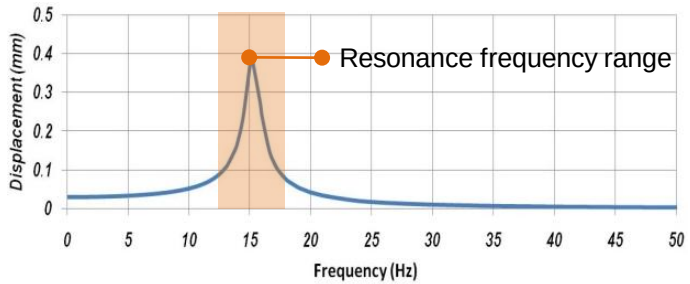
Mode analysis : Evaluate vibration characteristics of servo robot

Mode	Natural Frequency
1st	6.0 Hz
2nd	8.2 Hz
3rd	14. 6Hz



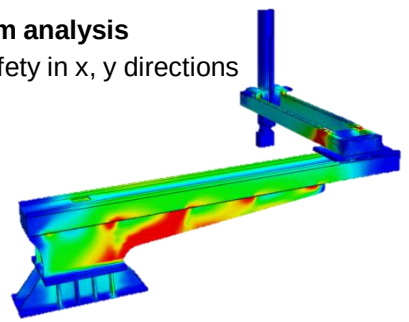
Frequency response analysis

- Resonance happens at 15 Hz under repetitive load.
- Resonance avoidance design is necessary



Response spectrum analysis

- Review seismic safety in x, y directions



Stress distribution under seismic load

Metal sheeting forming analysis

Overview

Objective

Analysis of deformations and stresses of a pipe at final state and different stages during a metal forming process

Analysis Type

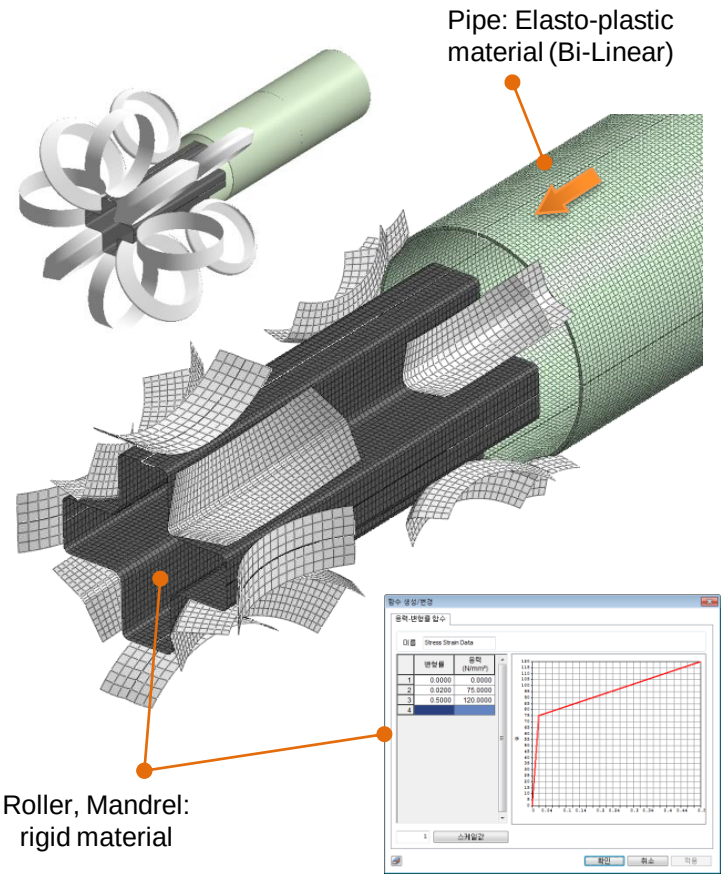
- Implicit nonlinear dynamic analysis

Pre Processing

Modeling: 2D Shell Element

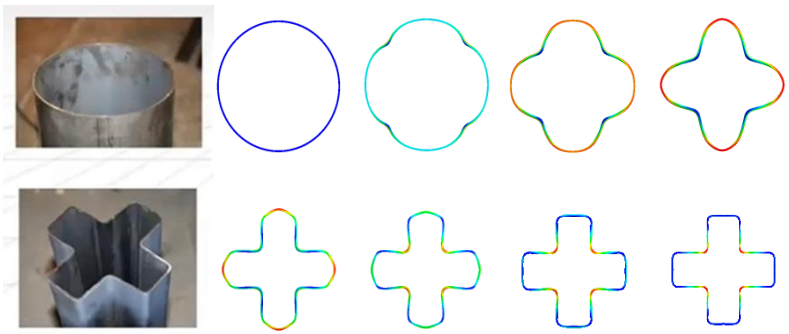
Contact condition:
- General contact (solver contact condition)

Analysis condition:
- ¼ symmetric model to save time
- Insert metal pipe to the roller: forced displacement

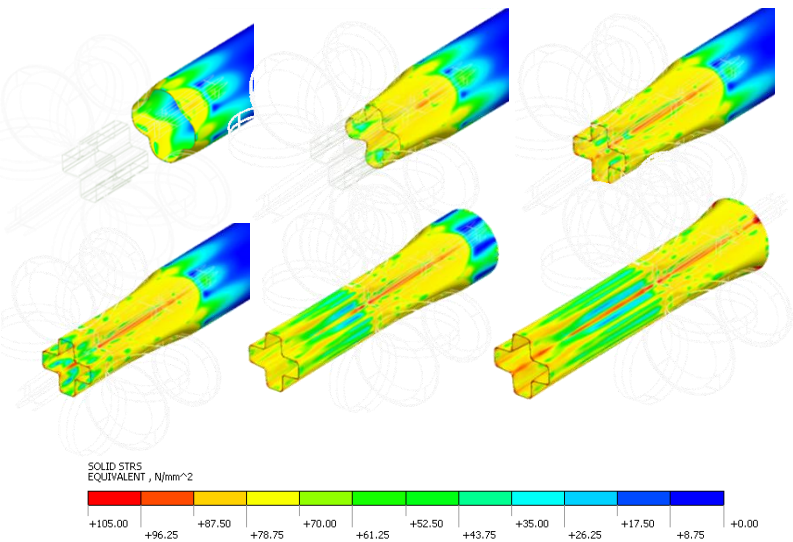


Post Processing

Shapes in different stages



Stress results



Jetting performance evaluation of a nozzle

Overview

Objective

Determination of jetting uniformity and force analysis of the nozzle

Analysis Type

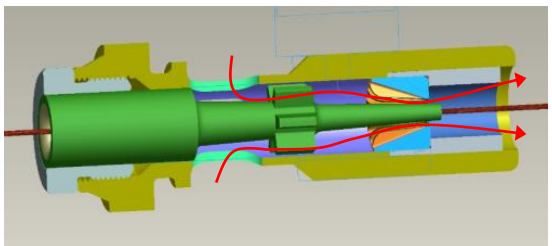
- Fluid flow analysis
- Multi-phase flow analysis

Pre Processing

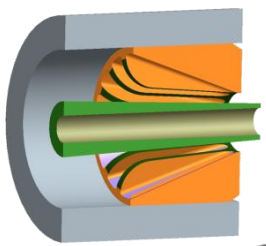
Modeling: 3D Solid Element

Initial fluid position: level 1

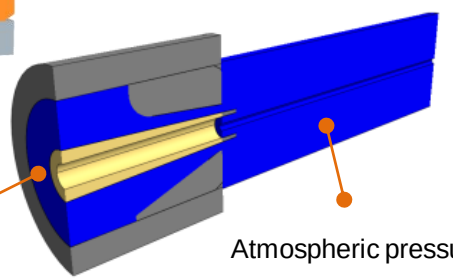
Analysis condition:
- Inlet: pressure
- Outlet: atmospheric pressure
- Surface: wall distance



Fluid path



CFD modeling area



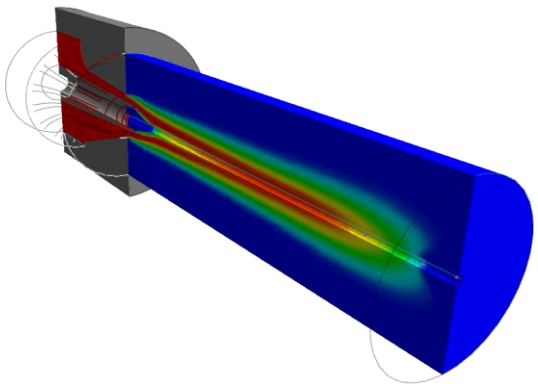
Inlet: pressure

Atmospheric pressure

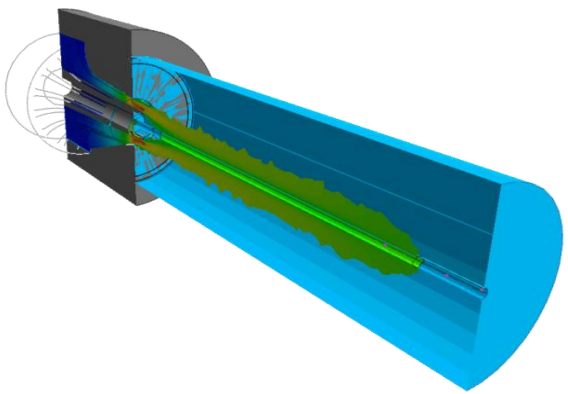
Post Processing

Evaluate nozzle jetting performance

Evaluate nozzle jetting force through velocity and pressure distribution



Level (red: liquid, blue: air)



Temperature distribution inside the system

Safety evaluation and optimum design of tidal power equipment

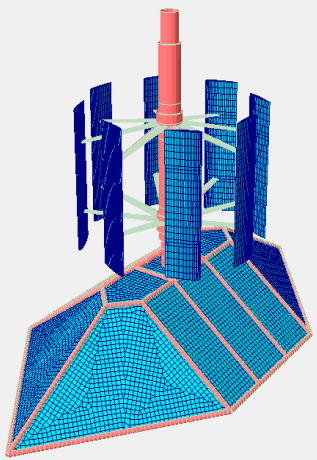
Overview

Objective

Safety evaluation and optimum design of equipment parts (bolt and blade stiffener)

Analysis Type

- Linear Static Analysis
- Direct Time Response Analysis



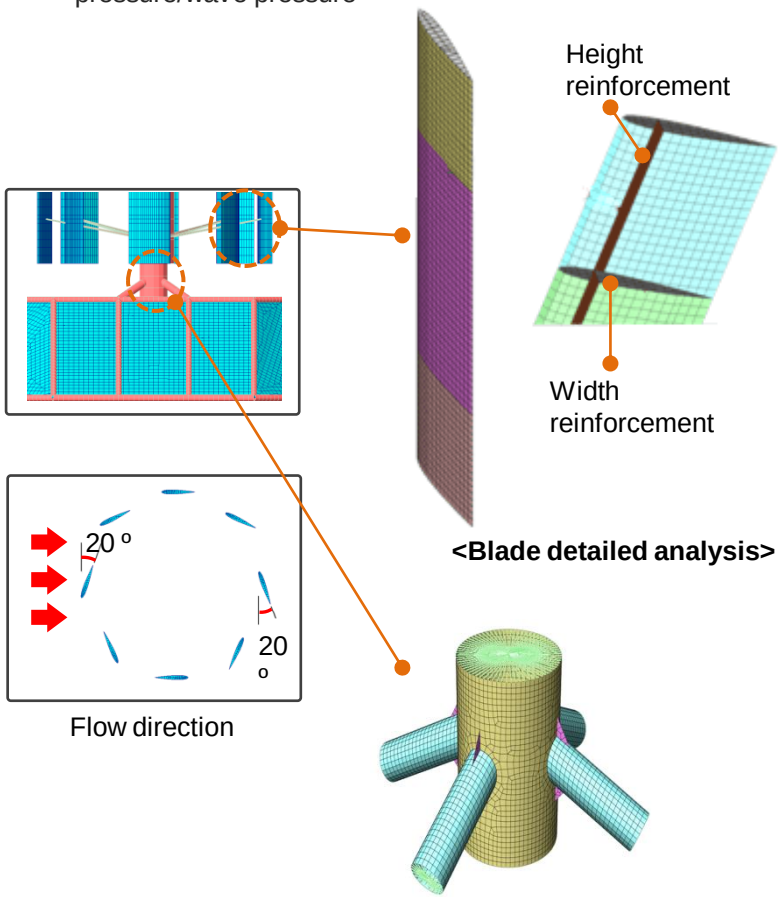
Pre Processing

Modeling:

- Assembly state : Bolt assembly
- Blade Type : NACA 0012

Loading Condition:

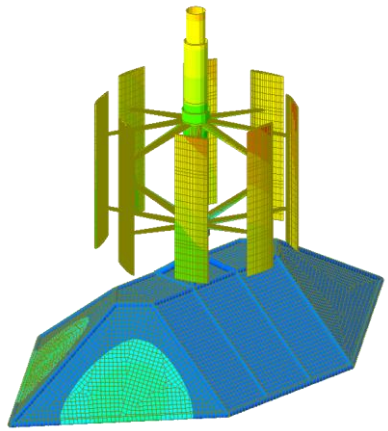
- Self weight/ buoyancy/ hydrostatic pressure/ hydrodynamic pressure/wave pressure

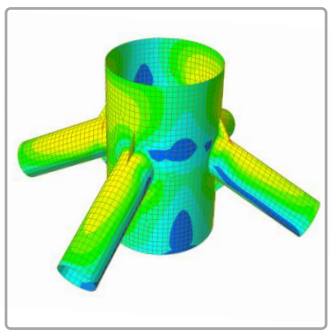


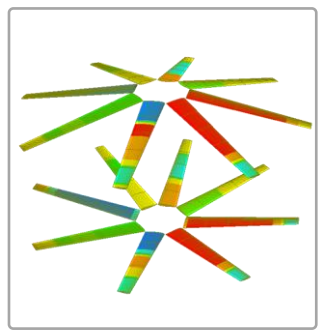
<Blade detailed analysis>

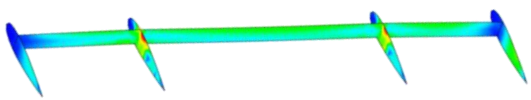
<Joint detailed analysis>

Post Processing









Safety evaluation of tank structure according to inner flow

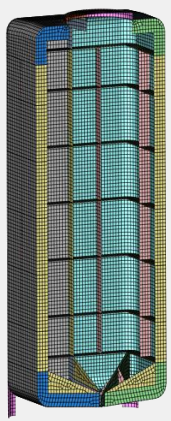
Overview

Objective

CFD analysis and reinforcement design for a transportation use fluid storage tank to ensure its structural safety under vibration and dropping impact conditions

Analysis Type

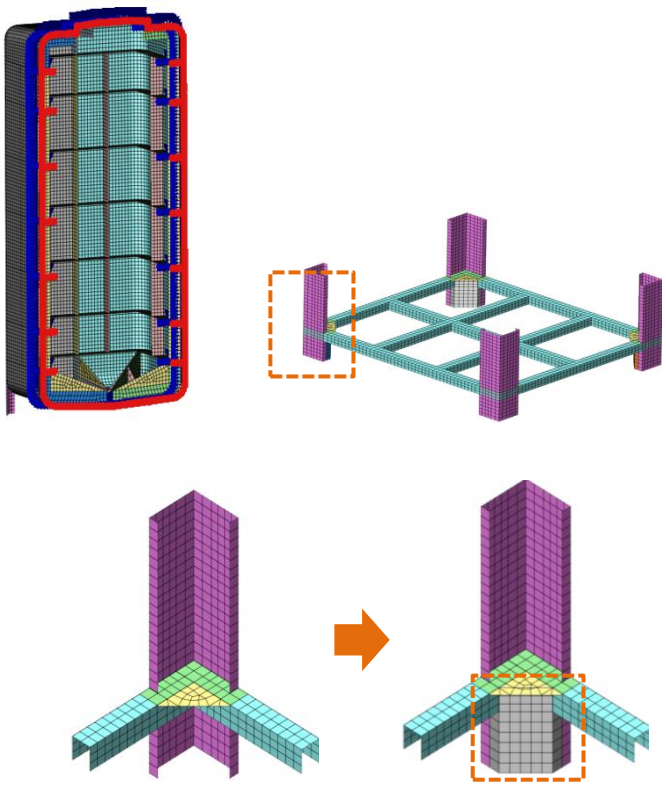
- CFD analysis
- Dynamic response analysis



Pre Processing

Analysis Condition:

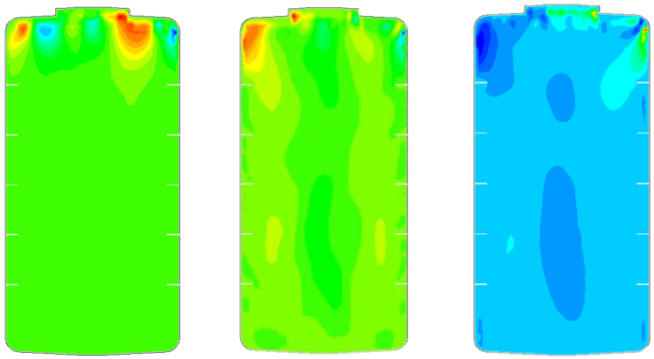
- Fix condition for lower part of the tank
- Symmetric boundary conditions on the cross section



Design optimization through local reinforcement

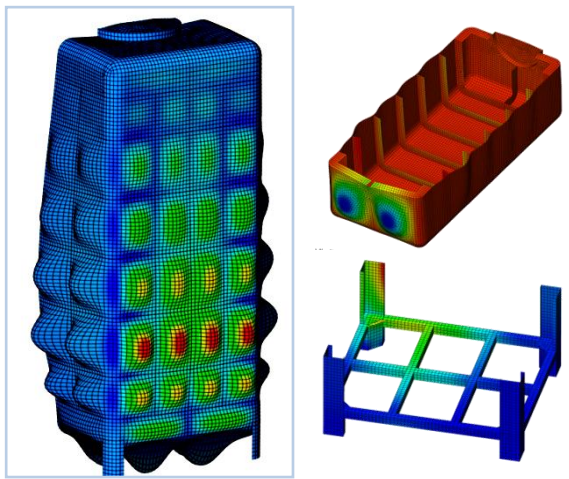
Post Processing

CFD Analysis:



Velocity distribution inside the tank according to time

Dynamic Analysis:



Displacement and stress distribution

Performance and safety review of boiler short soot wall blower

Overview

Objective

Efficiency comparison of fluid jetting before and after design modification of nozzle head.

Structural safety evaluation of redesigned nozzle based on CFD analysis results.

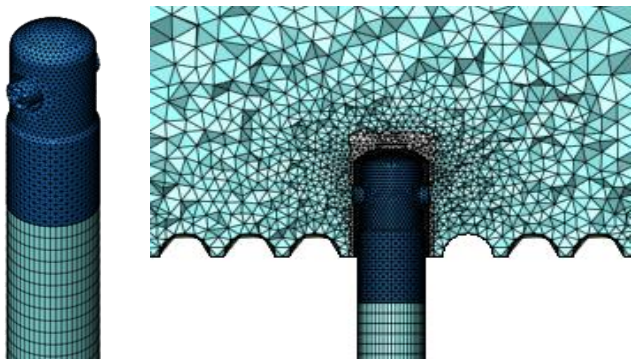
Analysis Type

- CFD analysis
- Linear static analysis

Pre Processing

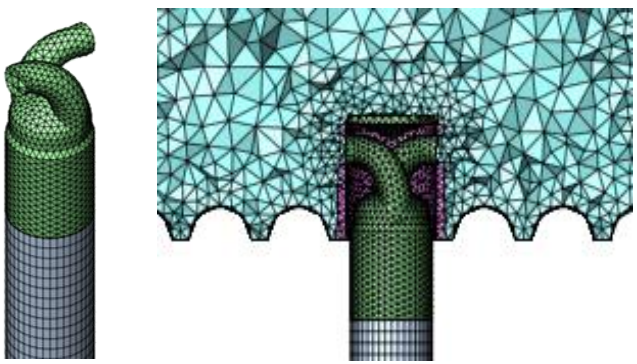
Original Design:

- Flux : 7 ton/hr
- Pressure : 12kgf/cm2



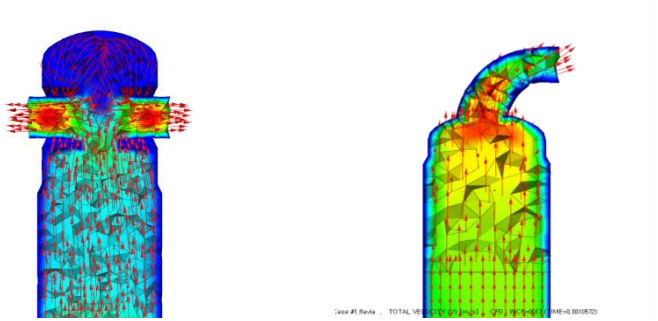
Original Design:

- Same flux, pressure
- Nozzle Head shape changed



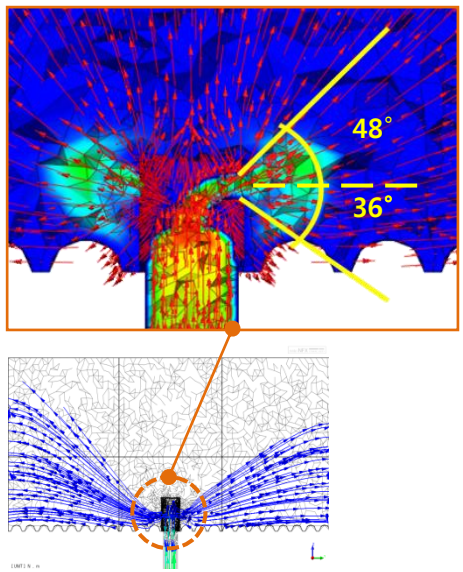
Post Processing

CFD Analysis:

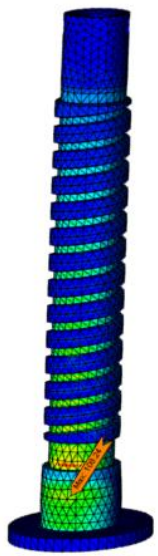


Velocity distribution of different designs

Linear Static Analysis:



Fluid flow path



Stress Distribution
Max. 106.24 MPa

Safety evaluation and dynamic characteristics review of a trailer tank

Overview

Objective

Structural safety evaluation of a trailer tank under self weight and pressure load.

Analyze dynamic characteristic of the tank under irregular dynamic loading during trailer movement

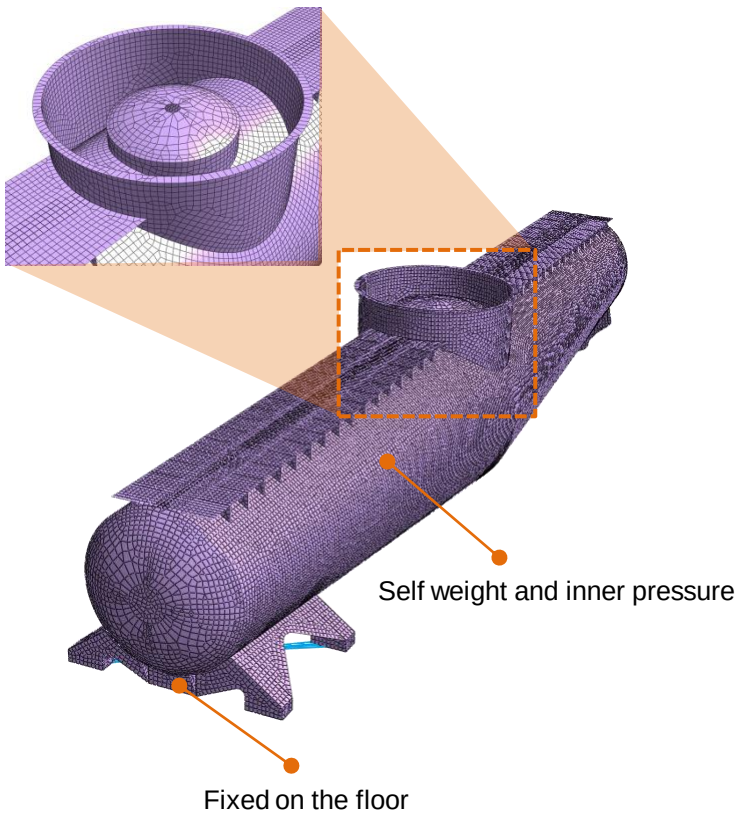
Analysis Type

- Linear static analysis
- Modal analysis
- Linear Dynamic analysis

Pre Processing

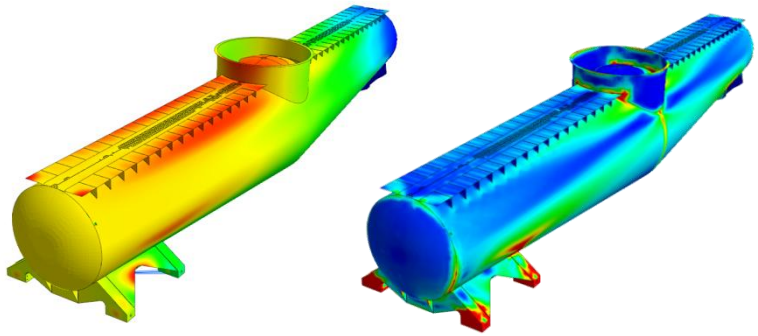
Modeling : 2D Quadrilateral Elements & Rigid Elements

Analysis Condition
- self weight of tank
- Inner pressure



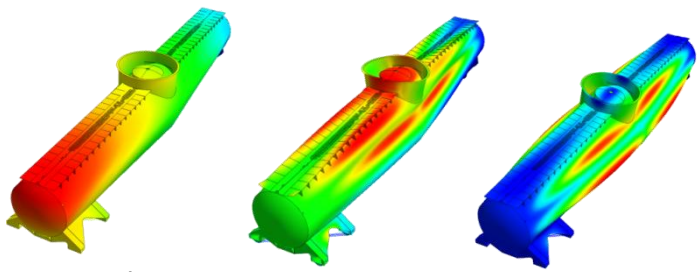
Post Processing

Deformation and stress distribution analysis under self weight and inner pressure

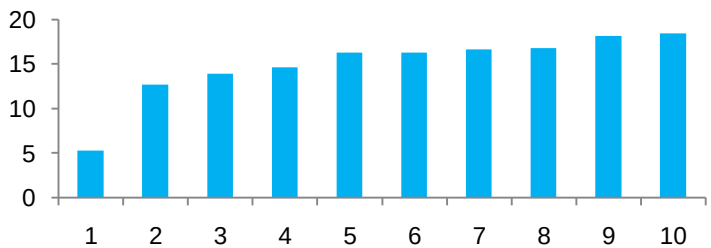


deformation shape Stress distribution

Dynamic characteristics review using modal analysis



1st Mode 2nd Mode 3rd Mode



Electronics

Cooling performance improvement of MOSFET heat sink

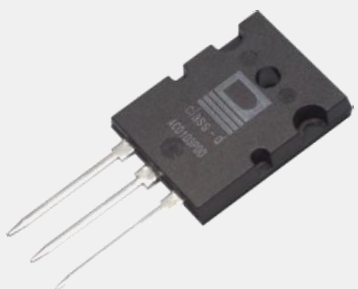
Overview

Objective

Prevention of the damage of MOSFET and improve the cooling performance of the stalled heat sink.

Analysis Type

- Transient heat transfer analysis



Pre Processing

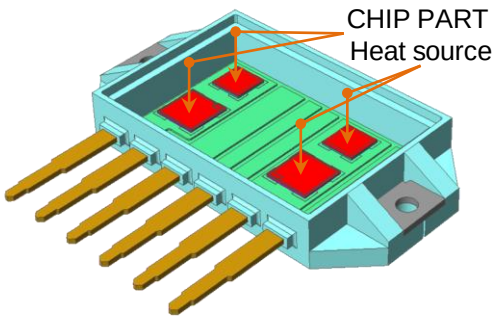
Modeling : 3D Solid Elements

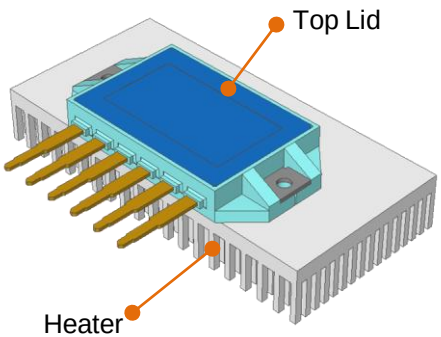
Loading Condition

- Chip part: heat generation load according to time

Design Condition

- Improve cooling performance through increasing heat dissipation area
- Improve cooling performance through changing material





Post Processing

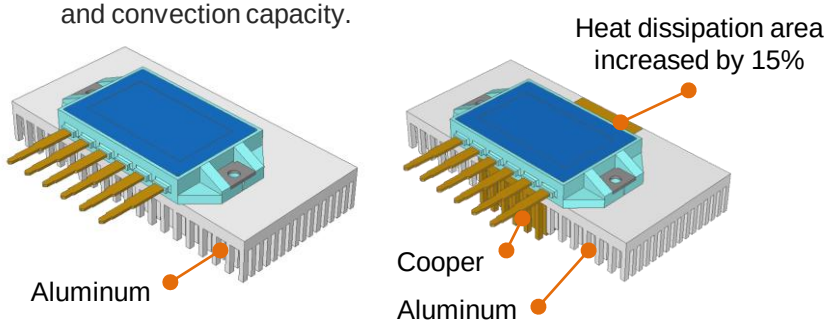
Current design:

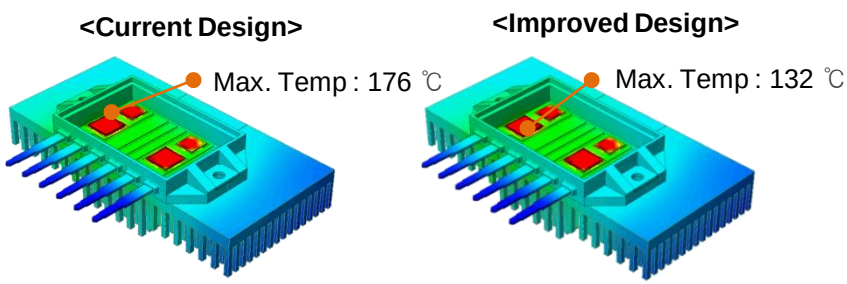
- Maximum temperature : 176 ℃ > allowance: 150 ℃

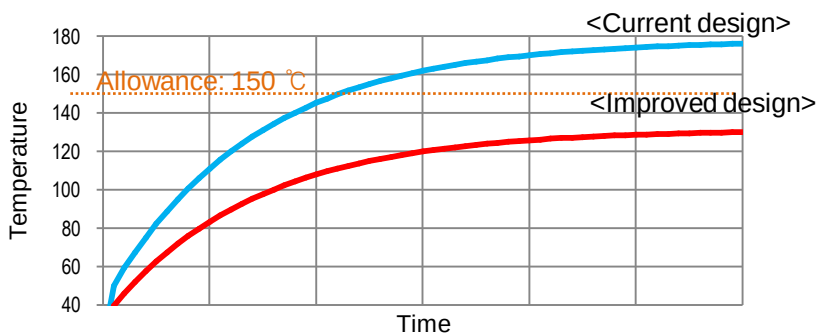
Improved design: increase heat dissipation area + material change (conduction and convection capacity improved)

- Maximum temperature: 132 ℃ < allowance: 150 ℃

Cooling performance is improved by increasing the conduction and convection capacity.







Heat transfer analysis of LED light's PCB

Overview

Objective

Verification of the temperature distribution on LED light's metal PCB part

Analyze heat transfer characteristic of the PCB through simulation method and compare the results with experiments.

Analysis Type

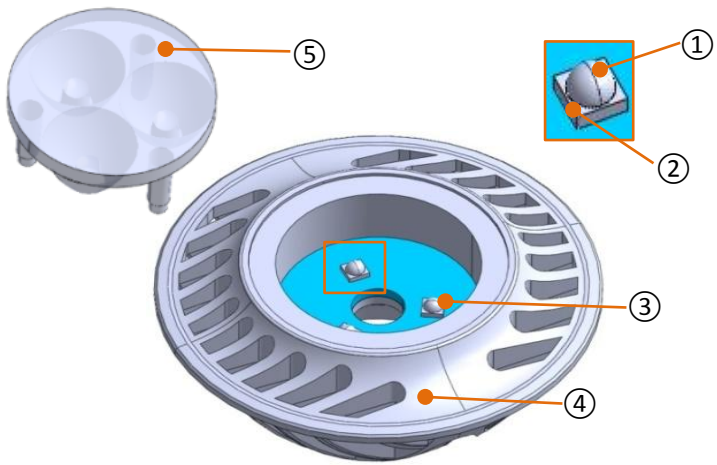
- Steady state heat transfer analysis

Pre Processing

Modeling: 3D Solid Elements

Analysis Condition:

- Convection coefficient : 20 W/m^2 [T]
- LED PKG initial temperature : fixed temperature 120 ℃
- Metal PCB's thermal conductivity: 180 W/m·K
- Identical conditions for simulation and experiment

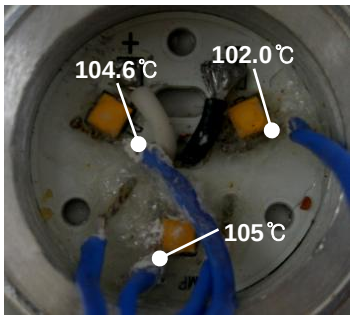


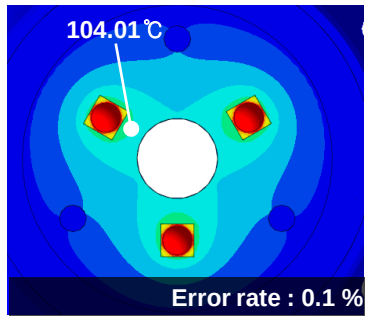
Part Number	Name	Material
①	LED Dome	-
②	LED Base	PCB
③	Metal PCB	Metal PCB
④	Body	AL6061
⑤	Lens	Plastic

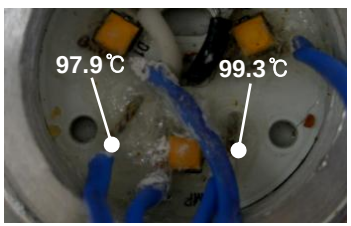
Post Processing

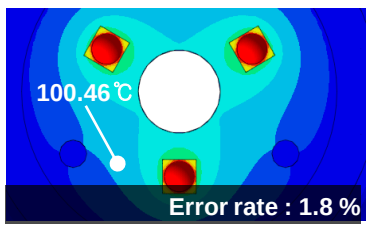
Temperature comparison at important locations between simulation and experiment

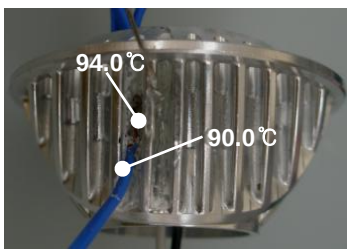
- Minimum error rate 0.1 %
- Maximum error rate 4.4 %
- Maximum error happens at the place where convection occurs

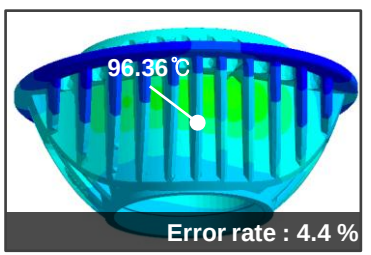












Analyze LED light's thermal performance in the air

Overview

Objective

Analysis of natural convection heat transfer of LED system in the air.

Assessment of LED's thermal safety by measuring the junction temperature.

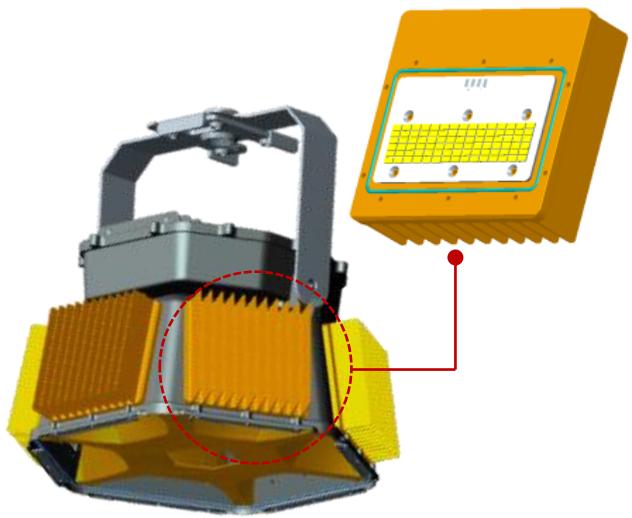
Analysis Type

- Transient flow analysis
- Fluid flow module
- Heat transfer module
- Solid heat transfer module

Pre Processing

Analysis condition:

- Floatability considered through incompressible ideal gas model.
- Inlet (pressure)
- Outlet (pressure)
- Wall condition(no slip)

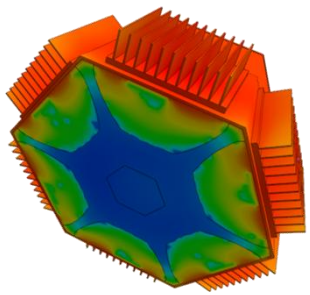


Analysis modeling

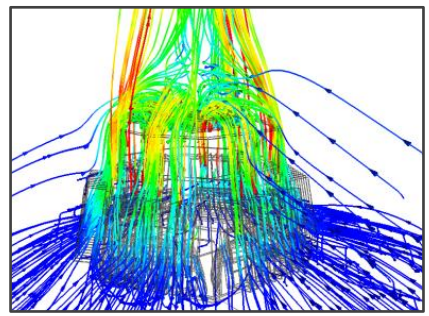
Post Processing

Validation of fluid temperature / speed distribution: performance assessment of heat sink

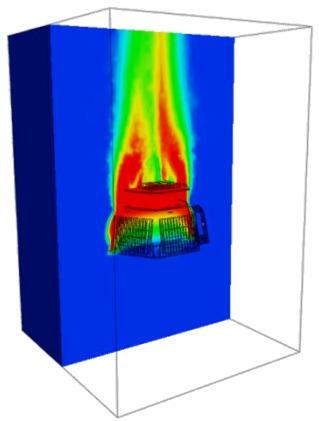
Junction thermal safety assessment by verifying temperature distribution on solid parts.



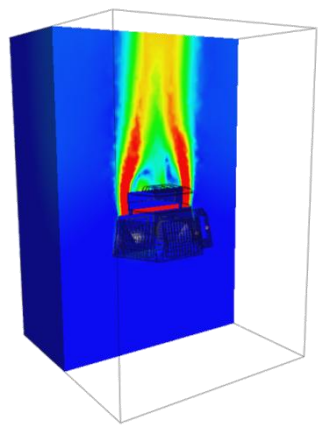
Temperature distribution on solid



Airflow around solid



Temperature distribution on entire model



Flux speed distribution

Safety analysis of contact pin

Overview

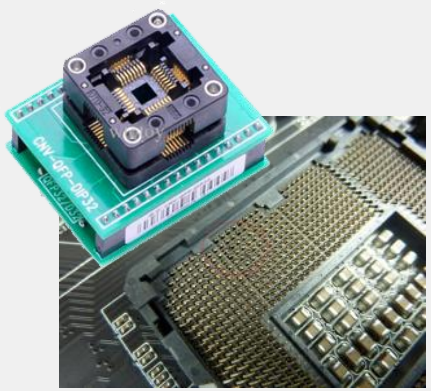
Objective

Examination of the safety of a contact pin and contact ball which are inserted in a Burn-in test socket during connection

Verification of the load on the contact ball and the reaction force value (Y-Dir) at the lower part of the slider

Analysis Type

- Material nonlinear analysis
- Nonlinear analysis
- Geometry nonlinear analysis



Pre Processing

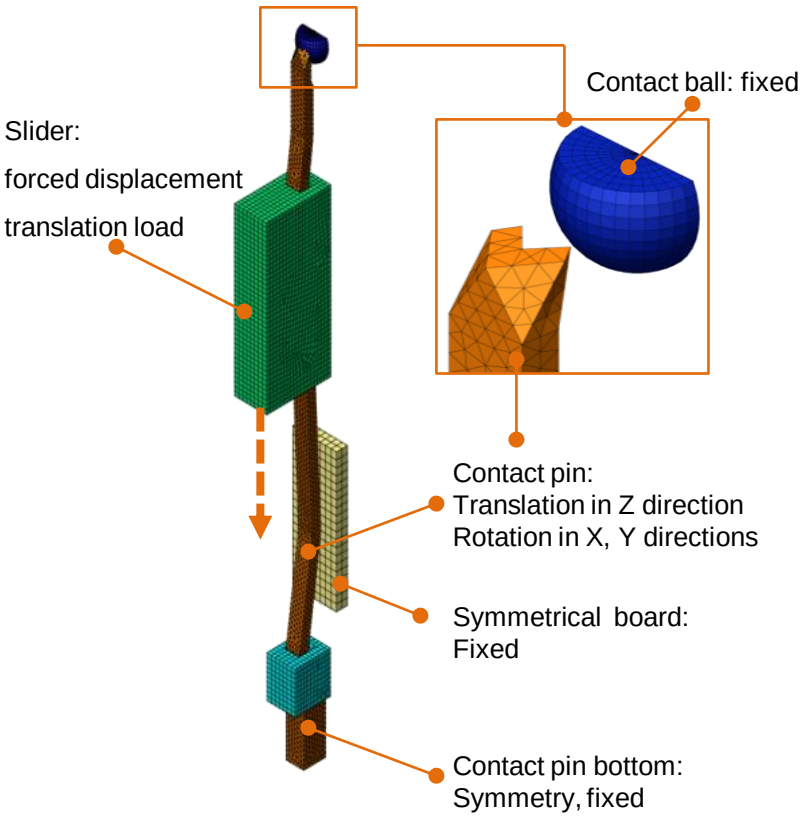
Modeling: 3D solid elements

Boundary Condition

- Sliding between slider and contact pin
- When the slider moves, the contact pin and contact ball cling down

Loading Condition

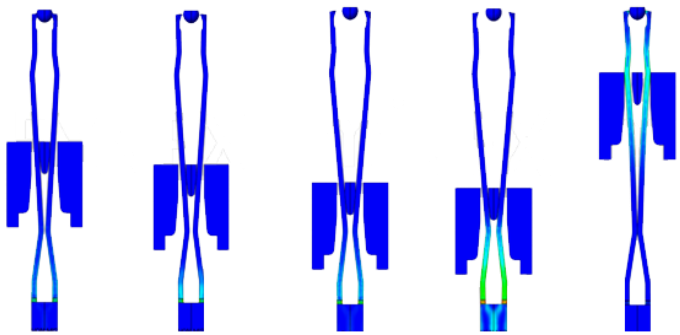
- Forced displacement

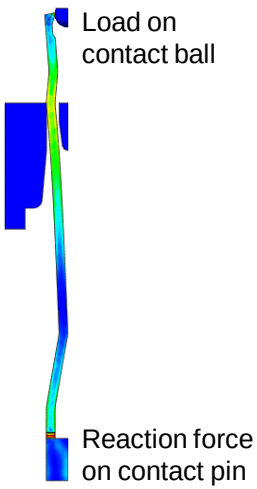


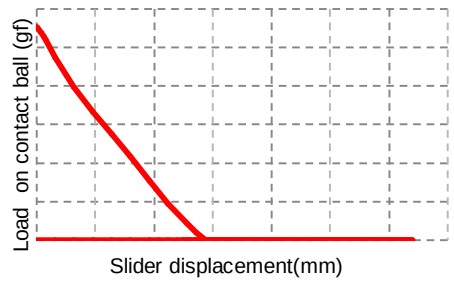
Post Processing

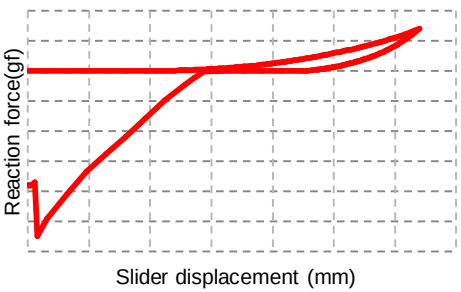
When the contact pin reaches its maximum, the contact condition between the slider and the contact pin disappears . Verify stresses during this spring-back phenomenon.

The maximum stress occurs at the bottom of the contact pin, and the stresses at this point are higher than the yield stress. Occurrence of plastic deformation can be predicted.









Safety analysis of cell phone connector during connection / separation

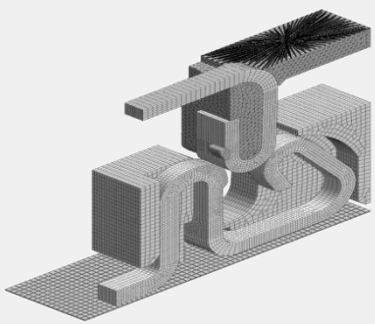
Overview

Objective

Examination of the safety of a cell phone connector by analyzing the maximum stress, residual stress and plastic deformation during connection and separation

Analysis Type

- Material nonlinear analysis
- Nonlinear analysis
- Geometry nonlinear analysis

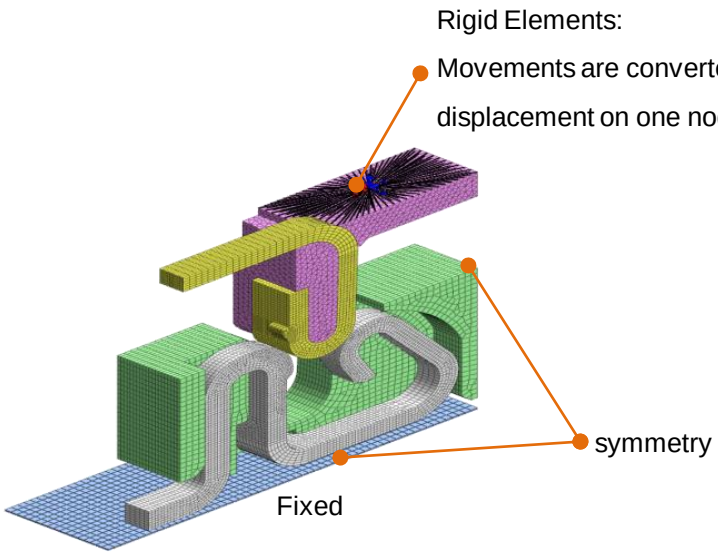


Pre Processing

Modeling : 3D solid elements

Boundary Condition:
- Sliding occurs between Male Terminal & Female Terminal
- Sliding occurs between Female Terminal & Plate

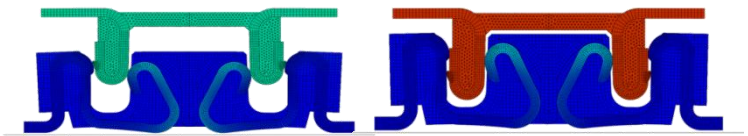
Loading Condition
- Forced displacement of 4 times material thickness in y direction



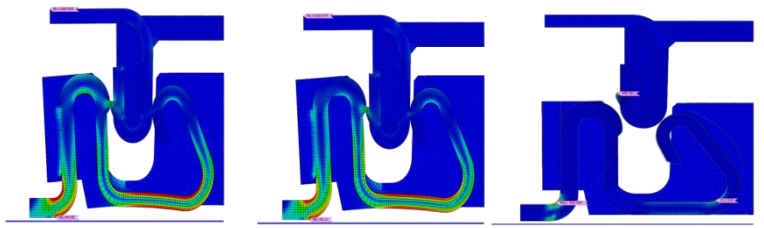
(1) **Connection**: Female Terminal moves toward -Z direction
(2) **Separation**: Female Terminal moves toward +Z direction

Post Processing

Stress and residual stress during connector's connection and separation

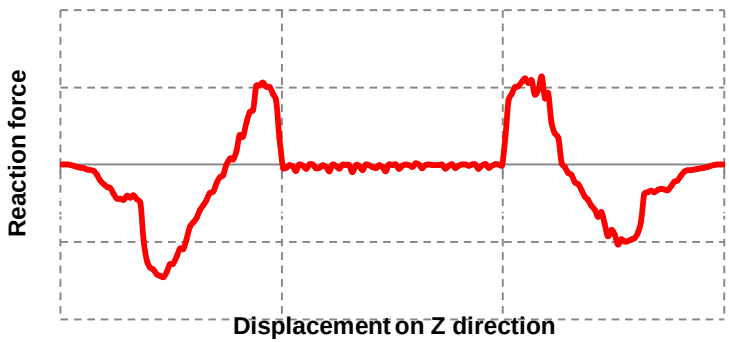


Displacement



Maximum Stress (Connection) | Maximum Stress (Separation) | Maximum Residual Stress

Verify reaction force value on the contact face



Offshore

Lifting Pad-Eye design of offshore plant

Overview

Objective

Design and safety review of Pad-eye during lifting work

Analysis Type

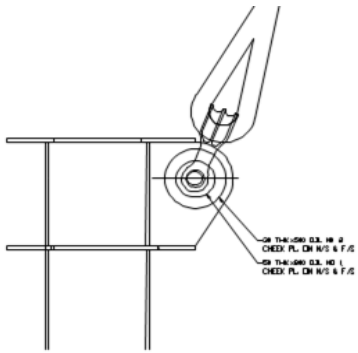
- Nonlinear static analysis

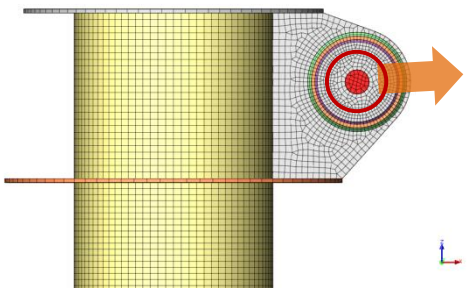
Pre Processing

Modeling: 3D Hexa Solid Element

Analysis Condition:

- Pad-eye design and safety evaluation during lifting work
- Lifting Pad-Eye Design



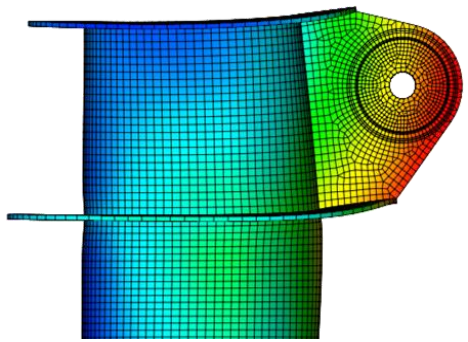


Pad-Eye Model

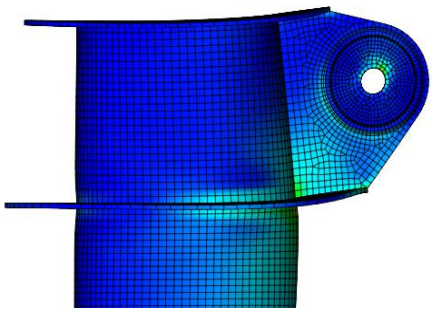
Post Processing

Lifting Pad-Eye analysis result

- Safety review of Pad-Eye structure under loading conditions suggested by API Code
- Main Plate and Cheek Plate, welded joint design



Deformed shape



Stress distribution

Boat Landing impact analysis

Overview

Objective

Impact analysis of boat landing area when the boat pulls into the shore

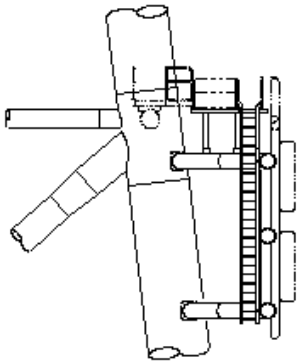
Analysis Type

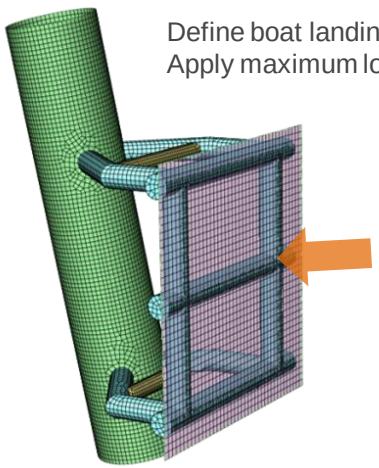
- Nonlinear static analysis
- Impact analysis

Pre Processing

Boat Impact Analysis

- Safety evaluation of boat Landing when the boat comes alongside





Define boat landing as rigid.
Apply maximum load.

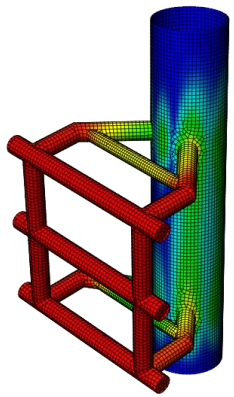
Impact condition

- Impact force: 3,000 DWT
- Impact speed: 1knots
- Side impact

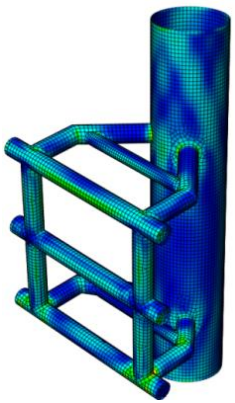
Post Processing

Boat Impact Analysis Result

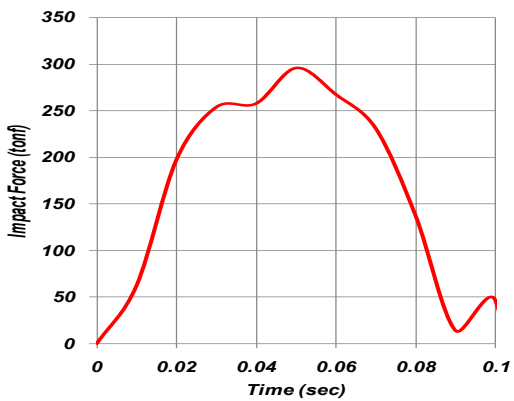
- Safety review of boat landing structure during impact
- Calculate the impact load value when the boat comes alongside.



Deformed shape
(maximum impact load)



Stress distribution
(maximum impact load)



Impact load according to time

Automotive

Radiation heat transfer analysis of a automotive lamp

Overview

Objective

Measure the temperature distribution at lens surface 2 hours after the bulb is turned on

Analysis Type

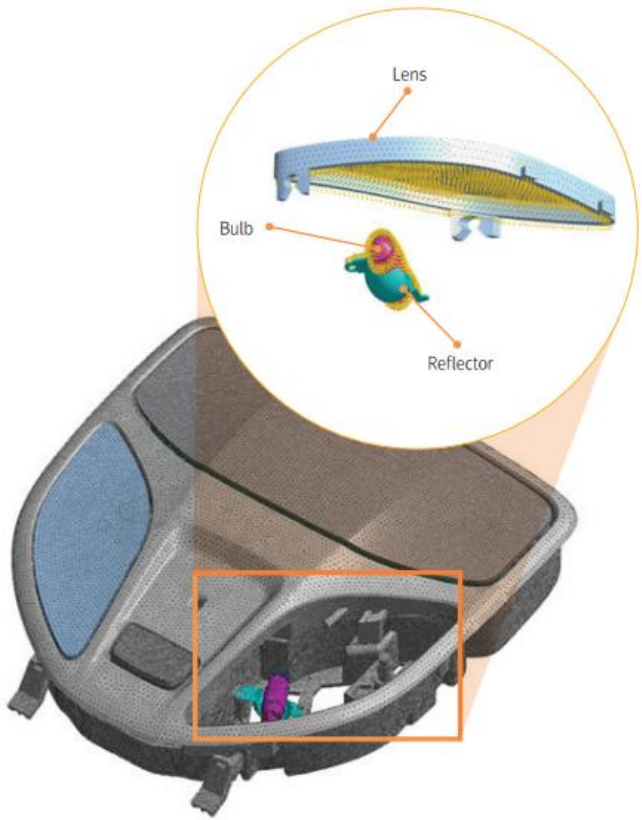
- Transient heat transfer analysis

Pre Processing

Modeling : 3D Solid Element

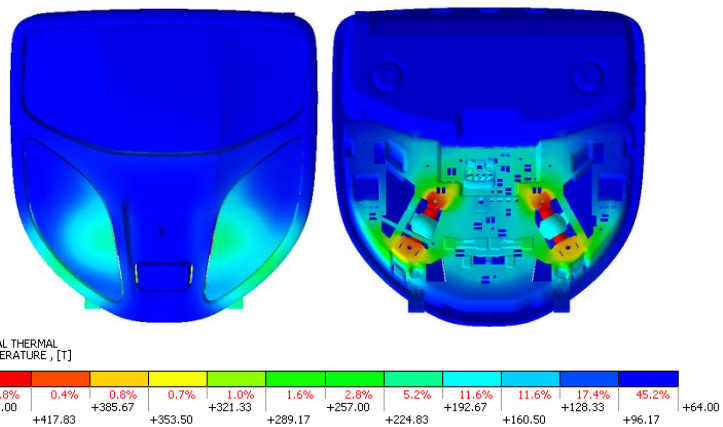
Contact condition :
- Welded contact between all the parts

Analysis condition:
- Generation (W/m^3)
- Bulb, Reflector, Lens : cavity radiation condition
- Convection: no wind (natural convection)



Post Processing

Measure temperatures at same locations on the lens surface as the experimental test.



Name	Ambient temp : 30℃		Ambient temp : 80℃	
	analysis	test	analysis	Test
Lens	58	50.4	90	88.1
Reflector	136	135.7	171	174.2

Drop analysis of automotive relay

Overview

Objective

Analysis of damage of the automotive relay during vehicle driving

Review the durability of PCB relay through drop analysis

Analysis Type

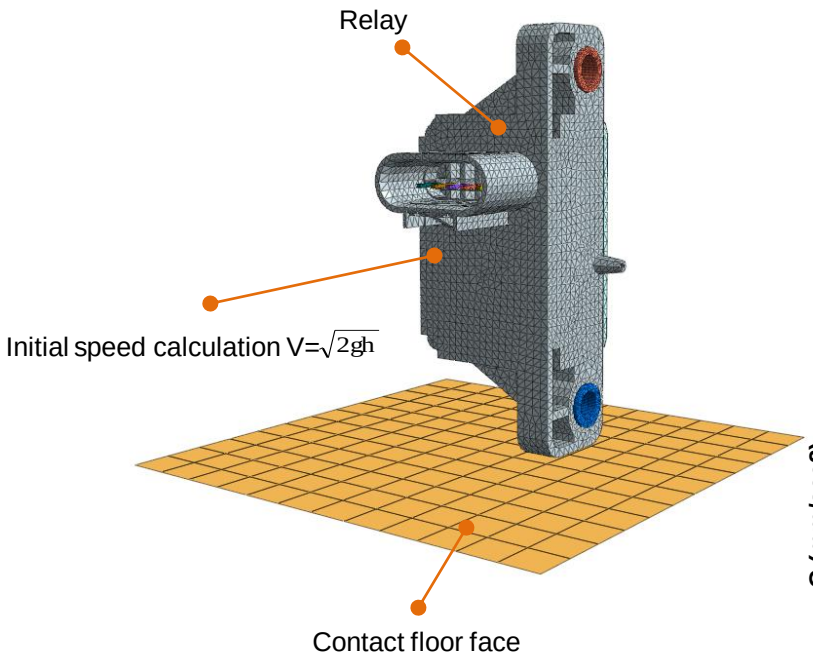
- Explicit nonlinear dynamic analysis

Pre Processing

Modeling : 2D Quad Element & 3D Solid Element

Loading Condition :
- Free drop from 1m height
- Apply initial speed to simulate the free drop
- Self weight

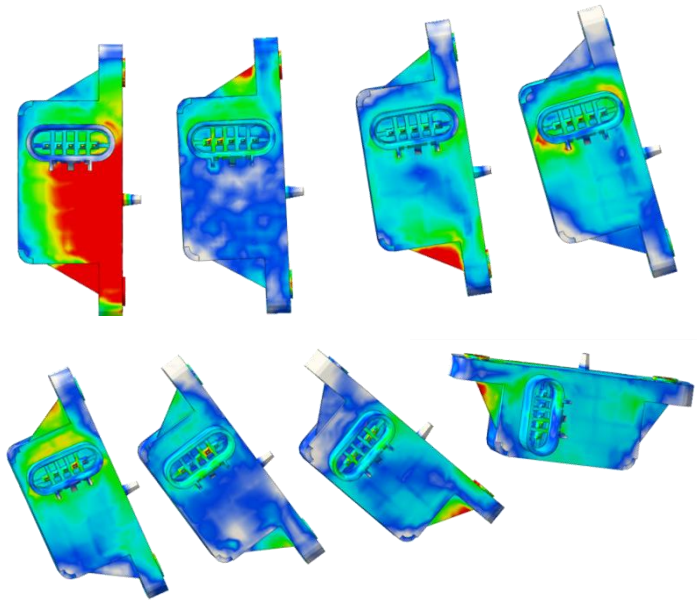
Boundary condition:
- PCB Module : Welded contact
- Relay / floor: General contact

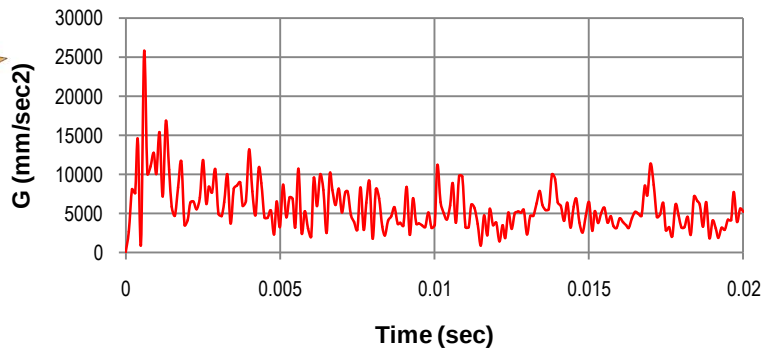


Post Processing

Review deformation of PCB relay according to time (after contact with the floor)

Measure PCB's acceleration according to time.





Graph of PCB speed according to time

Structure safety evaluation of automotive hood latch

Overview

Objective

Structure safety evaluation of automotive hood latch under sudden load.

Analysis Type

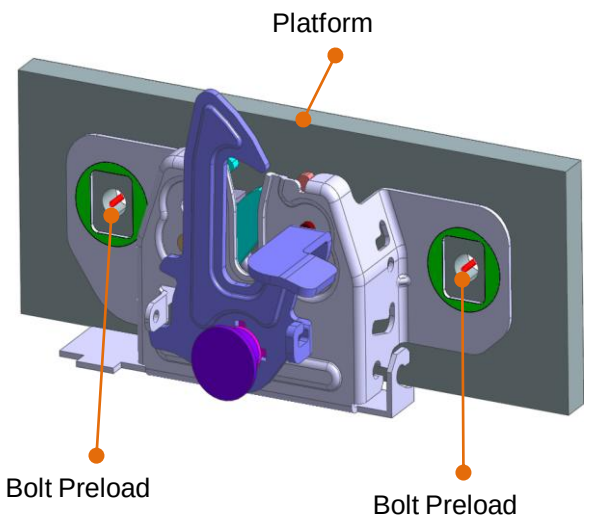
- Nonlinear static analysis

Pre Processing

Modeling :
- 3D Element
- Bolt connection between platform and hood latch. Apply bolt preload.

Loading Condition :
- Apply load at each part of the Hood latch
- Catch part : 6,200N
- Hook Safety Lever Part : 3,500N

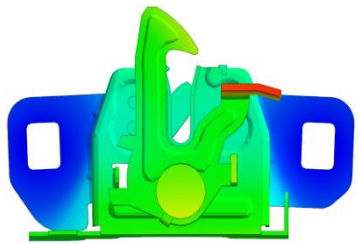
Boundary Condition :
- Between platform and Base , Catch and Pawl, apply general contact.



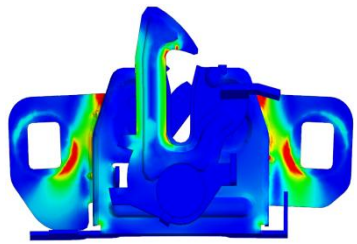
Post Processing

Maximum displacement and maximum stress comparison according to different loads applied to Hood Lever
- Maximum displacement: 2.26mm
- Maximum stress : 132.48MPa

Real deformation happens at the same location as indicated in the analysis result.

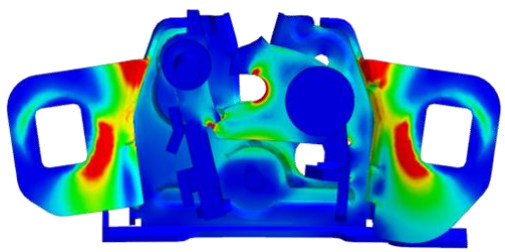


Max displacement: 2.261 mm



Max stress : 132.48 N/mm^2

Real test result



NFX analysis result (stress distribution)

Deformation capacity analysis of automobile brake hose

Overview

Objective

Analysis of deformation capacity of brake hose considering material properties

Analysis Type

- Nonlinear analysis
- Material nonlinearity: hyper-elastic model

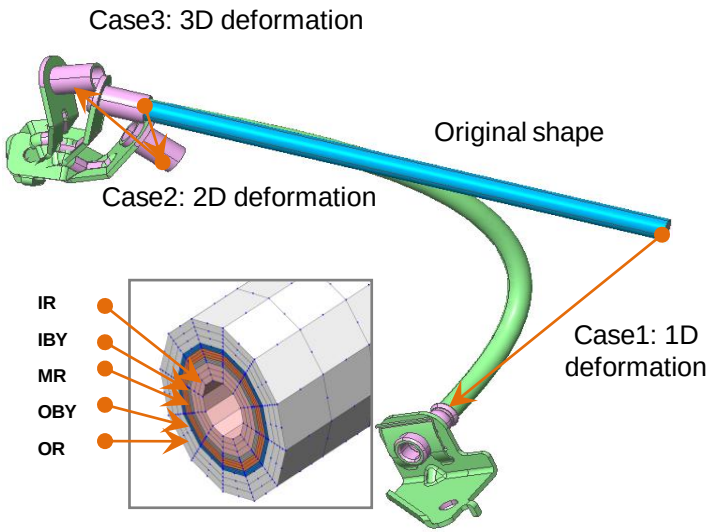
Pre Processing

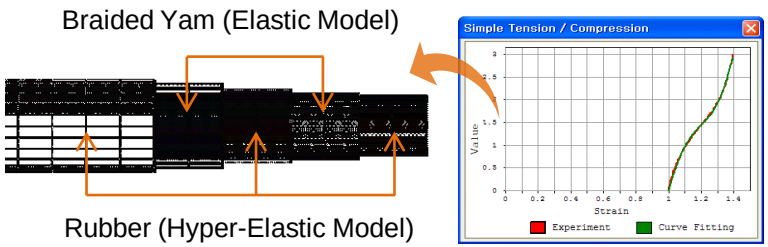
Modeling:

- Rubber : 3D Hexa Element (hyper-elastic model)
- Braided Yam : 3D Hexa Element (equivalent properties)

Analysis condition:

- Deformation of the brake hose during installation
- Deformation of the brake hose during operation

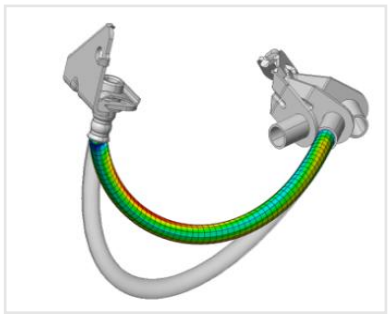




Stress-Strain Curve Fitting
(Mooney-Rivlin constants auto-calculation)

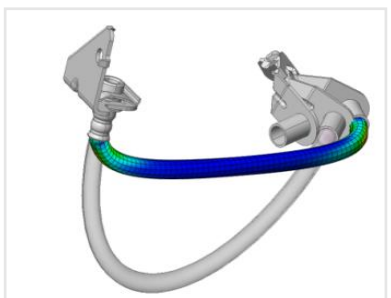
Post Processing

Check maximum / minimum stresses under cases of 1D~3D deformation



Case-1	Von Mises Stress (MPa)
Max.	5.13E-01
Min.	7.38E-03

Case-2	Von Mises Stress (MPa)
Max.	9.06E-01
Min.	2.01E-03



Case-3	Von Mises Stress (MPa)
Max.	1.60E-00
Min.	2.62E-02

Instrument panel head impact analysis - ECE R21

Overview

Objective

Control the injury value of passenger's head when impacting with the vehicle instrument panel, analyses are carried out according to regulations of different countries

Analysis Type

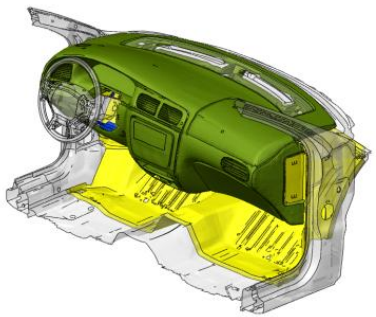
- Nonlinear Explicit Analysis

Pre Processing

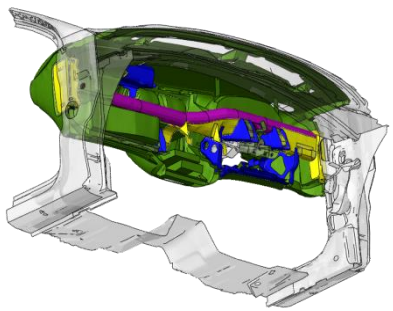
Modeling: 2D Quad Element

Analysis Condition:

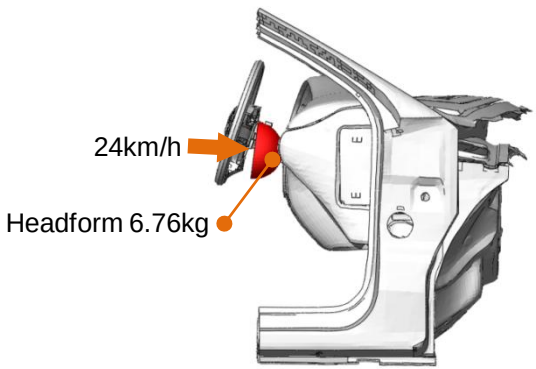
- Impact speed between passenger head and instrument panel is 24km/h
- Head form is around 5~7 kg when impacting with audio instrument.



Instrument panel (front view)
Nodes: 97 032
Elements: 93 950



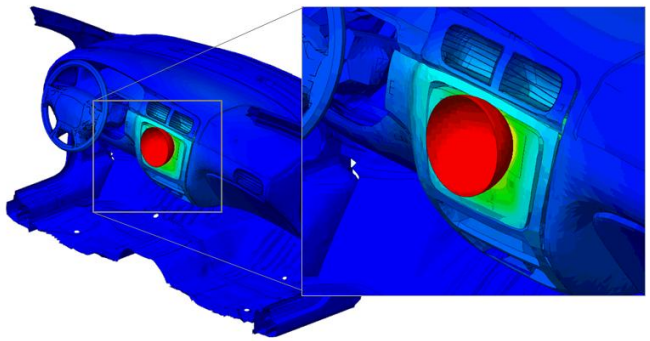
Instrument panel (rear view)



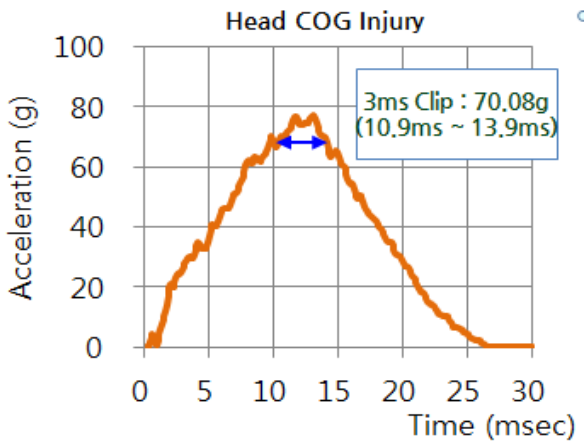
Post Processing

Review the analysis result using 3ms Clip method

- according to regulation, impact should be redistricted under 80G.
- in this analysis, the impact is under 70G, therefore the regulation is complied.



Displacement distribution



Free motion B-pillar head impact analysis - FMVSS201U

Overview

Objective

Control the injury value of passenger's head when impacting with the vehicle interior trim, analyses are carried out according to North American regulation.

Analysis Type

- Nonlinear Explicit Analysis

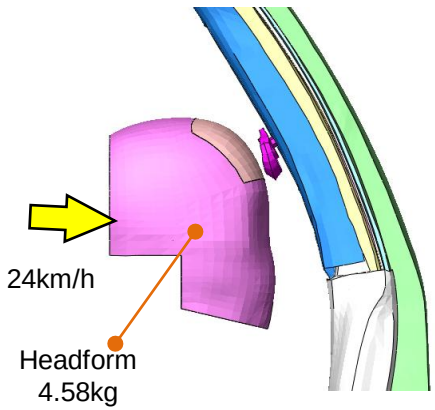
Pre Processing

Modeling: 2D Quad Element

Analysis Condition:

- Impact speed between passenger head and B-pillar is 24km/h
- Head form is around 5~7 kg when impacting with audio instrument.

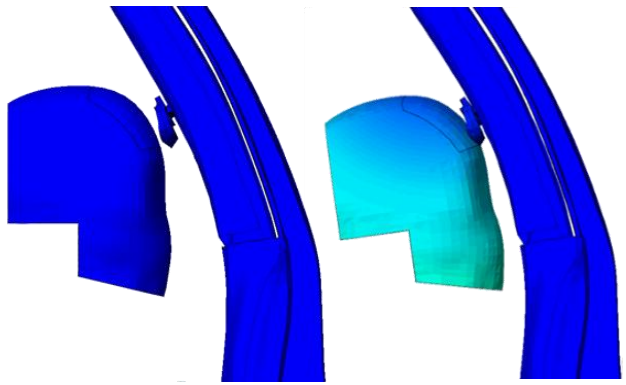
B-pillar model
Nodes: 25850
Elements: 23,537



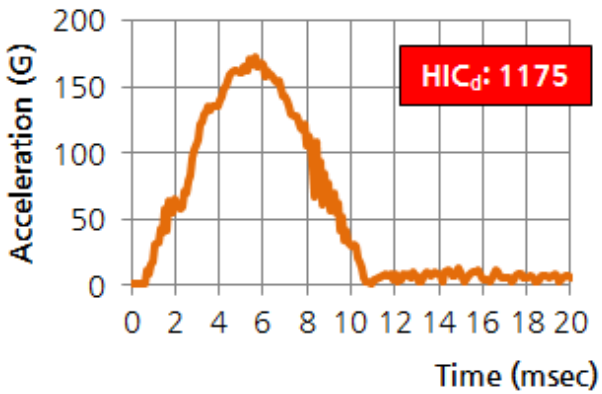
Head B-pillar impact model

Post Processing

The head injury reference value (HICd) is equal to 1175, which is higher than the regulation value (HICd =1000). Therefore the design needs to be improved



Displacement distribution



Pedestrian Protection Child Head Impact Analysis

Overview

Objective

Investigation of the condition when child pedestrian ‘s head accidently hits on the car hood and propose improvement plan.

Analysis Type

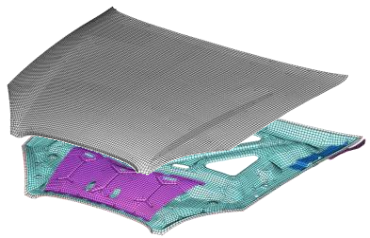
- Explicit nonlinear dynamic analysis

Pre Processing

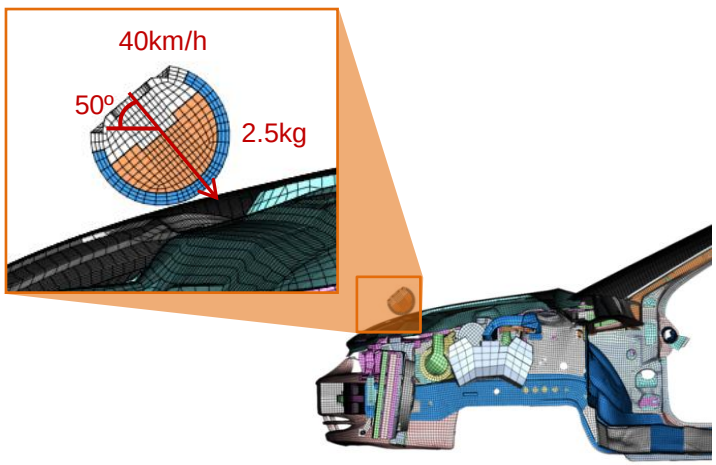
Modeling: 2D Quad Element & 3D Hexa Element

Analysis Condition:

- After the child is hit by the car front and fall forward, head will impact on the car hood
- Consider material nonlinearity for car hood.
- Impact angle: 50°, speed 40km/h



Hood model

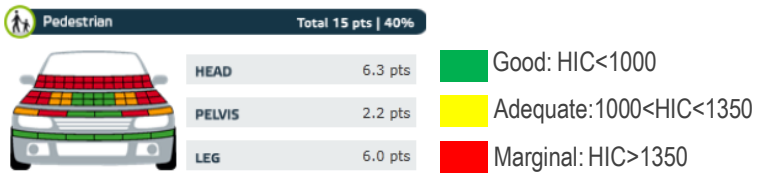


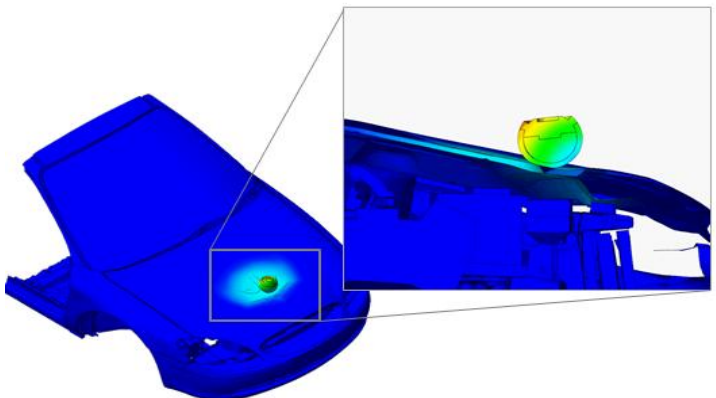
Head - hood impact model

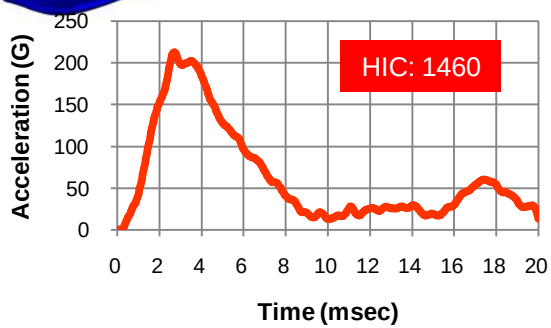
Post Processing

After head model impact test , calculate value of Head Injury Criterion (HIC) and evaluate the result

Through the analysis, HIC is calculated to be 1460 (Marginal). To get a better safety score, design plan need to be improved.







RCAR low speed impact analysis - front

Overview

Objective

Protect ion of important parts of the automotive during low speed impact, evaluate the performance of crashing can.

Analysis Type

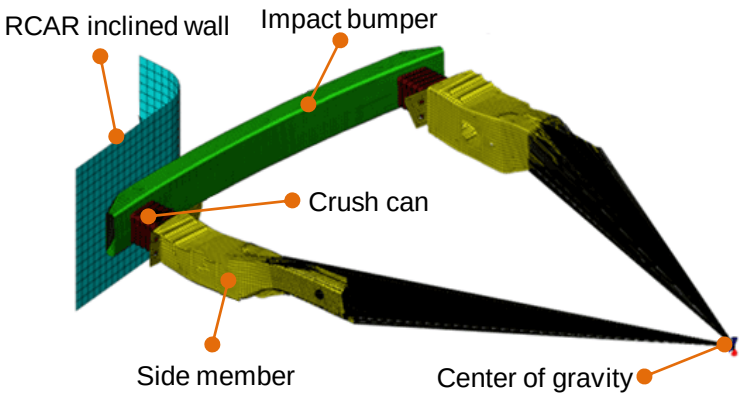
- Explicit nonlinear dynamic analysis

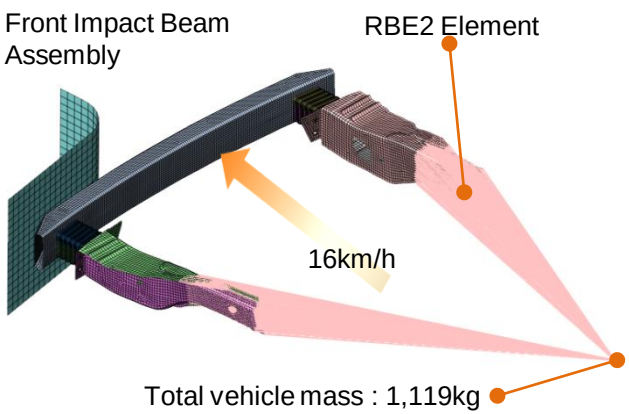
Pre Processing

Modeling: 2D Quad Element & Rigid/Mass Element

Analysis Condition:

- Apply 1119kg to the gravity center.
- low speed impact: 16km/h

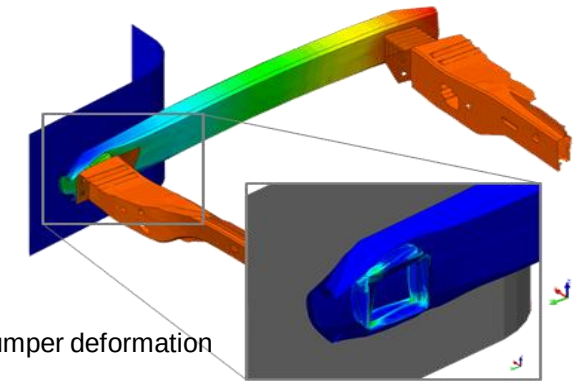




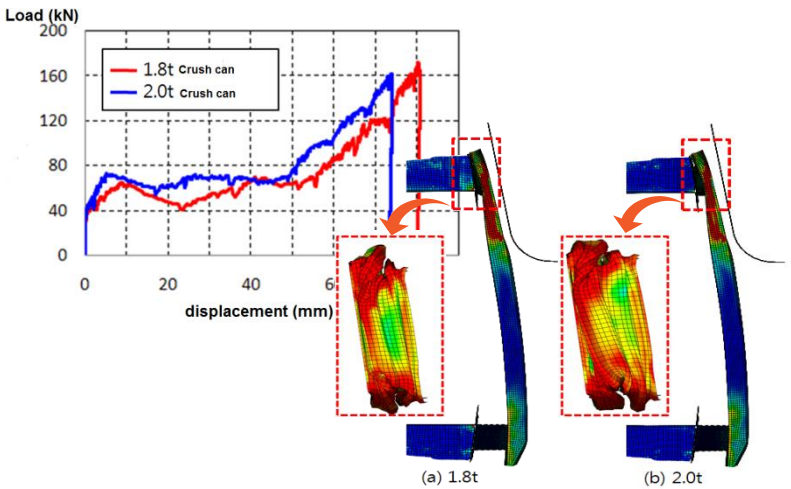
Post Processing

Comparison of analysis result between crash cans of different thicknesses (1.8t and 2t)

Energy absorption capacities of the two crush cans are similar, while the deformation difference appears. Choose the design which protect better side members.



Effective plastic strain (crash can)



Deformation of the front bumper and crush can

RCAR low speed impact analysis -rear

Overview

Objective

Protect ion of important parts of the automotive during low speed impact, evaluate the performance of crashing can.

Analysis Type

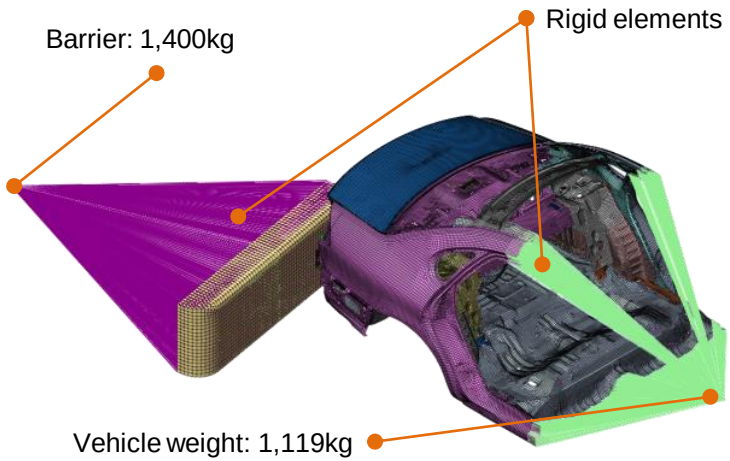
- Explicit nonlinear dynamic analysis

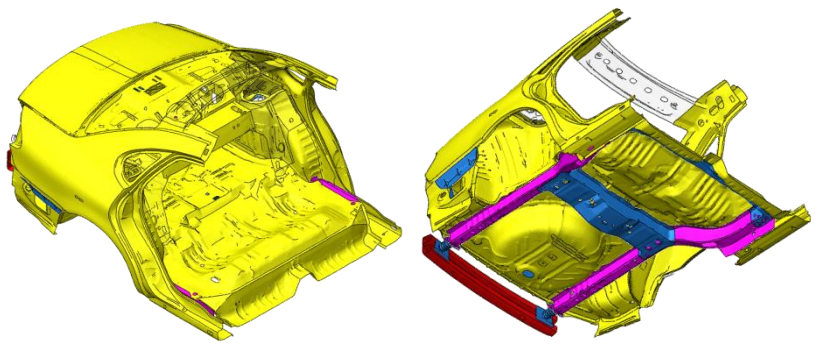
Pre Processing

Modeling: 2D Quad Element & Rigid/Mass Element

Analysis Condition:

- Apply 1119kg to the gravity center.
- low speed impact: 16km/h





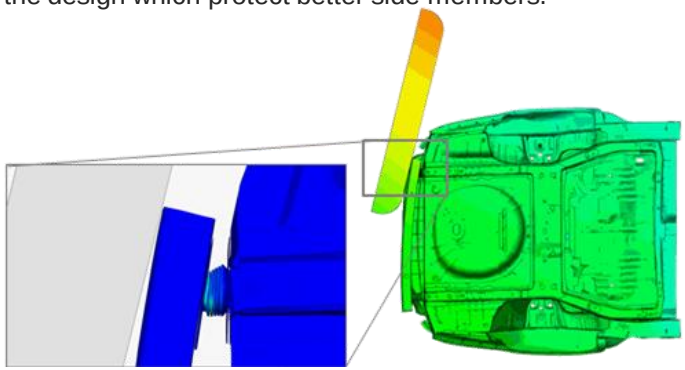
Top view　　Bottom view

Automotive rear model

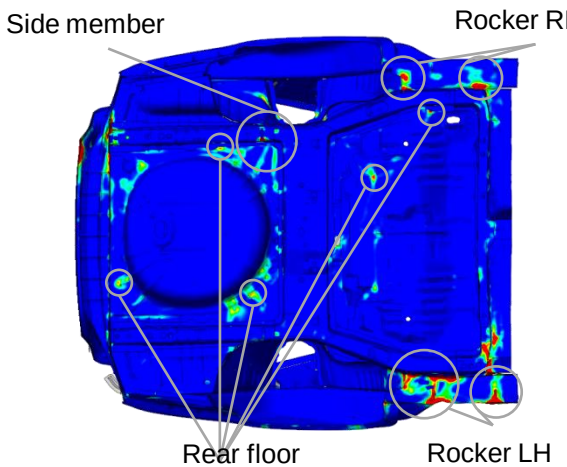
Post Processing

Comparison of analysis result between crash cans of different thicknesses (1.8t and 2t)

Energy absorption capacities of the two crush cans are similar, while the deformation difference appears. Choose the design which protect better side members.



Effective plastic strain (crash can)　　Displacement distribution (bottom view)



Rear structure damage (bottom view)

Seat belt anchorage analysis – ECE R14

Overview

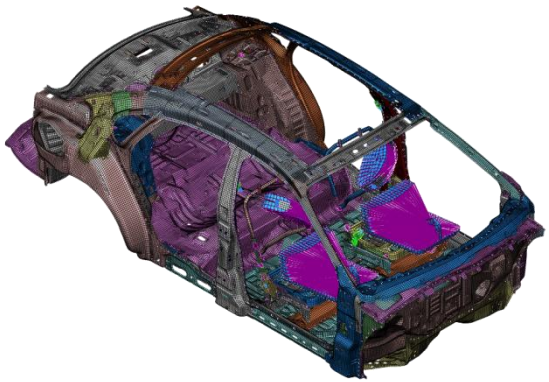
Objective

In order to ensure the reliable performance of the vehicle Seat Belt System under constant load, analysis is carried out according to European regulation.

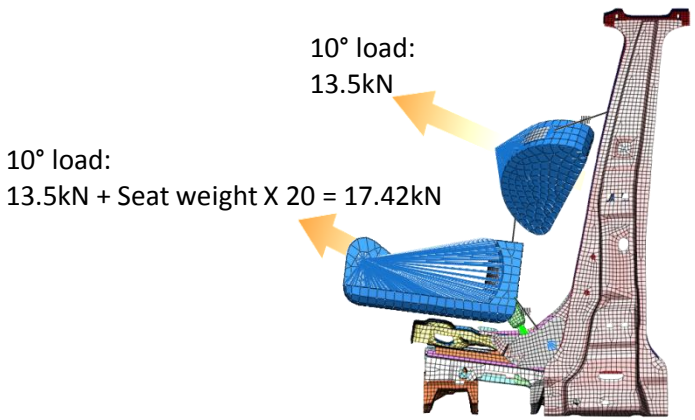
Analysis Type

- Explicit nonlinear dynamic analysis

Pre Processing



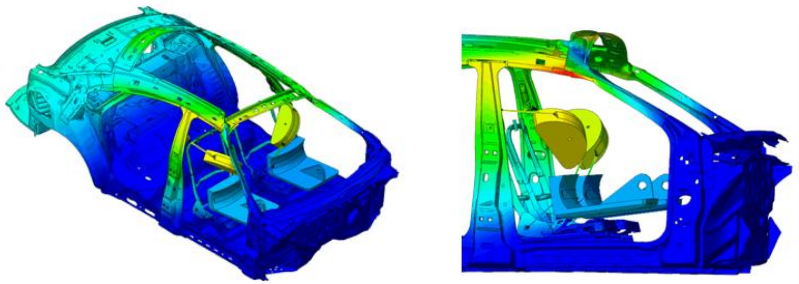
BIW & Seat assembly FE model



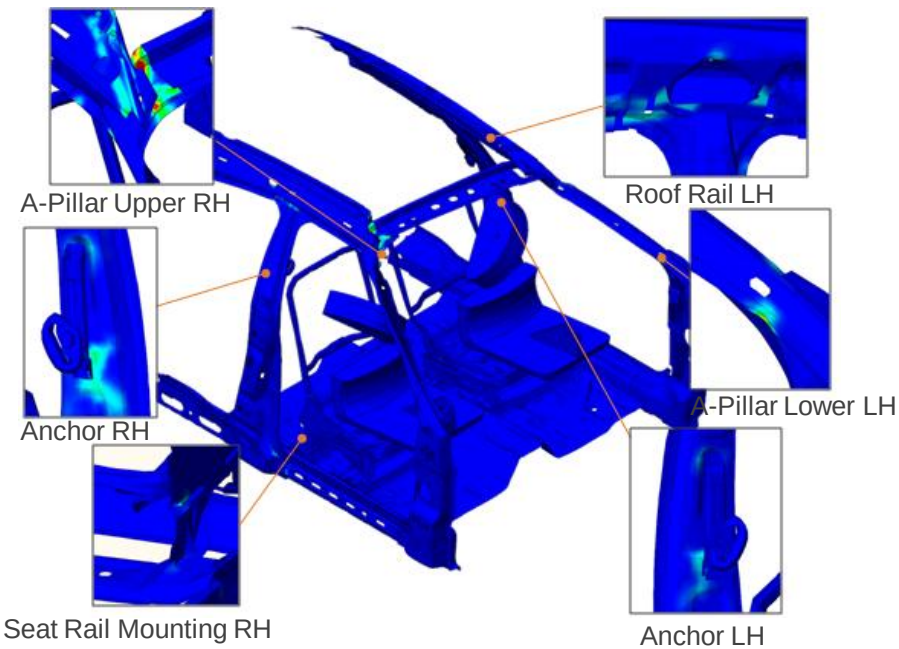
Analysis model and loading condition

Post Processing

Excessive deformation is observed at the joint between Roof Header and Roof Rail.



Displacement distribution



Effective plastic strain

Side impact protection analysis - FMVSS214S

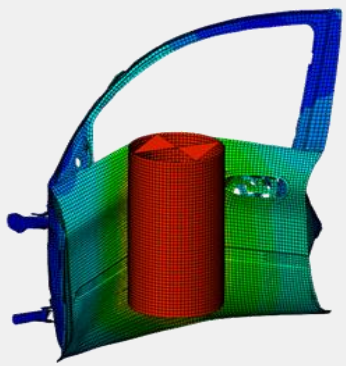
Overview

Objective

Protection of passenger's safety during automotive side impact, evaluate car door's stiffness according to FMVSS214S regulation.

Analysis Type

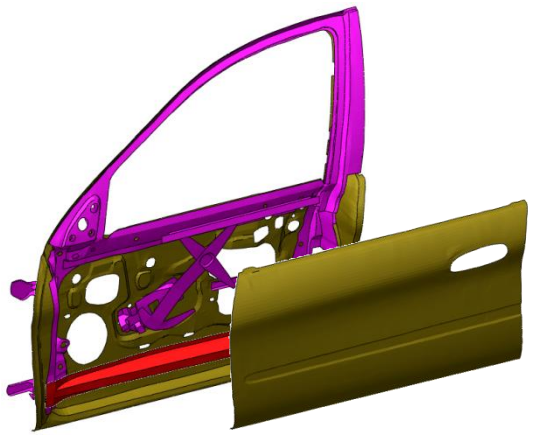
- Explicit nonlinear dynamic analysis



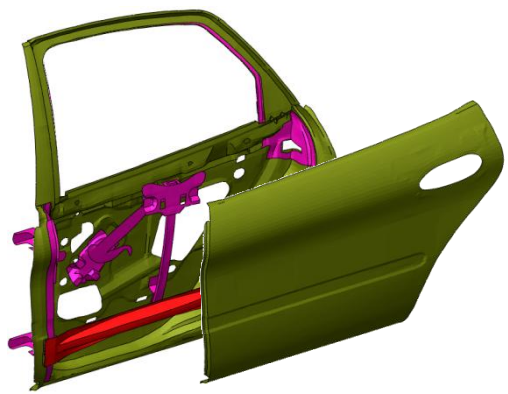
Pre Processing

Loading Condition:

- Load is applied vertically on the side doors according to the impacting cylinder.



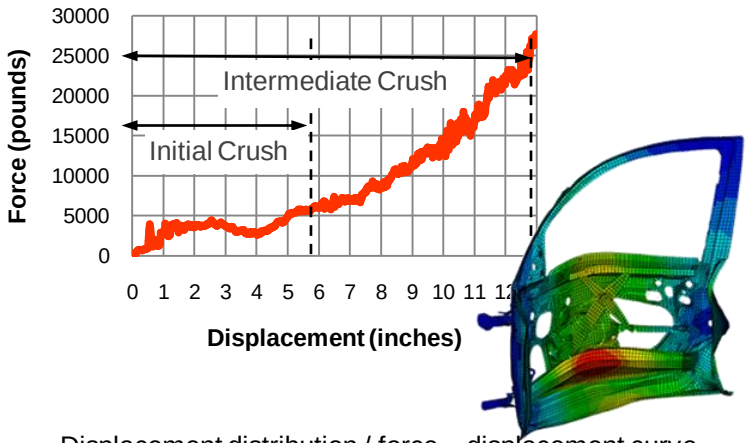
Analysis model – front door



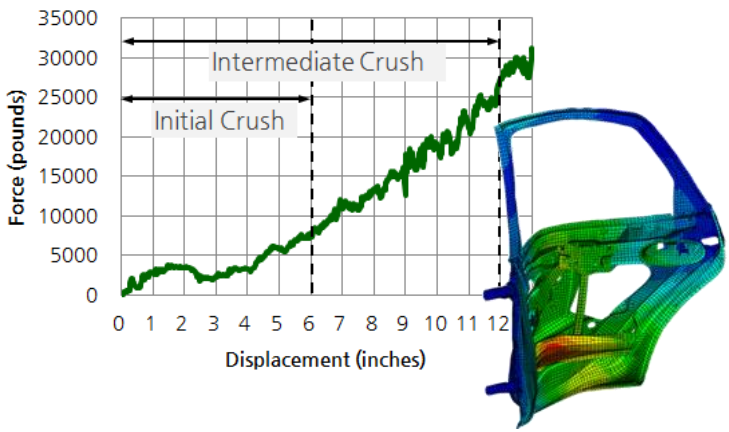
Analysis model – rear door

Post Processing

Static stiffness of the side door can be calculated from the load-displacement curve



Displacement distribution / force – displacement curve (front door)



Displacement distribution / force – displacement curve (rear door)

Door trim linear impact analysis

Overview

Objective

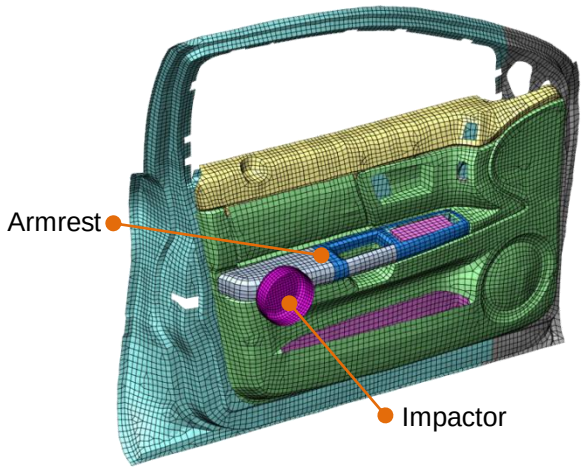
Evaluation of the impact load of door trim to passenger during automotive side impact.

Evaluation of the door trim design to determine passenger safety.

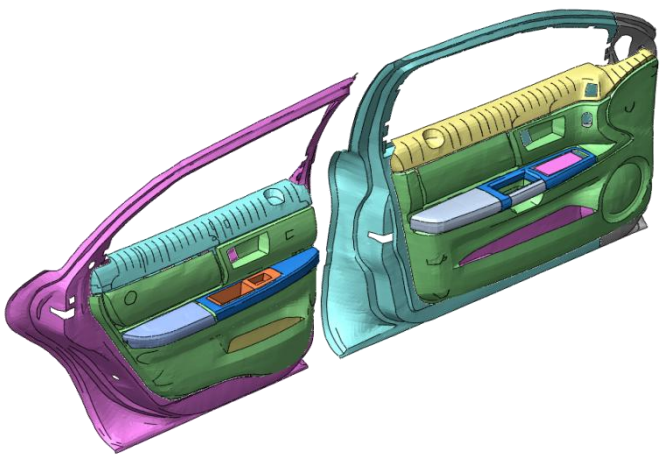
Analysis Type

- Explicit nonlinear dynamic analysis

Pre Processing



Door trim FE model

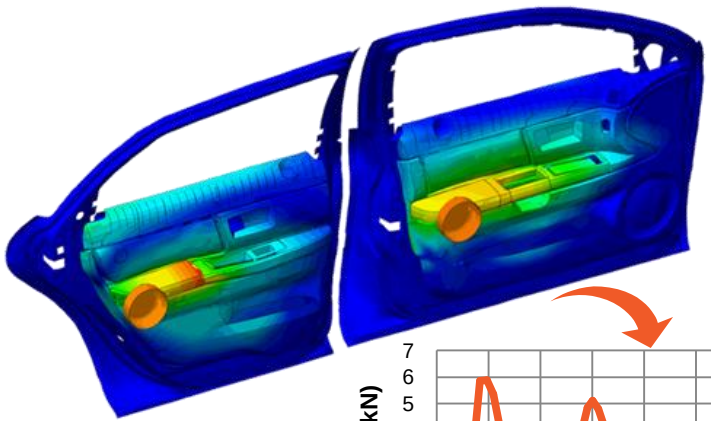


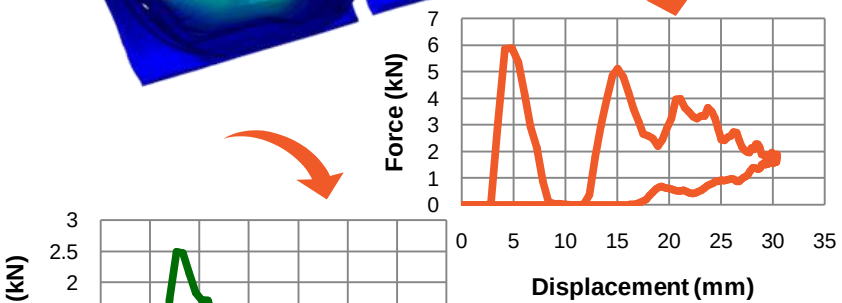
Front, Rear door trim structure

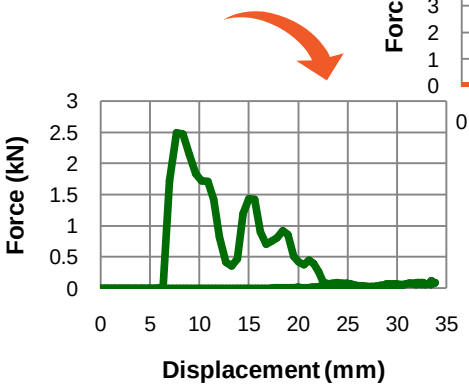
Post Processing

Door trim round Impactor is set with an initial velocity, and then after shock, the peak force can be calculated

Peak force of the rear door trim is lower, therefore low rear-seat passengers have less risks to be injured in case of a side collision.







	Peak force
Front door	5.9kN
Rear door	2.5kN

Identify the dynamic characteristics of automotive body using modal Analysis

Overview

Objective

Identification of natural frequencies and mode shapes of car body.

.

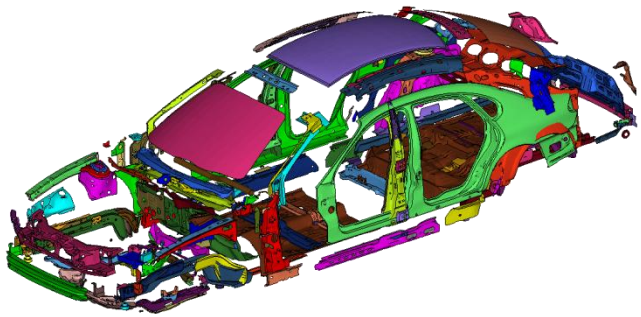
Analysis Type

- Modal analysis

Pre Processing

Analysis Condition:

- Frequency range of interest : 0Hz ~100Hz
- Total weight: 411kg

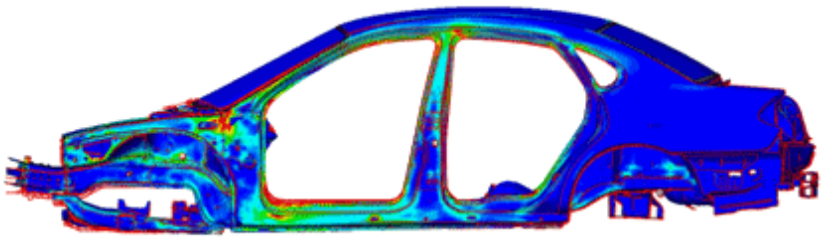


Automotive body structure

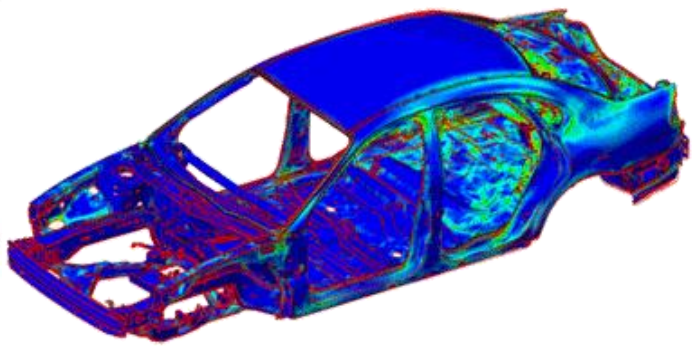
Mode	Natural frequency	Mode shape
1st	26.4 Hz	Front upper
2nd	30.9 Hz	Front lateral
3rd	32.9 Hz	Front upper
4th	36.7 Hz	**1st Front bending**
5th	37.8 Hz	2nd Rear bending
6th	39.4 Hz	**Rear torsion**

Post Processing

1st front bending mode and rear torsion mode are observed in the modal analysis. Dynamic stiffness can be increased to stabilize the automotive body.



1st front bending mode : 36.7Hz



Rear torsion mode : 39.4Hz

Door vertical stiffness analysis

Overview

Objective

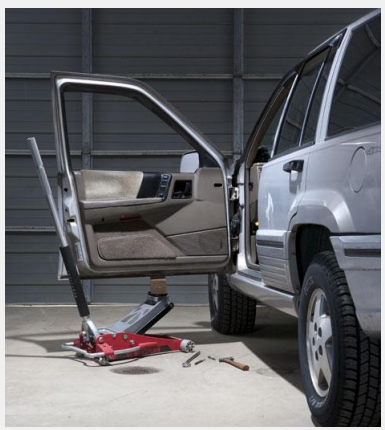
Investigation of stiffness of vehicle door under self weight and excessive vertical load

Review if damage happens at hinge and inner panel.

.

Analysis Type

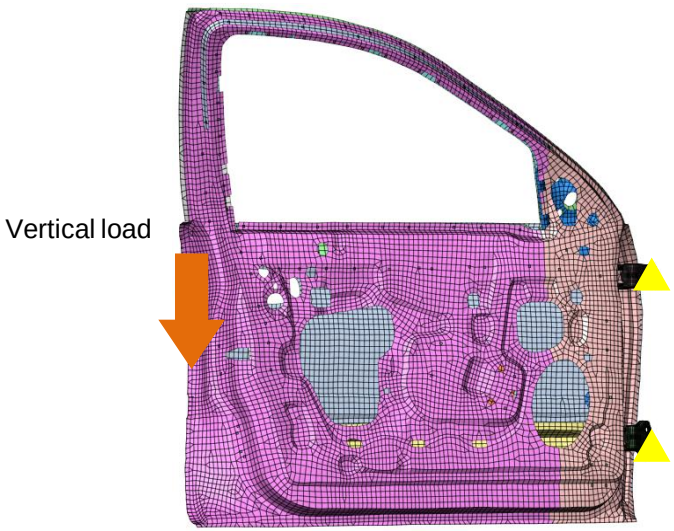
- Nonlinear static analysis



Pre Processing

Analysis Condition:

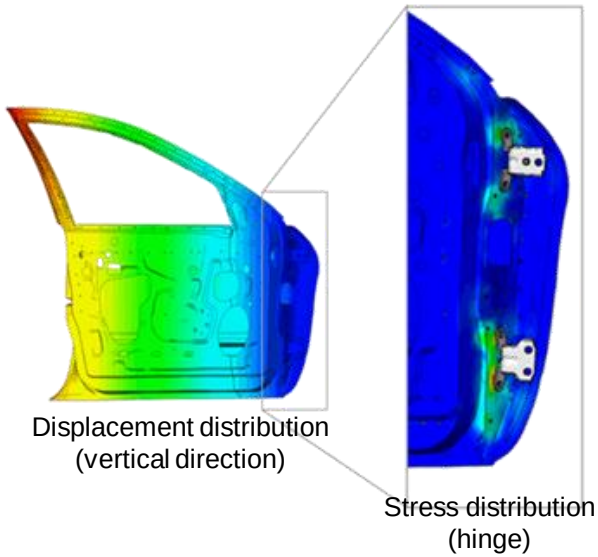
- Self weight
- Vertical direction load

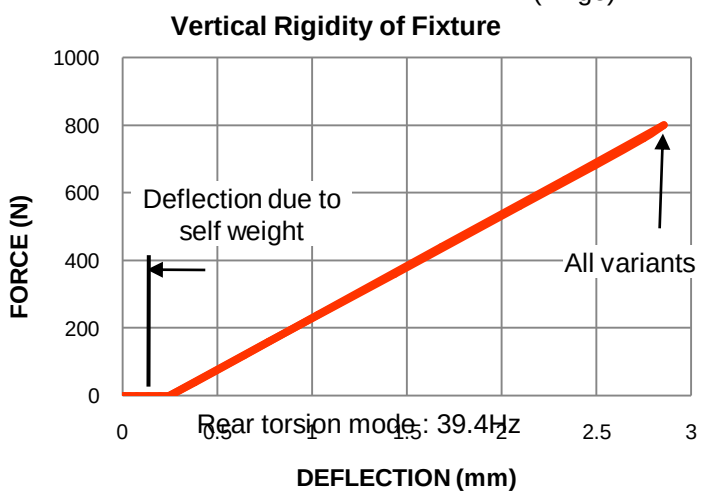


Post Processing

Identify the deflection of door due to self weight and excessive vertical load.

Review if damage occurs at hinge and the inner panel areas.





Rear torsion mode : 39.4Hz

Brake disc FRF analysis

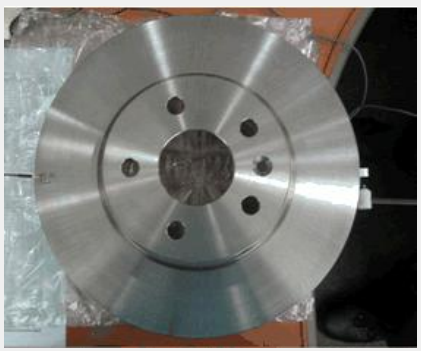
Overview

Objective

Review of the dynamic characteristics of brake disc to avoid squeal noise.

Analysis Type

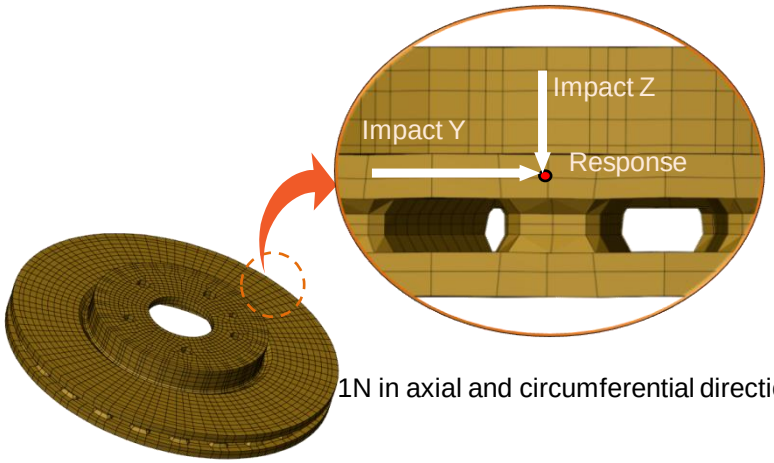
- Modal analysis
- Frequency response analysis



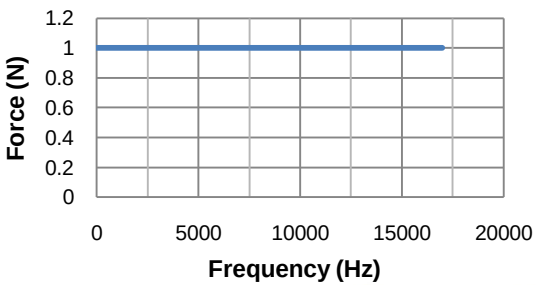
Pre Processing

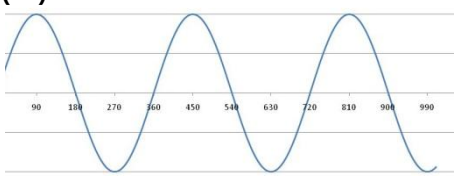
Analysis Condition:

- 1N unit load in axial and circumferential direction



1N in axial and circumferential directions



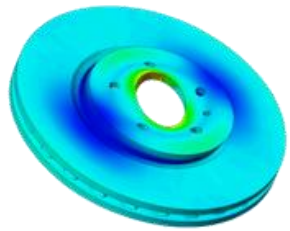


Unit loads are converted sine wave form load

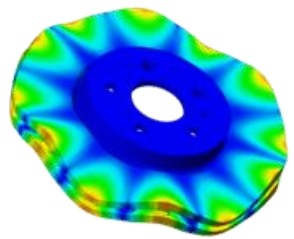
Post Processing

Frequencies of two modes are very close to 7000Hz, where squeal noise is most likely to occur.

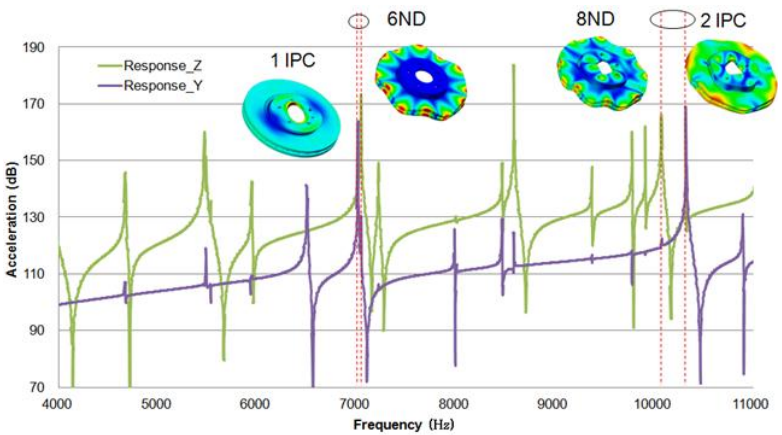
Mode	Natural frequency
2ND	1027 Hz
3ND	2394 Hz
4ND	3897 Hz
5ND	5468 Hz
1st IPC	7014 Hz
6ND	7045 Hz
7ND	8592 Hz
8ND	10068 Hz
2nd IPC	10285 Hz



1st IPC



6ND



Brake disc coning analysis

Overview

Objective

Prediction of the thermal deformation of brake disc during operation to avoid judder phenomenon.

.

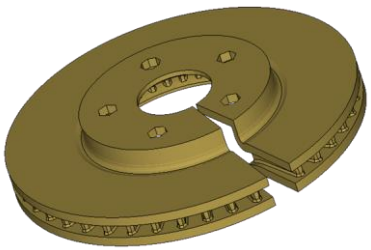
Analysis Type

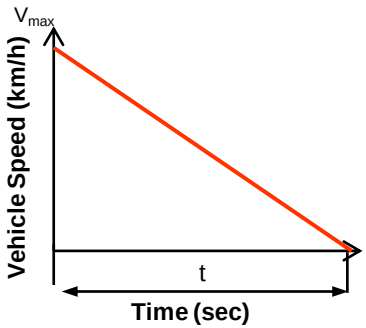
- Thermal stress analysis

Pre Processing

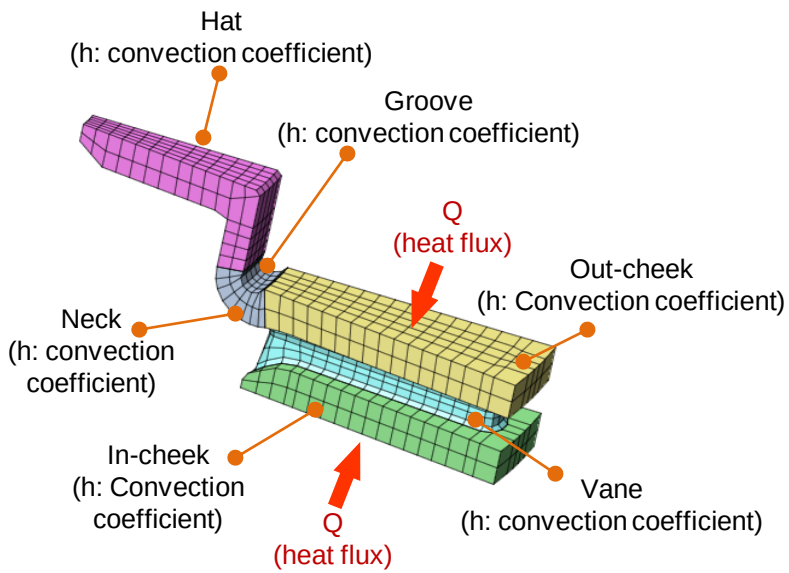
Apply heat load to brake disc which is generated during velocity decrease from maximum speed to 0.

Convert temperature result to static load for stress analysis.





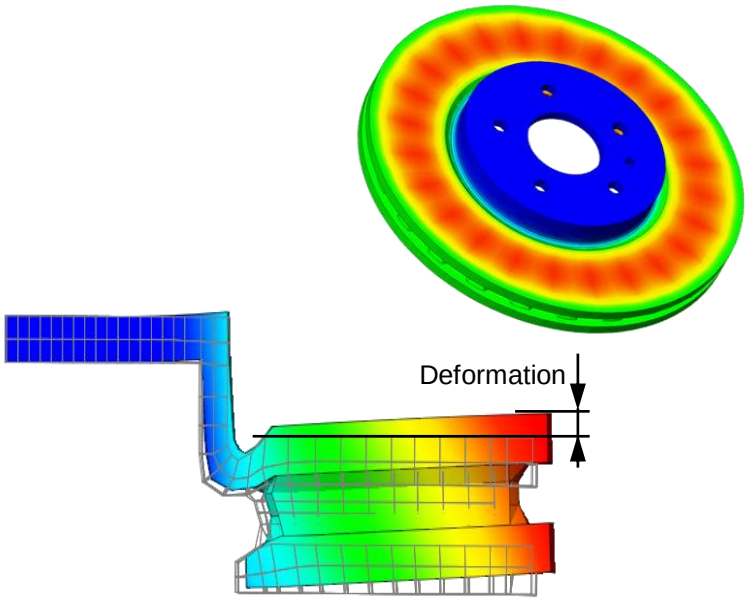
Brake disc



Analysis model

Post Processing

Maximum heat deformation is 0.702mm at 423℃



Thermal deformation

Temperature	Deformation
423℃	0.702mm

The final temperature and deformation results

Brake fluid temperature analysis

Overview

Objective

Prediction of the temperature transferred to brake fluid to avoid Vapor Lock phenomenon.

.

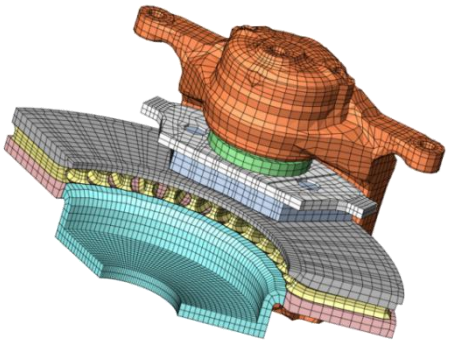
Analysis Type

- Heat transfer analysis (transient)

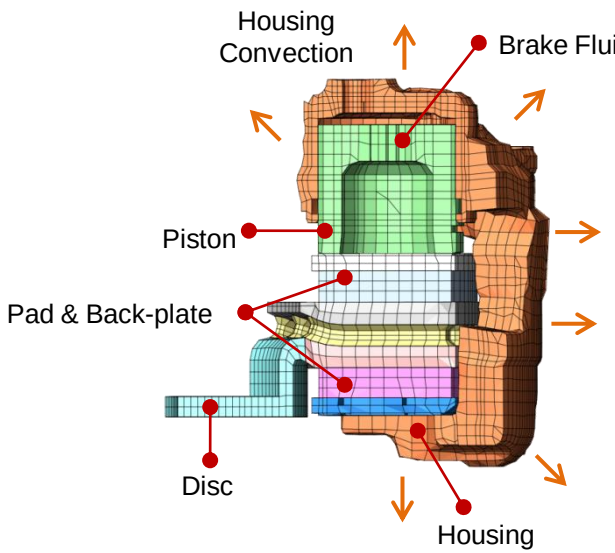
Pre Processing

When brake fluid is above boiling point, gas bubbles appear under elevated temperature and the hydraulic pressure can't be properly transferred to the pad.

To prevent this vapor lock phenomenon, heat transfer analysis is performed to predict the brake fluid temperature.

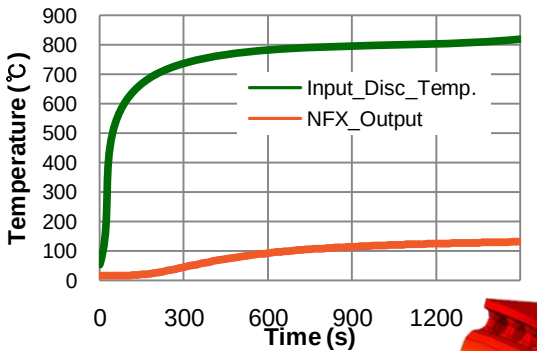


Front brake system

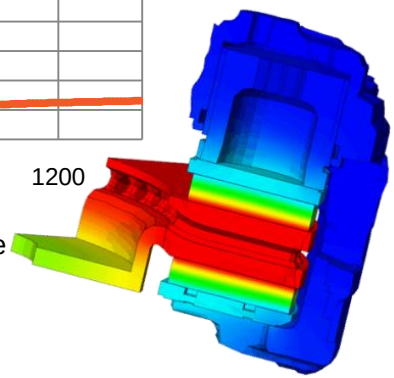


Post Processing

The graph below predicts fluid temperature during the temperature increase of brake disc. The maximum temperature is 131℃. It is below boiling point of any provided fluid material, therefore either of DOT3, 4, 5 type brake fluids can be used.



Brake fluid temperature according to time



Temperature distribution

No	Fluid	Boiling Point (Dry)	Boiling Point (Wet)	Applied Vehicles
1	DOT3	205 ℃	140 ℃	Middle/ small car
2	DOT4	230 ℃	155 ℃	Large luxury car
3	DOT5	260 ℃	180 ℃	Sports Car

Durability analysis of knuckle

Overview

Objective

Prediction of service life of automotive knuckle under driving mode.

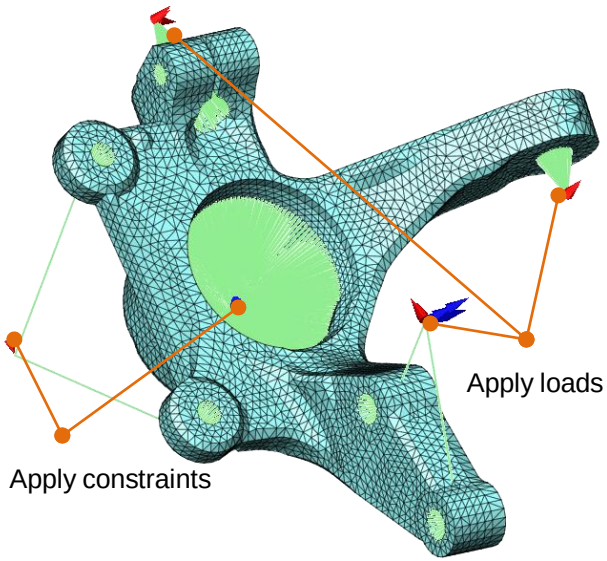
.

Analysis Type

- Linear static analysis
- Fatigue analysis

Pre Processing

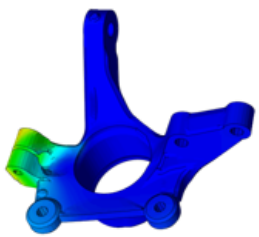
Linear static analysis is performed under different loading cases. And service life of the knuckle structure is predicted by combining the results from different loading cases together.



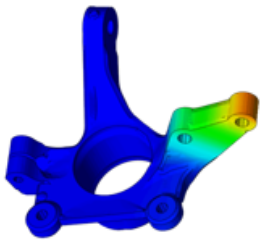
FE model of knuckle

Post Processing

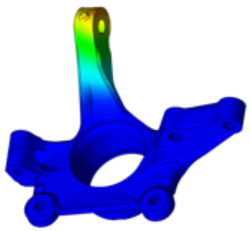
Linear static analysis



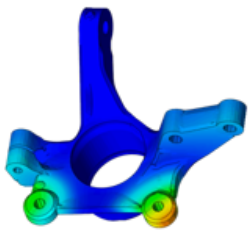
Case 1



Case 2



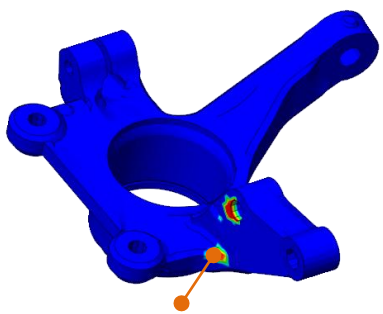
Case 3



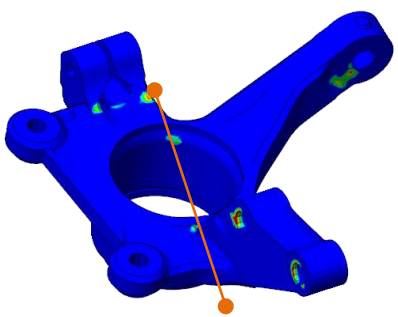
Case 4

Case	Static Stiffness
1	172.2 kN/mm
2	139.3 kN/mm
3	28.9 kN/mm
4	1129.4 kNm/rad

Fatigue analysis



Life cycle : 18291



Life cycle : 5983

Performance evaluation of turbo charger considering blade rotation

Overview

Objective

Jetting performance evaluation of automotive turbo charger, in which flow is generated by rotating blade.

.

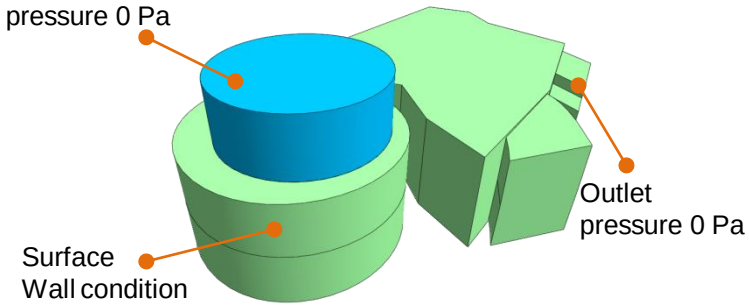
Analysis Type

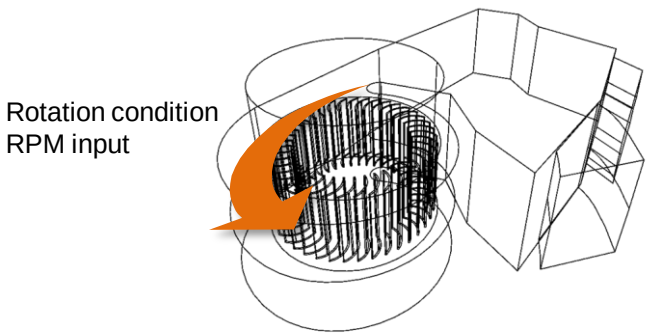
- Transient flow analysis
- General flow module
- Mesh deformation module

Pre Processing

Analysis Condition:

- Apply mesh deformation module according to blade rotation. Rotation axis . Application of angular rotation in RPM around the rotation axis.
- Flux calculation by applying 0 Pa at entrance and exit
- Wall distance type of wall applied to increase the convergence rate



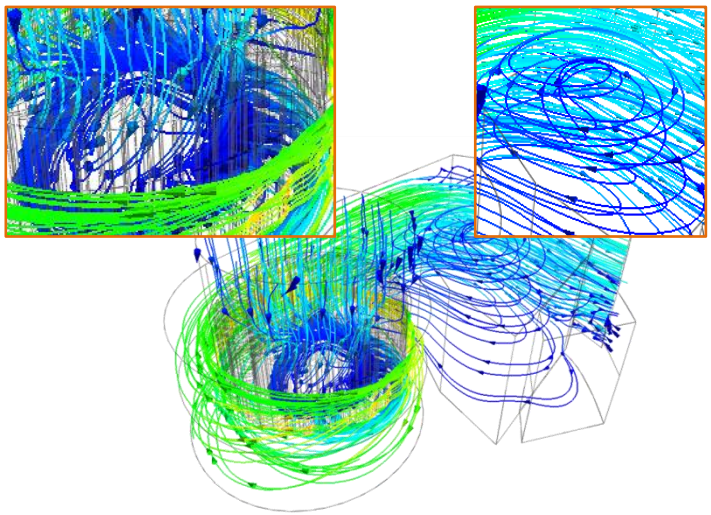


Post Processing

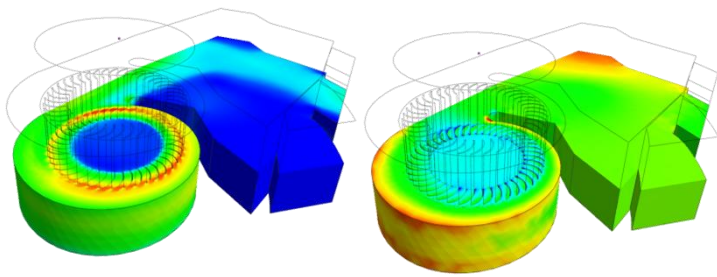
Confirm flux using flux automatic review function

Review fluid flow

Structural safety review through blade loading evaluation



Review flow lines and circulation area



velocity

hydraulic pressure

Construction

Design of guide vane for chemical plant pipe

Overview

Objective

Performance improvement of pipes that are used in the power plant system.

Select location for guide vane. Predict flow pattern before and after installation of the guide vane.

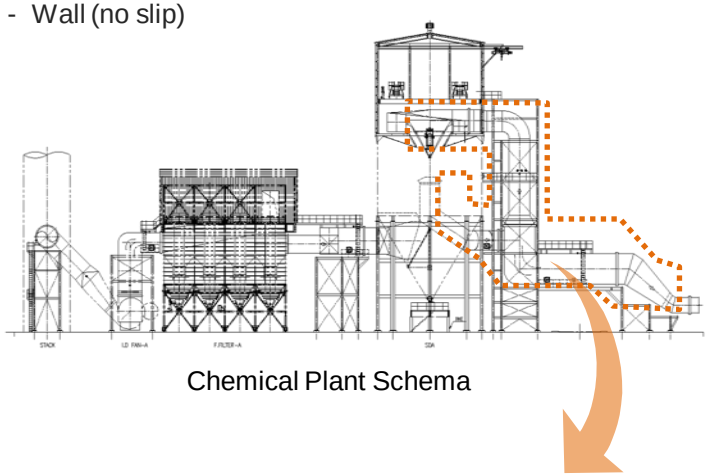
Analysis Type

- Transient flow analysis
- General fluid module

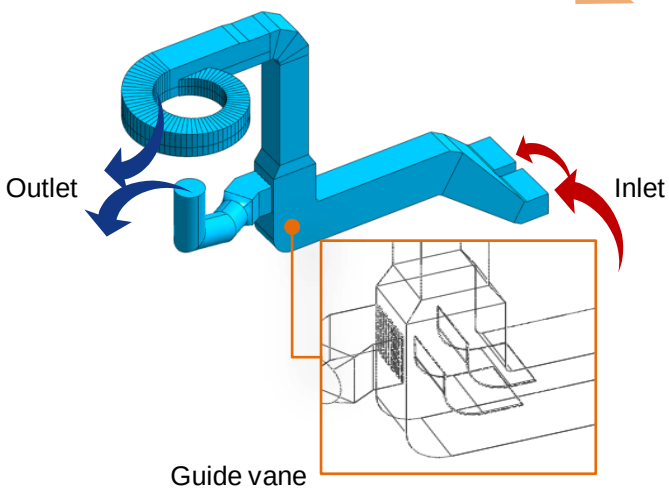
Pre Processing

Analysis Condition:

- k-ε turbulence model, apply initial stabilization step
- Inlet(speed)
- Outlet (pressure)
- Wall (no slip)



Chemical Plant Schema



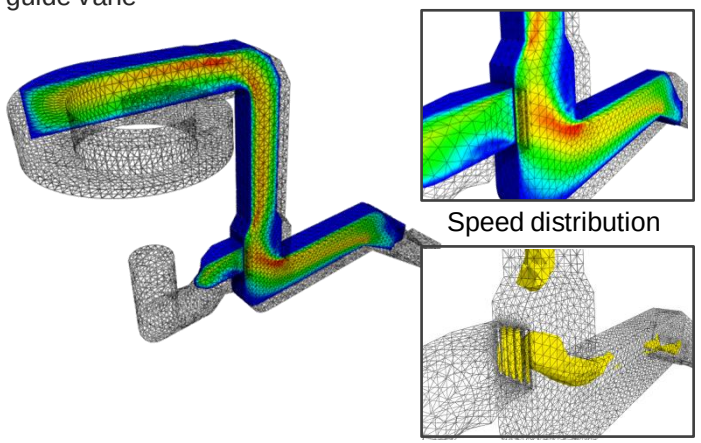
Geometrical model for inflow part of the denitrification equipment

Post Processing

Investigate detachment, high-speed area by plotting fluid speed distribution

Display detachment area directly on the geometries. High speed phenomenon can be observed.

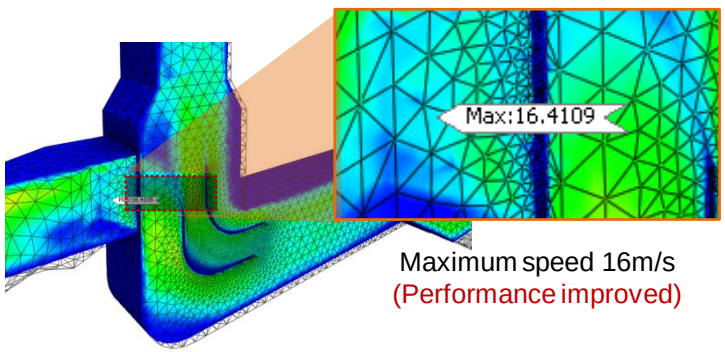
Compare speed distribution before and after the installation of guide vane



Speed distribution

High speed area (20m/s)

Before guide vane installation



Maximum speed 16m/s
(Performance improved)

After guide vane installation

Safety evaluation of steel making furnace

Overview

Objective

This analysis is carried out to evaluate safety of a newly installed steel making furnace at POSCO manufacturing plant .

Analysis Type

- Nonlinear static analysis
- Heat transfer analysis

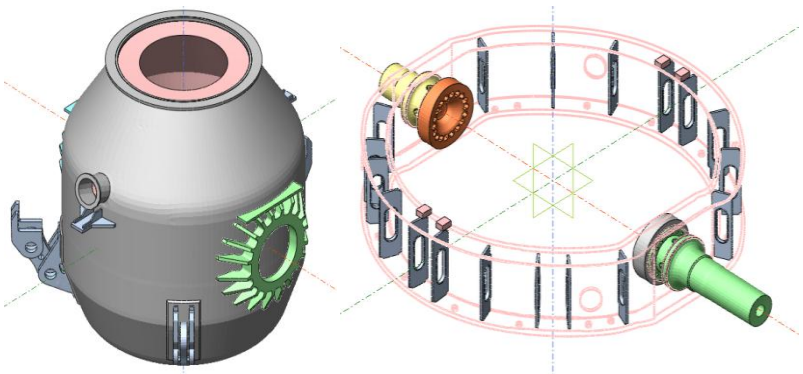
Pre Processing

Loading condition:

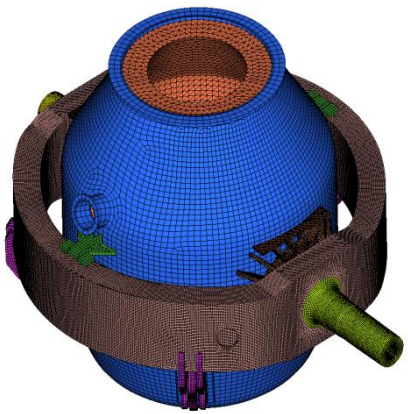
- self weight / heat /inner pressure

Allowance stress:

- ASME Section VIII Division 2



Revolving behavior of furnace: -180° ~ 180 ° around the shaft

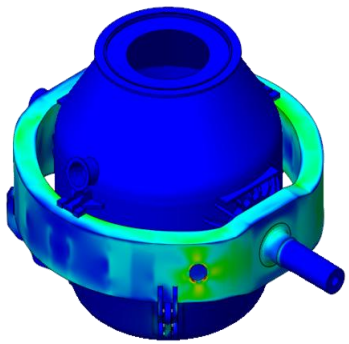


FE model of furnace

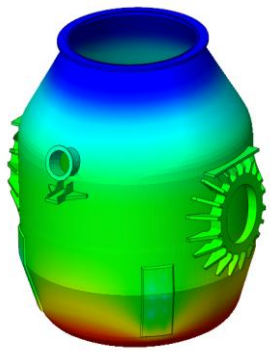
Post Processing

Deflection/ stress according to revolving angle

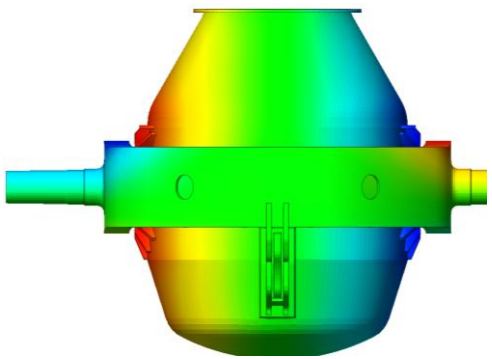
Thermal expansion due to heat transfer



Stress distribution



Temperature distribution



Deformation shape according to heat load

Safety evaluation of round silo dust collector

Overview

Objective

Evaluation of the safety of a round dust collector during conflagration.

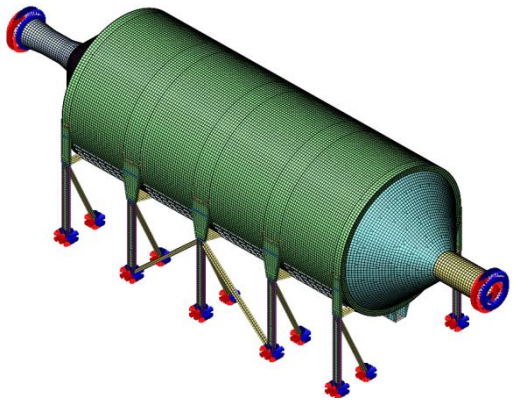
Analysis Type

- Heat transfer analysis
- Thermal stress analysis

Pre Processing

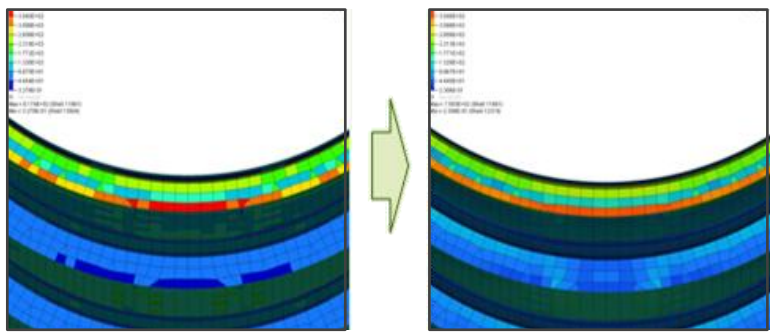
Apply boundary conditions at entrance, exit and supporting points.

In reality there is no constraint on the structure, however since unusual displacement occurs during the analysis, spring element is used to resist the displacement.



FE model of silo dust collector

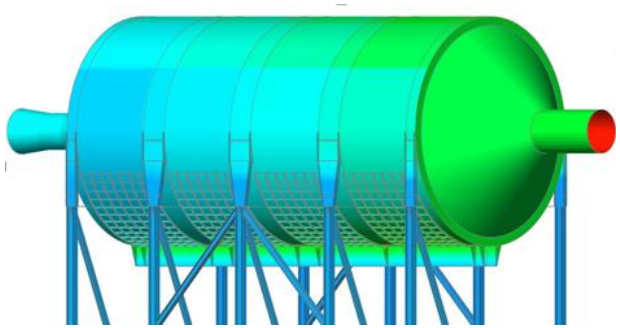
After thickness change, the stress allowance standard can be reached.



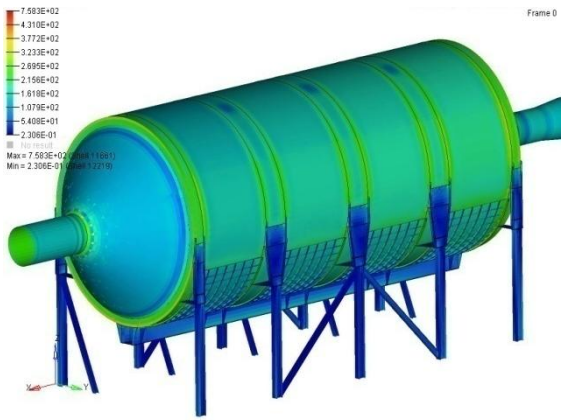
Stress distributions of original and improved designs

Post Processing

Observe temperature and thermal stress distribution on the dust collector.



Temperature distribution



Stress distribution outside the structure

Structure review of a piping according to pressure rise

Overview

Objective

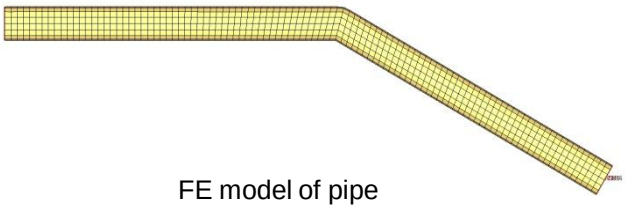
Analysis of piping structure's safety when pressure rises because of interior gas.

Analysis Type

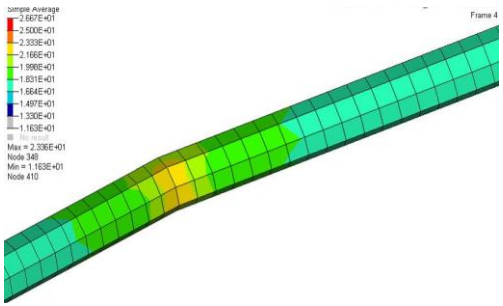
- Linear static analysis
- CFD analysis

Pre Processing

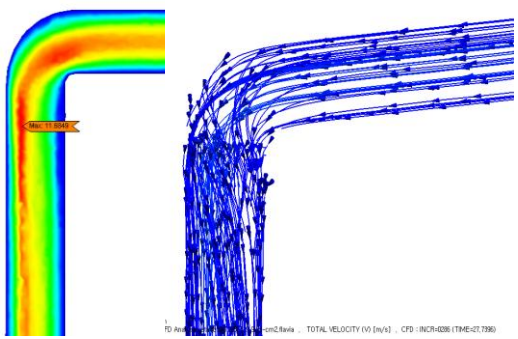
The worst case (the curve part of a pipe) of piping structures (nitrogen piping and oxygen piping) is chosen to be analyzed



FE model of pipe



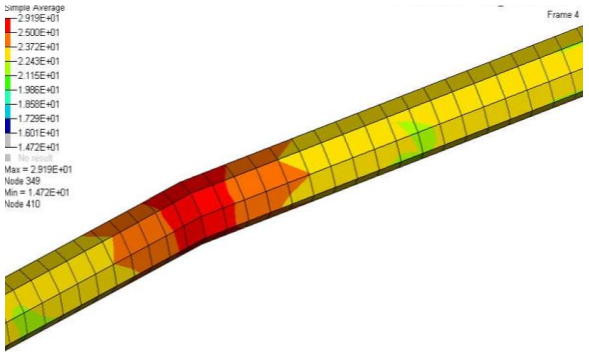
Stress distribution
(before pressure rise)



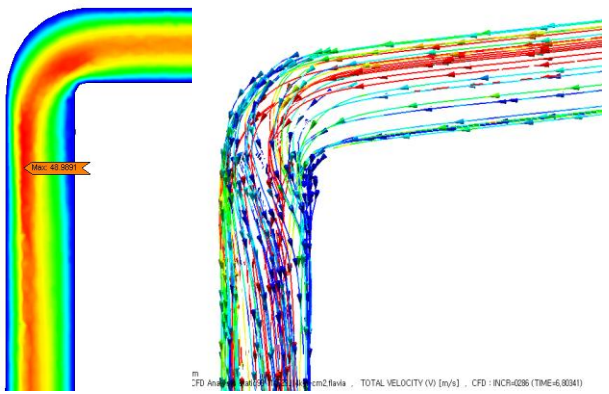
Velocity and flow path
(before pressure rise)

Post Processing

Identify structure's ignition probability by analyzing fluid's velocity and flow path before / after the pressure rise.



Stress distribution
(after pressure rise)



Velocity and flow path
(after pressure rise)

Safety evaluation of chimney's inner flue of thermoelectric power plant

Overview

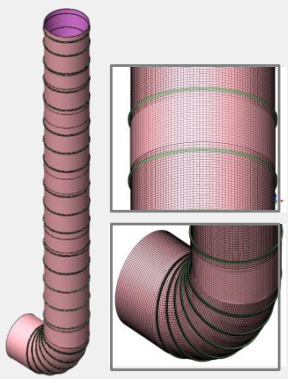
Objective

To evaluate chimney inner flue's structural safety during operation

Improve design of inner flue's components.

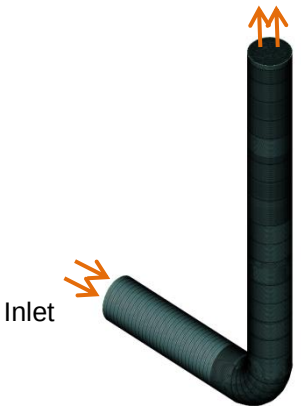
Analysis Type

- Heat & fluid analysis
- Heat transfer analysis
- Thermal stress analysis



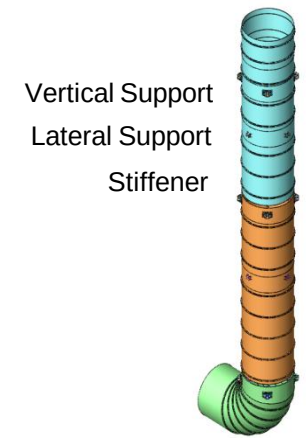
Pre Processing

Heat & fluid analysis

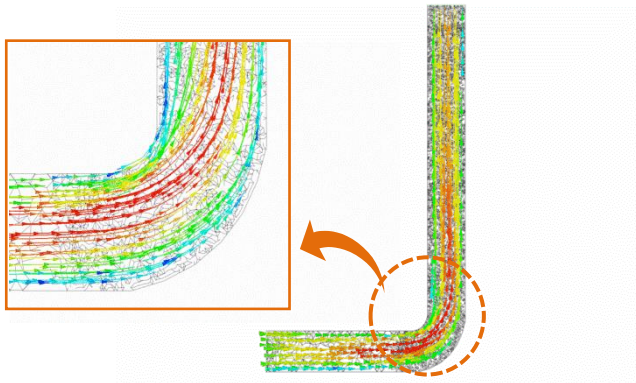


Heat transfer, thermal stress analysis

By using solid elements, inner restraint stress according t temperature difference inside, outside the structure can b considered.

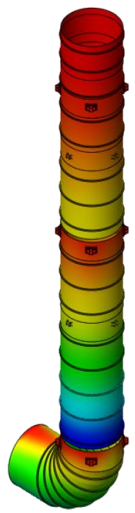


Post Processing



Flow lines of inner fluid

Verify stress at joints



Assess interval distance between expansion joints

Evaluation of reinforcement plan for fuel storage facility

Overview

Objective

Abrasion occurs to the inner wall as a result of repetitive friction. Structural stability is analyzed to set up a reinforcement plan for damaged parts.

Analysis Type

- Linear static analysis

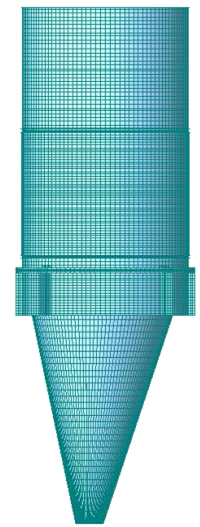
Pre Processing

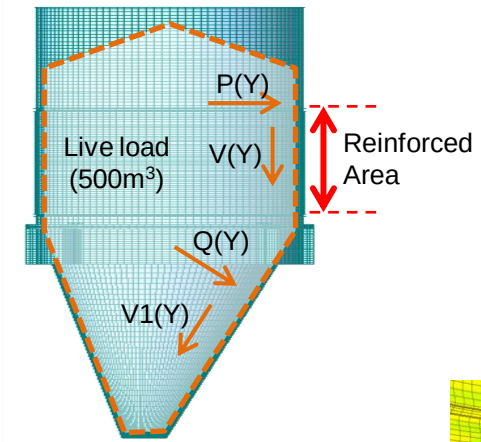
Loading condition: Self weight/live load/vibration

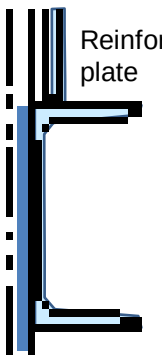
Load combination: KBC2009

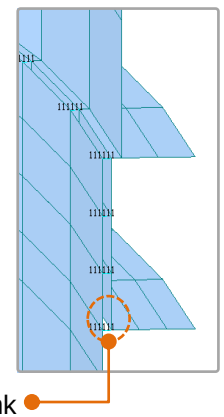
Allowance stress: ASME Section Ⅷ Division 2

Deflection check: KBC2009 (L/300)

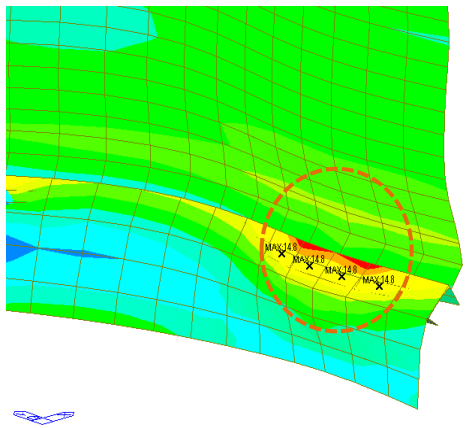




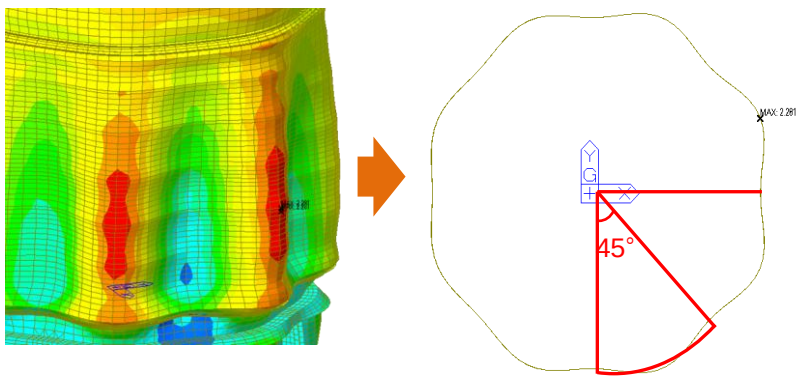




Post Processing



Maximum stress occurs at lower channel



Horizontal deflection according to live load

Evaluation of reinforcement plan for furnace cyclone

Overview

Objective

Analysis of the structure's stability to set up a reinforcement plan for damaged parts.

Analysis Type

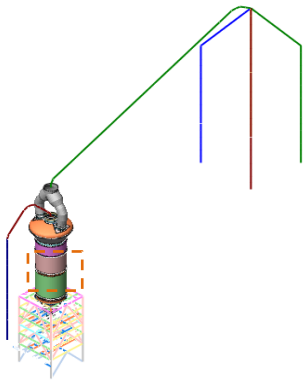
- Linear static analysis
- Response Spectrum Analysis

Pre Processing

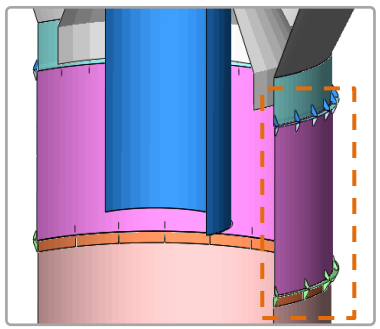
Analysis Condition:

- Dead / live load
- Vibration / wind (KBC2009)
- Inner pressure
- Node temperature

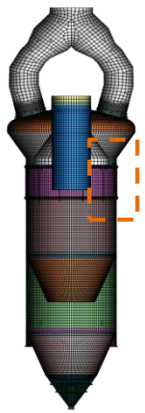
The duct and surrounding structures are modeled with 1D beam elements to investigate influence of wind and vibration load.

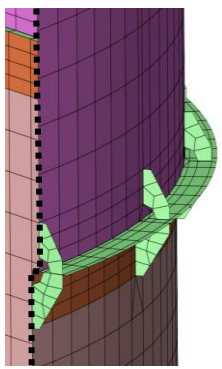


Entire model



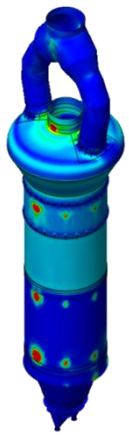
Reinforcement area

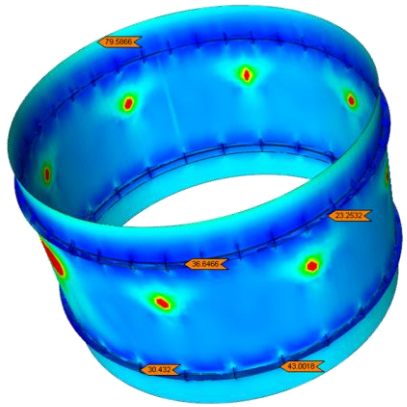




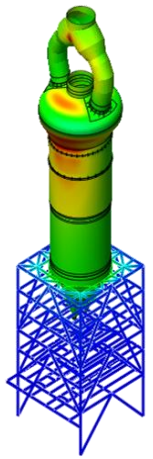
FE model of furnace cyclone

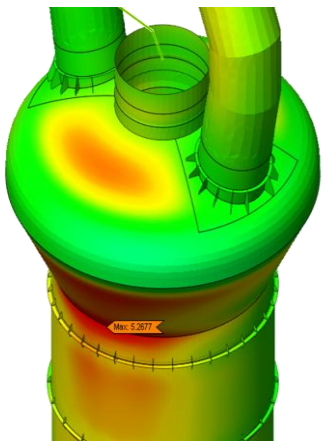
Post Processing





Stress distribution





Displacement distribution

Safety analysis of marine refrigeration machine under vibration

Overview

Objective

Verification if crack occurs on the structure and piping of a marine refrigeration machine under vibration load.

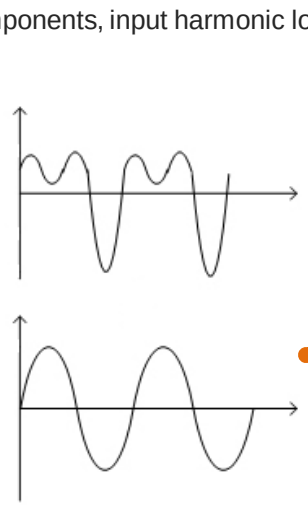
Analysis Type

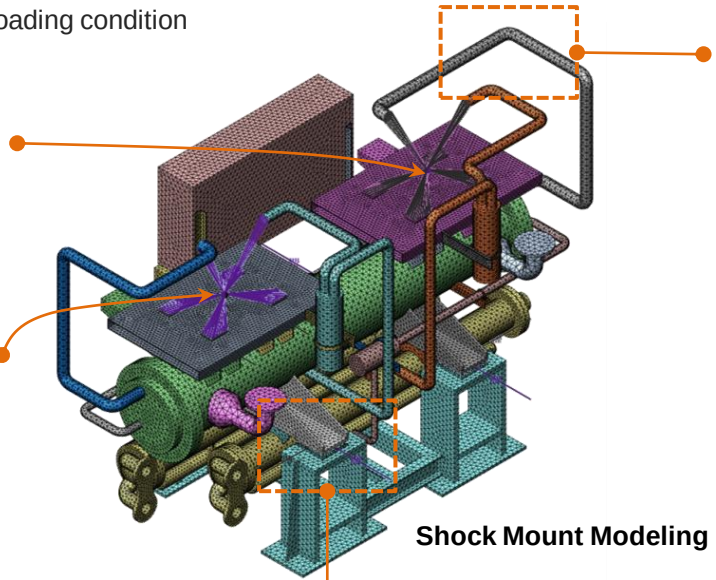
- Modal analysis
- Frequency response analysis

Pre Processing

Modeling : 3D Solid Element & Spring Element

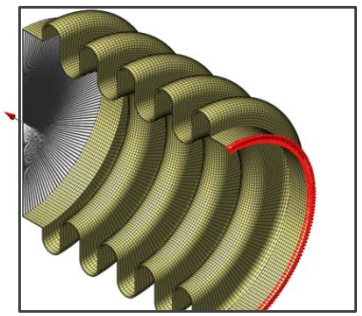
Loading condition: Analyze motor's vibration components, input harmonic loading condition

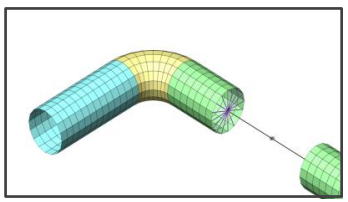


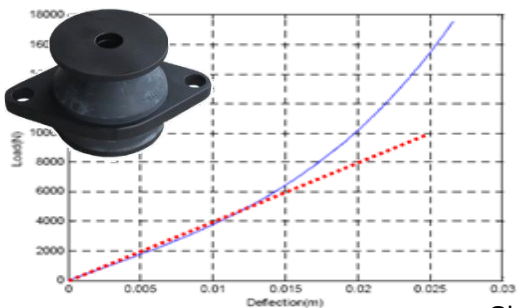


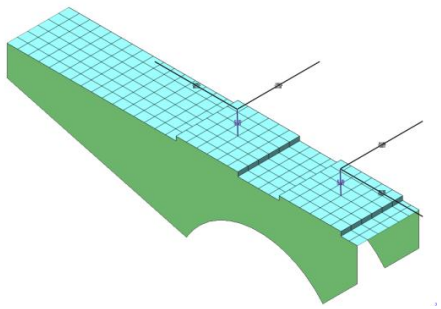
Shock Mount Modeling

Flexible Pipe Modeling
- Linear static analysis
- After equivalent stiffness calculation, Spring Element is used









Shock Mount's stiffness is evaluated in vertical / horizontal direction, Spring Element is used

Safety analysis of marine refrigeration machine under vibration

Overview

Objective

Verification if crack occurs on the structure and piping of a marine refrigeration machine under vibration load.

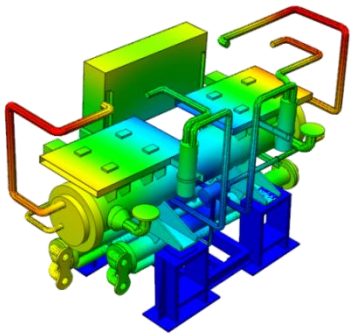
Analysis Type

- Modal analysis
- Frequency response analysis

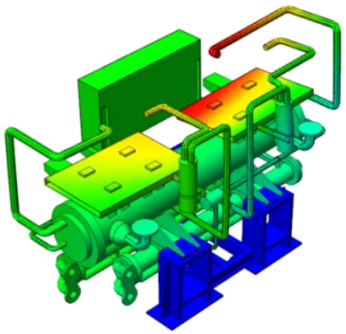
Post Processing

Modal analysis

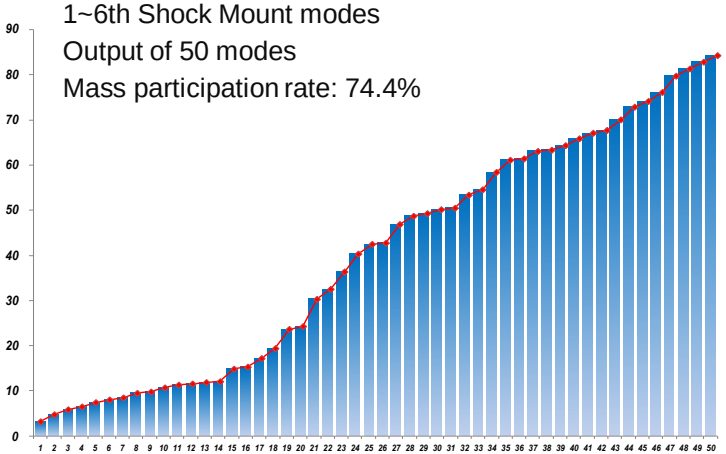
- Identify structure's vibration characteristics
- Identify structure's natural frequency and natural mode



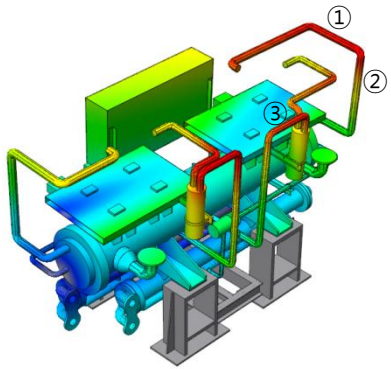
1st Mode Shape



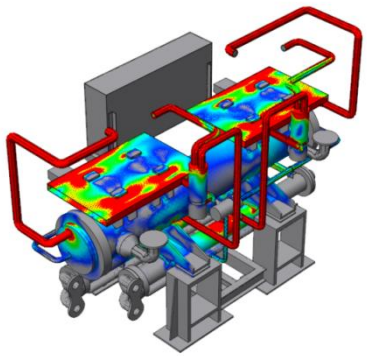
2nd Mode Shape



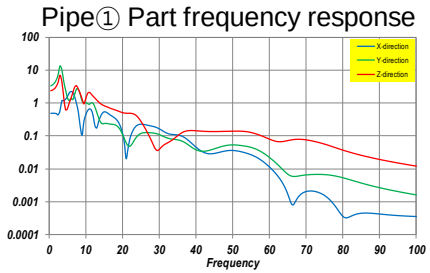
• Frequency response analysis
Main frequency response result output at piping part.

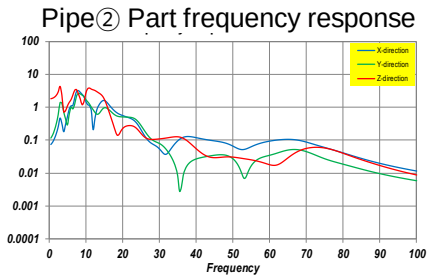


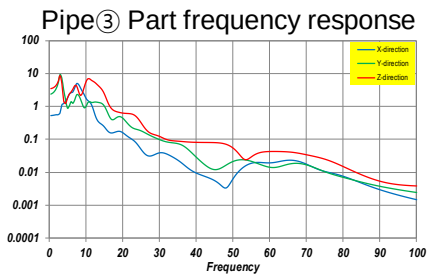
Resonant displacement distribution



Resonant stress distribution







Crane stability analysis under impulse load

Overview

Objective

Analysis of Goliath crane's stability under the impulsive load when a bolt at the main connection comes loose.

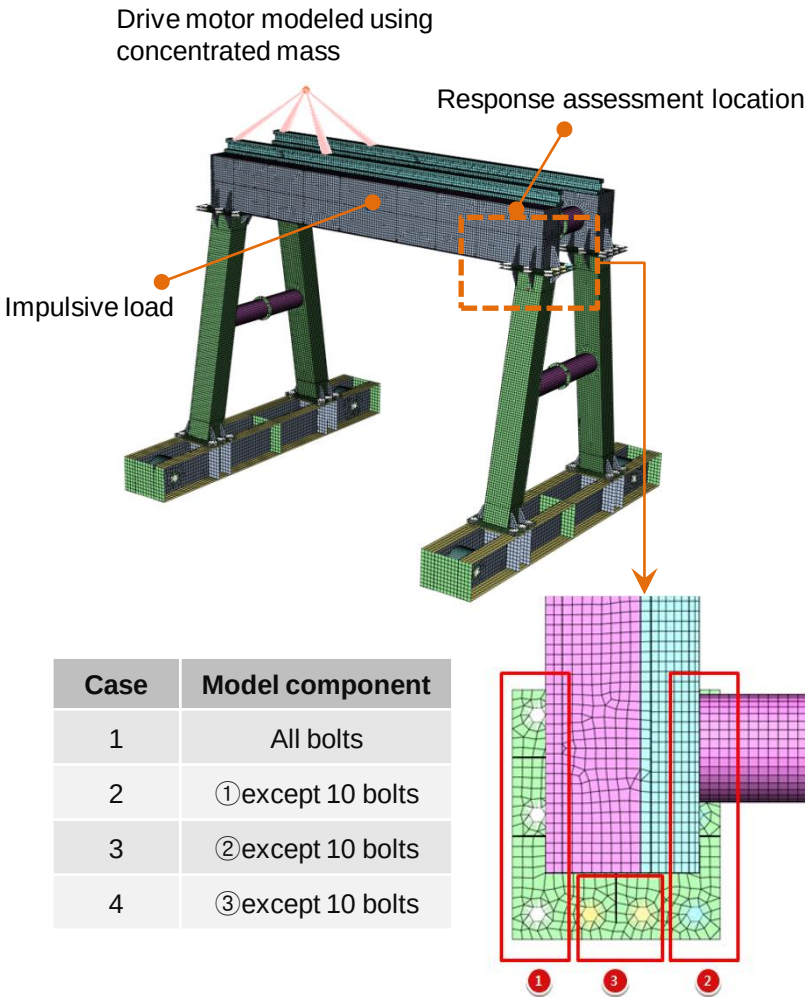
Analysis Type

- Transient response analysis

Pre Processing

Modeling: 3D Hexa, 2D Quad & Tri Element

Impulse Load: Identical impulse load as experiment

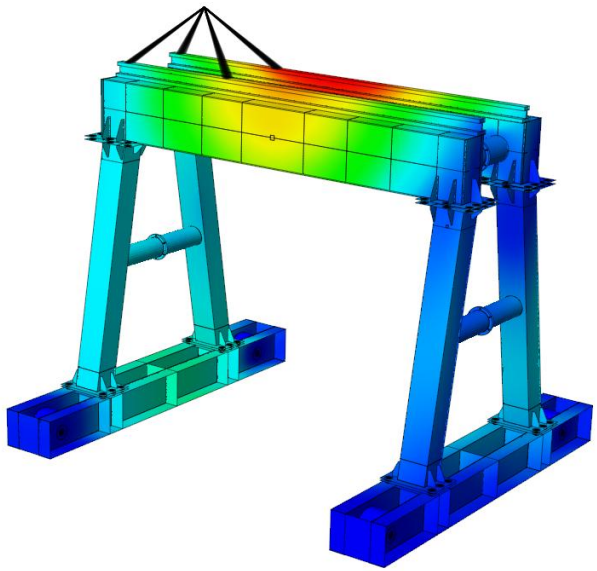


Case	Model component
1	All bolts
2	①except 10 bolts
3	②except 10 bolts
4	③except 10 bolts

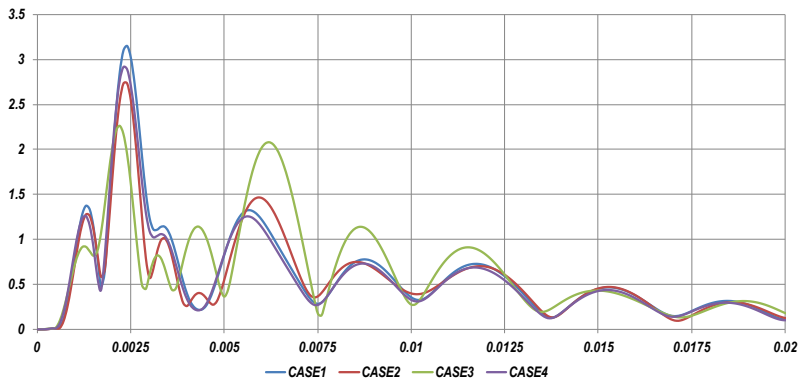
Model details around bolts

Post Processing

Review accelerations at different response measurement locations



Deformation shape



Impulsive response by analysis case

Structural stability analysis of stacker

Overview

Objective

After the light weight design of a stacker, analyze its structural stability during the movement of conveyor, gearbox and bucket wheel.

Analyze buckling characteristics of the stacker structure

Analysis Type

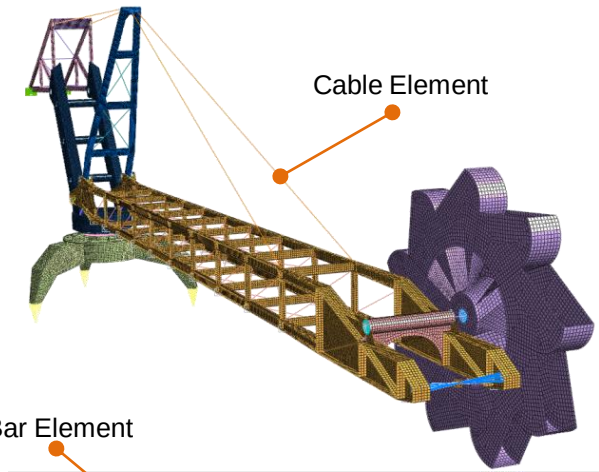
- Linear static analysis
- Buckling analysis

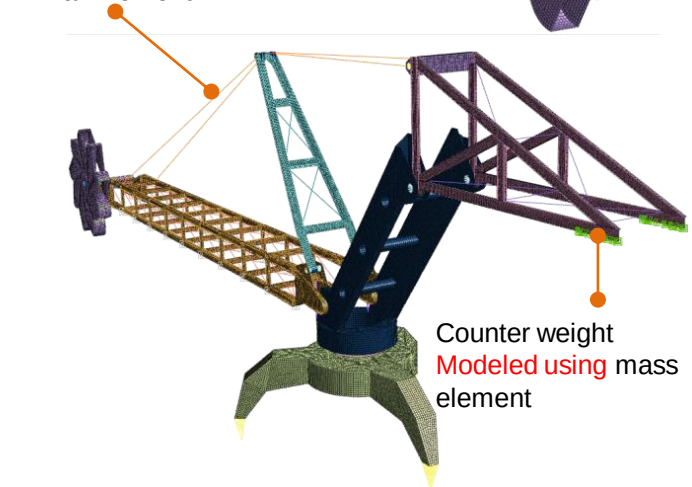
Pre Processing

Modeling: 1D Bar Element & 2D Quad Element

Analysis Condition:

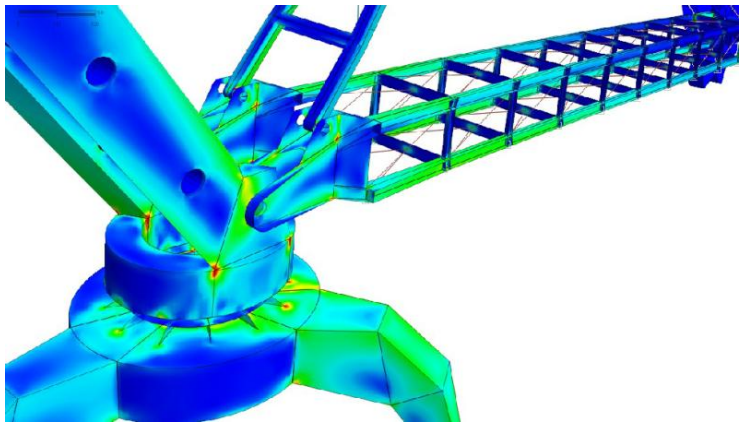
- Stacker load
- Loads generated during the movement of conveyor, gearbox and bucket wheel.



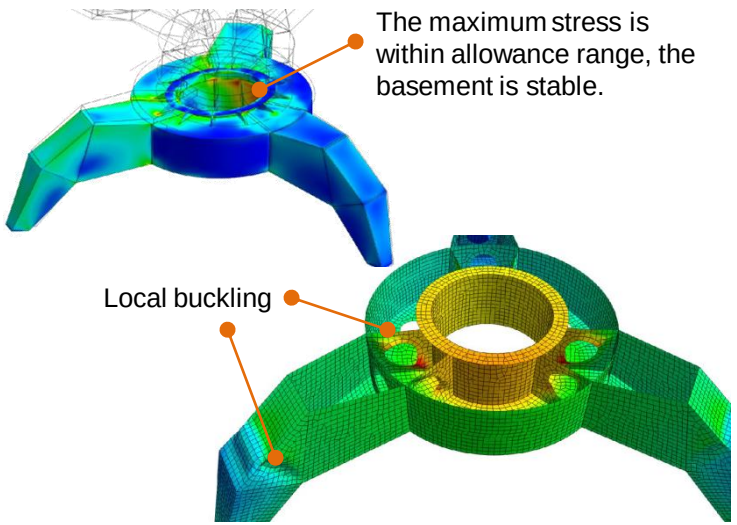


Post Processing

Structure stability evaluation



Stability evolution of basement



Turbulence influence analysis of nuclear power plant sump pump

Overview

Objective

Use flow analysis to control influence of vortex

Investigate of vortex influence using vorticity result.

Analysis Type

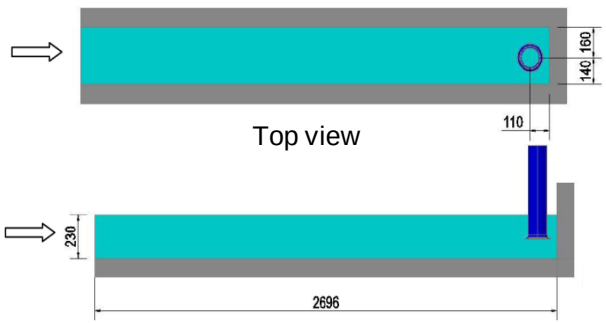
- Transient flow analysis
- General flow module
- Free water surface analysis

Pre Processing

Use free water surface level to realize fluid - air 2 direction flow

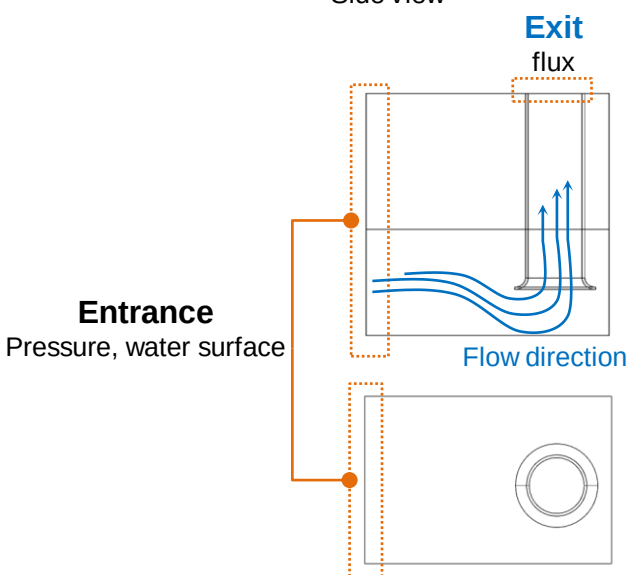
Apply flux at exit to simulate effect of sump pump

Model details (Matsui, 2007 journal extract)



Top view

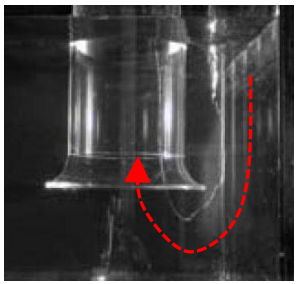
Side view



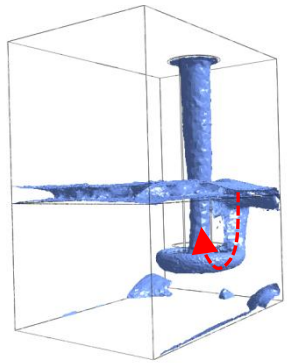
Post Processing

Fluid movement assessment according to time history

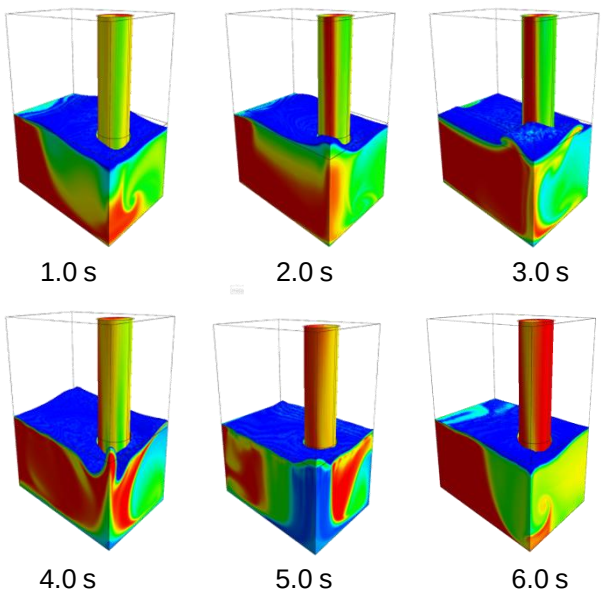
Analyze factors that influence pump performance by vorticity.



Experimental result
(Matsui, 2007 journal extract)



Vortex analysis results



Time history free water surface technique

Kuwait Olympic stadium wind dynamic analysis

Overview

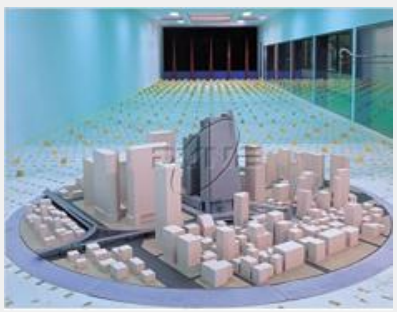
Objective

Due to the Olympic Stadium's irregular shape, standard wind load according to construction code cannot be applied.

Through CFD analysis wind load can be calculated according to the shape

Analysis Type

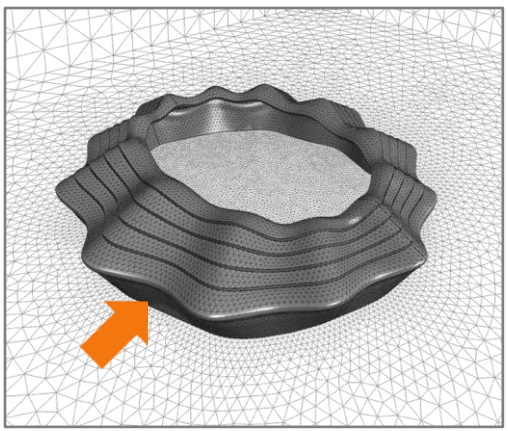
- Transient flow analysis
- General flow module



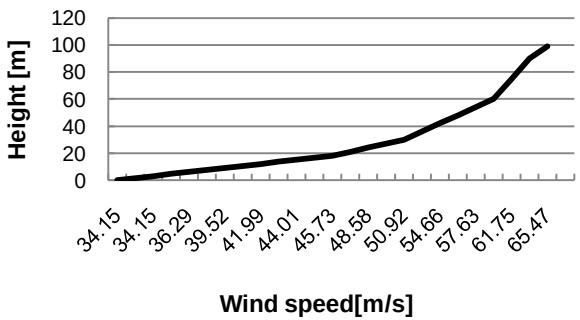
Pre Processing

Analysis Condition:

- k-ε turbulence model, apply initial stabilization step
- Inlet(speed)
- Outlet (pressure)
- Wall(no slip)
- Free end boundary [vertical speed 0]
- Wind load function is applied to the entrance



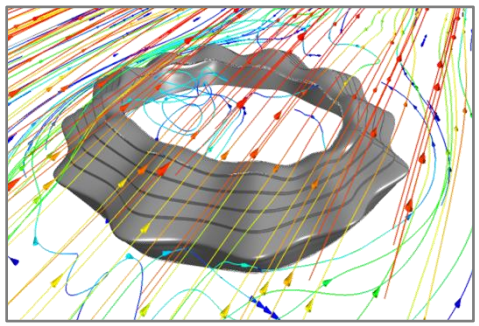
Flux function input: speed profile input



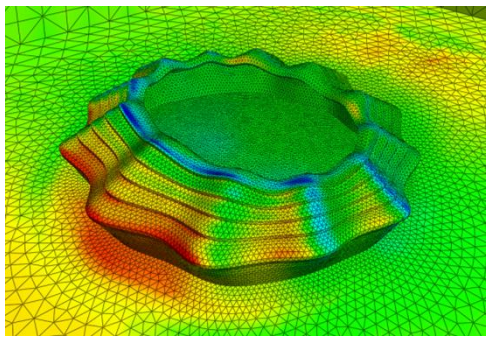
Post Processing

Possible to calculate pressure table on the stadium surface

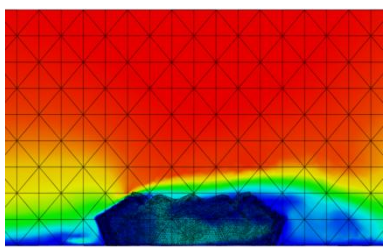
Possible to analyze wind flow under strong wind load



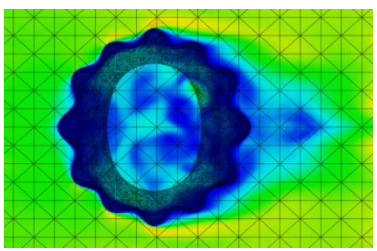
Flux visualization



Load distribution



Pressure distribution



Cross-section flux distribution

Seismic capacity review of steel connection and design of steel damper

Overview

Objective

Seismic capacity evaluation of a steel connection considering concrete slab.

Seismic capacity evaluation of steel damper

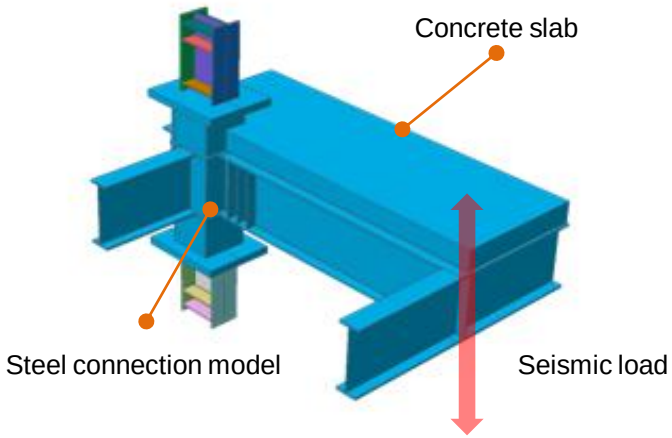
Analysis Type

- Nonlinear static analysis
- Material nonlinearity
- Geometrical nonlinearity

Pre Processing

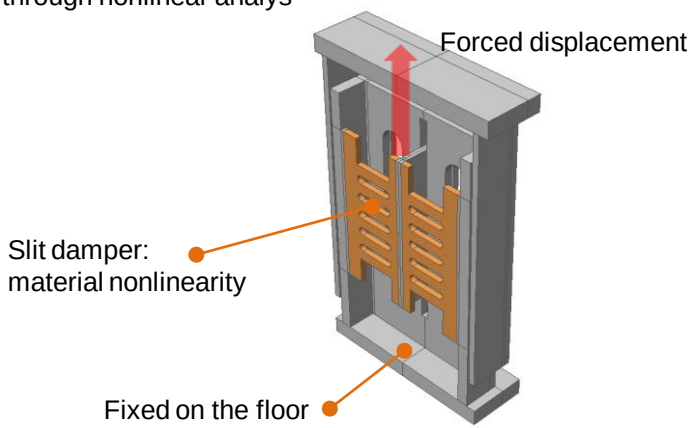
Seismic capacity evaluation of steel connection

- Consider concrete slab above
- Apply repetitive load same as real experiment



Seismic capacity evaluation of steel damper

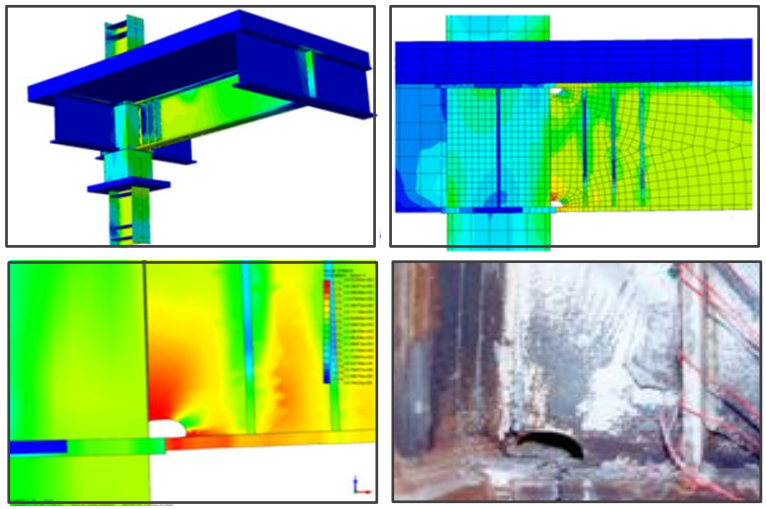
- Review energy absorption capacity of the slit damper through nonlinear analysis



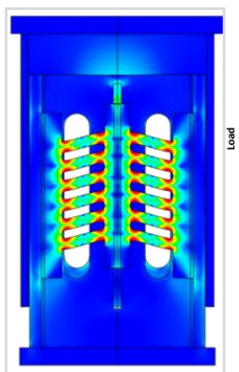
Post Processing

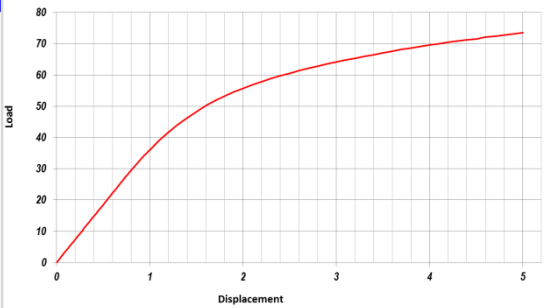
Influenced by the upper concrete slab, neutral axis of steel connection is displaced towards the concrete slab axis.

- Stress concentration at scallop part of the steel connection
- Influenced by upper concrete slab. The seismic capacity is expected to be reduced



Review energy absorption capacity of the slit damper





P-δ Curve

Impact analysis of an antenna tower

Overview

Objective

Analysis of the behavior of an antenna tower after being impacted with an airplane

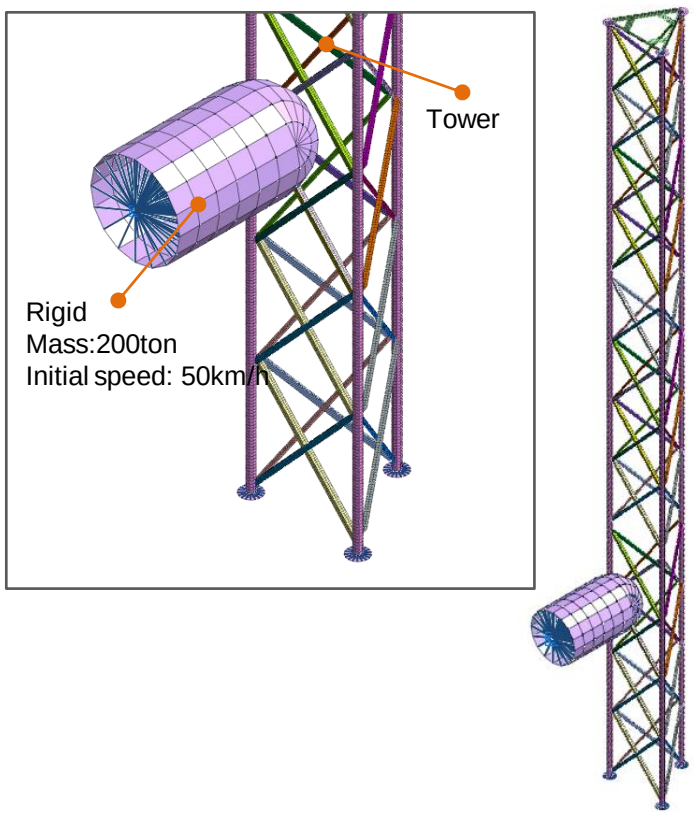
Analysis Type

- Nonlinear static analysis
- Explicit nonlinear dynamic analysis

Pre Processing

Modeling: 2D Shell Element

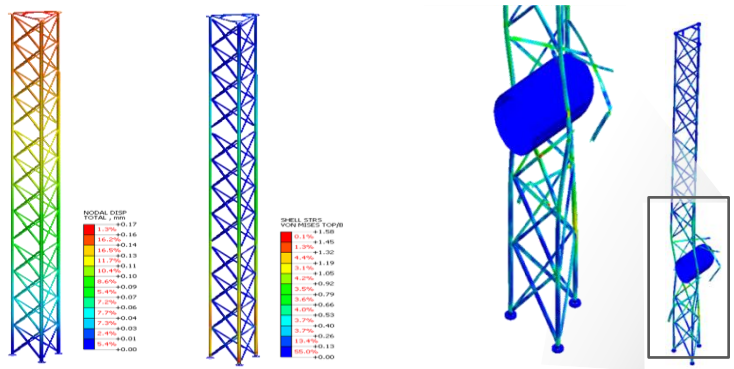
Contact Condition:
- Pin contact between all parts of the tower
- General contact between tower and the rigid flying object



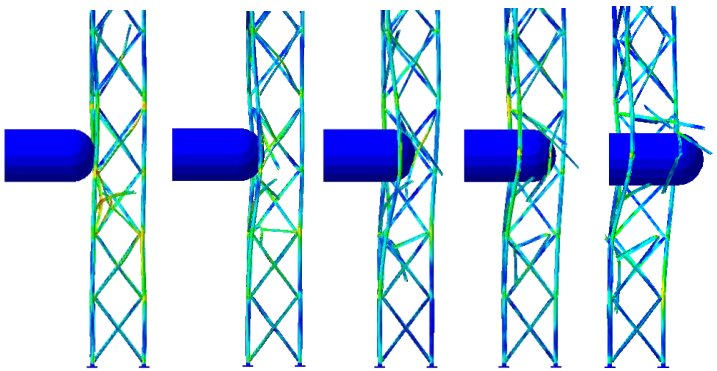
Post Processing

Tower's deformation and stress according to weight

Sequential nonlinear analysis during the impact considering static load



Fracture behavior of antenna tower during the impact



Consumer

Office chair simulation considering BIFMA standard

Overview

Objective

Prediction of results of various experiments on an office chair according to BIFMA(Business and Institutional Furniture Manufacturers Association) standard

Analysis Type

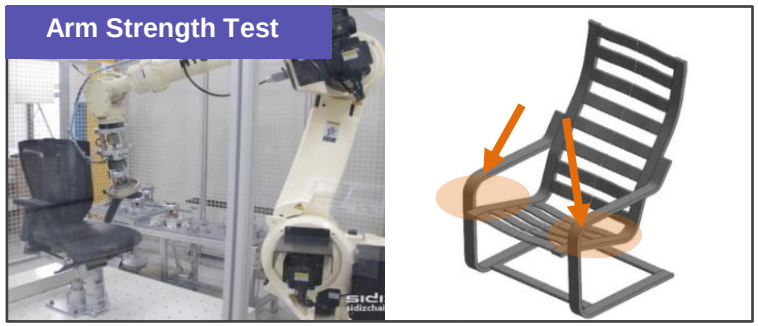
- Linear static analysis
- Topology Optimization

Pre Processing

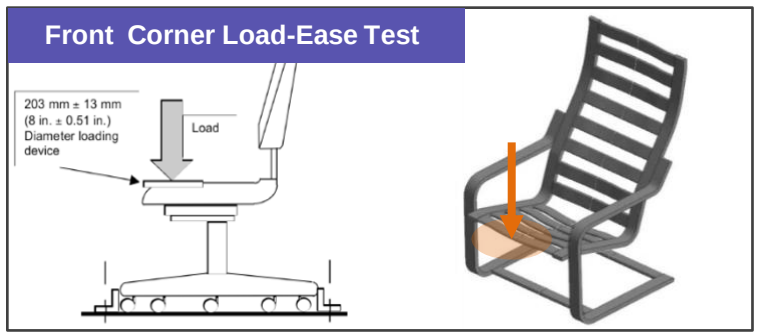
Apply conditions according to various experiments indicated in BIFMA standard.

By introducing simulation in chair design , we can also predict various situation in daily life.

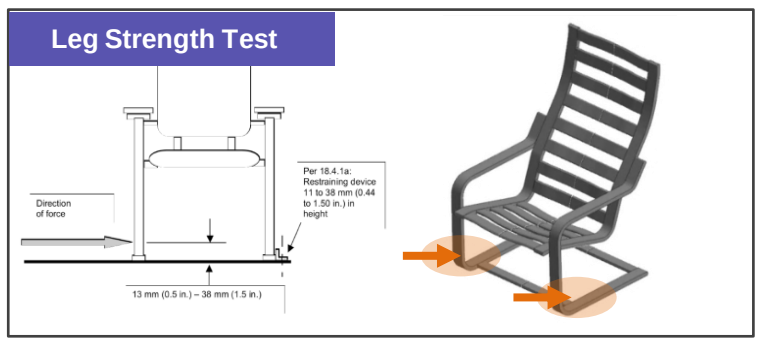
Arm Strength Test



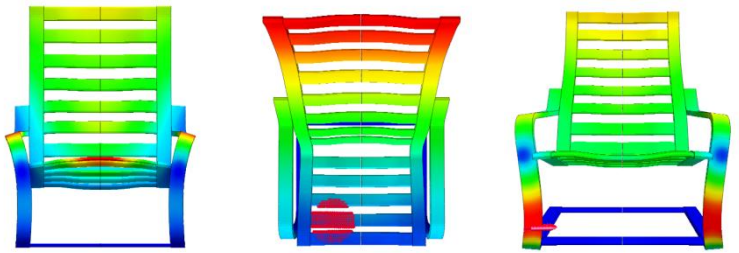
Front Corner Load-Ease Test



Leg Strength Test

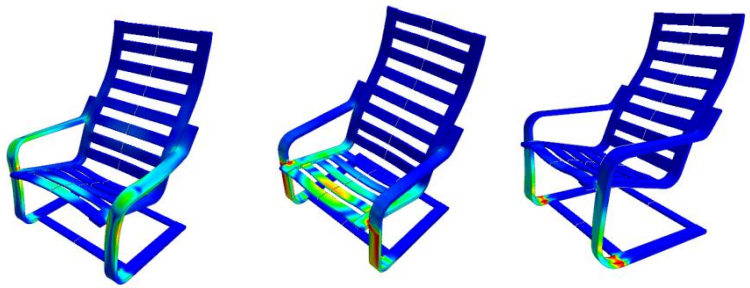


Post Processing



Arm Strength Test | Front Corner Load-Ease Test | Leg Strength Test

<Displacement distribution>

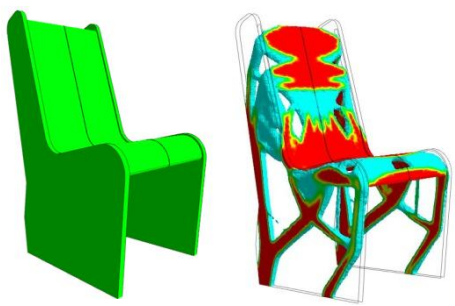


Arm Strength Test | Front Corner Load-Ease Test | Leg Strength Test

<Stress Distribution>

Topology Optimization

- Save material
- Obtain best design
- alternatives



Health

Arteriostenosis evaluation considering blood flow

Overview

Objective

Evaluation of arteriostenosis

Calculation of FFR values between entrance and target location.

Prediction of arteriostenosis through the changes of the FFR values.

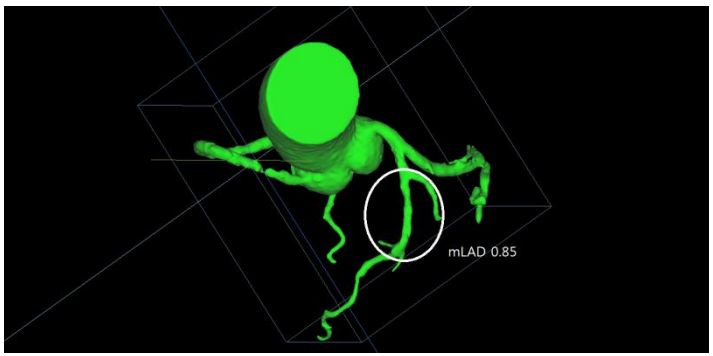
Analysis Type

- Transient flow analysis
- General flow module

Pre Processing

Analysis Condition:

- Apply initial stabilization step.
- General flow module
- Inlet condition(speed) : flux input
- Outlet condition(pressure) : pressure function of flux input
- Wall condition (no slip)

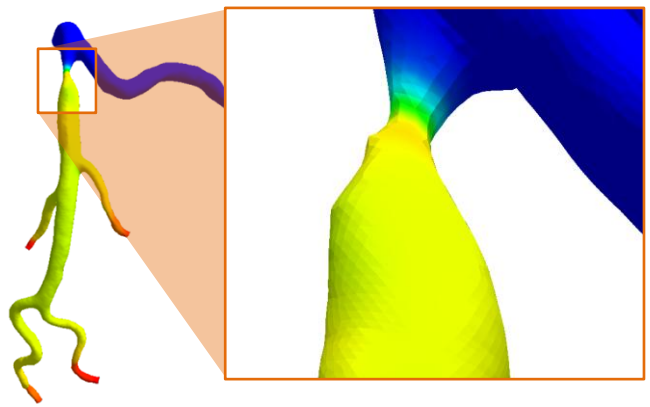


Model shape through angiography

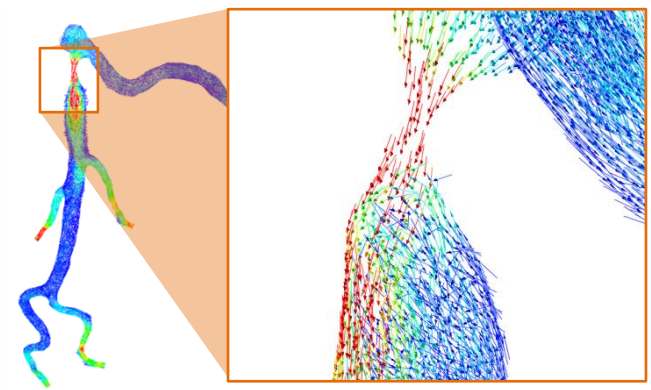
Post Processing

Blood pressure analysis (FFR : pressure between entrance and target location)

Blood flow visualization through velocity vectors



Pressure review around arteriostenosis area



Fluid flow review around arteriostenosis area

Structural Safety Analysis of Dental Implant

Overview

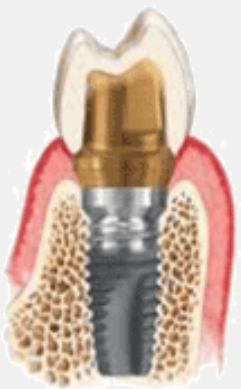
Objective

Investigation of state of jaw (maxillary bone) during movement

Analysis of contacting area between fixture and maxillary bone

Analysis Type

- Contact nonlinear analysis
- Fatigue analysis



Pre Processing

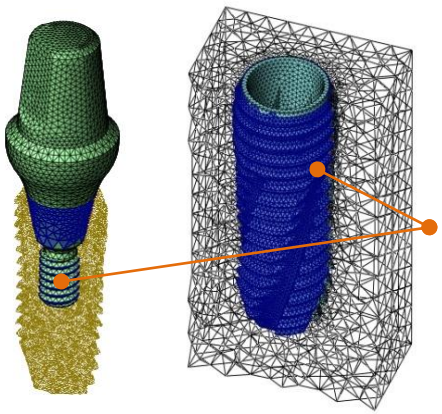
Modeling: 3D Solid Element

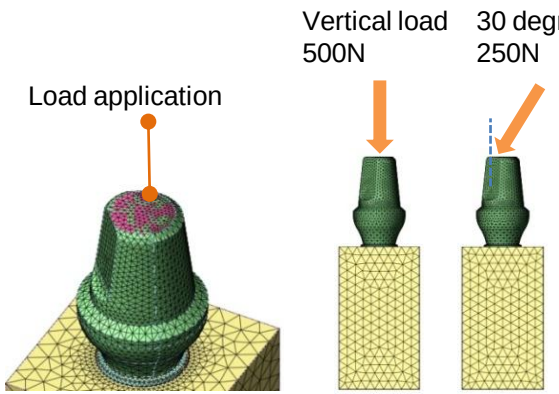
Loading condition:

- Vertical direction 500N; 30 degree direction 250N

Boundary Condition:

- Apply friction factors to contact face between abutment and fixture, and contact face between fixture and maxillary bone.

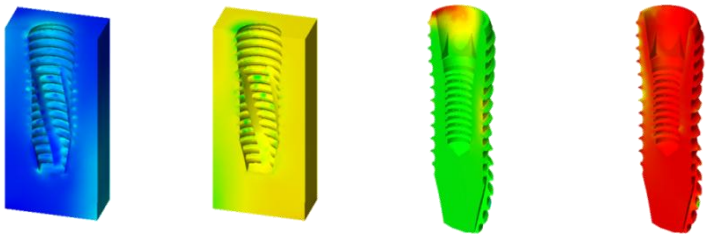




Post Processing

Stress review at implant contact area according to loading direction.

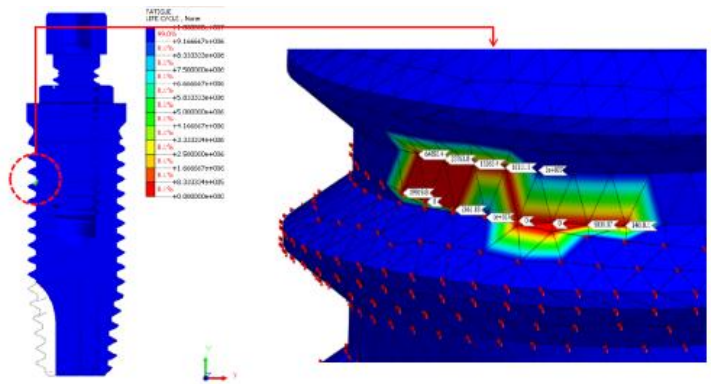
Maximum stress happens at cross corner of the maxillary bone under 30 degree direction load.



Maxillary bone principle stress

Fixture principle stress

For fatigue analysis result, from 400N, the screw of fixture has a short service life.



Fatigue life cycle under 500N