

- **Engineering**
- **Machinery**
- **Physics**

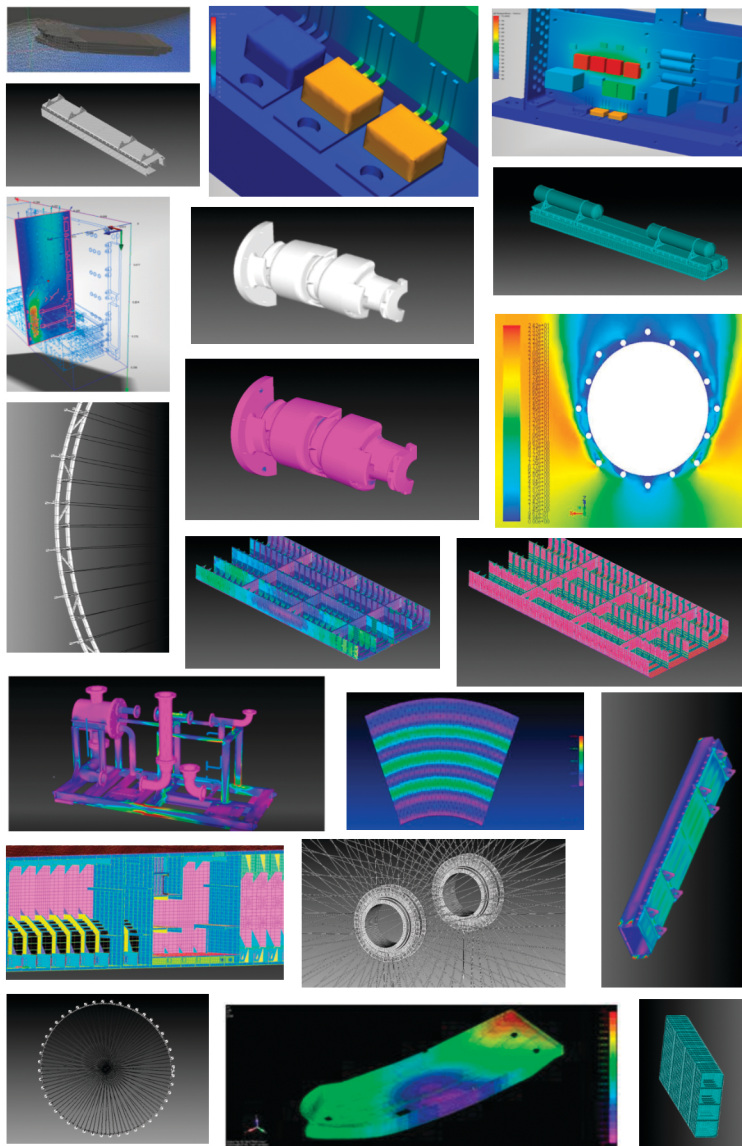
EMPUA

✉ Info@empua.com

☎ **+38 093 5513506**
+38 093 1464627

www.empua.com

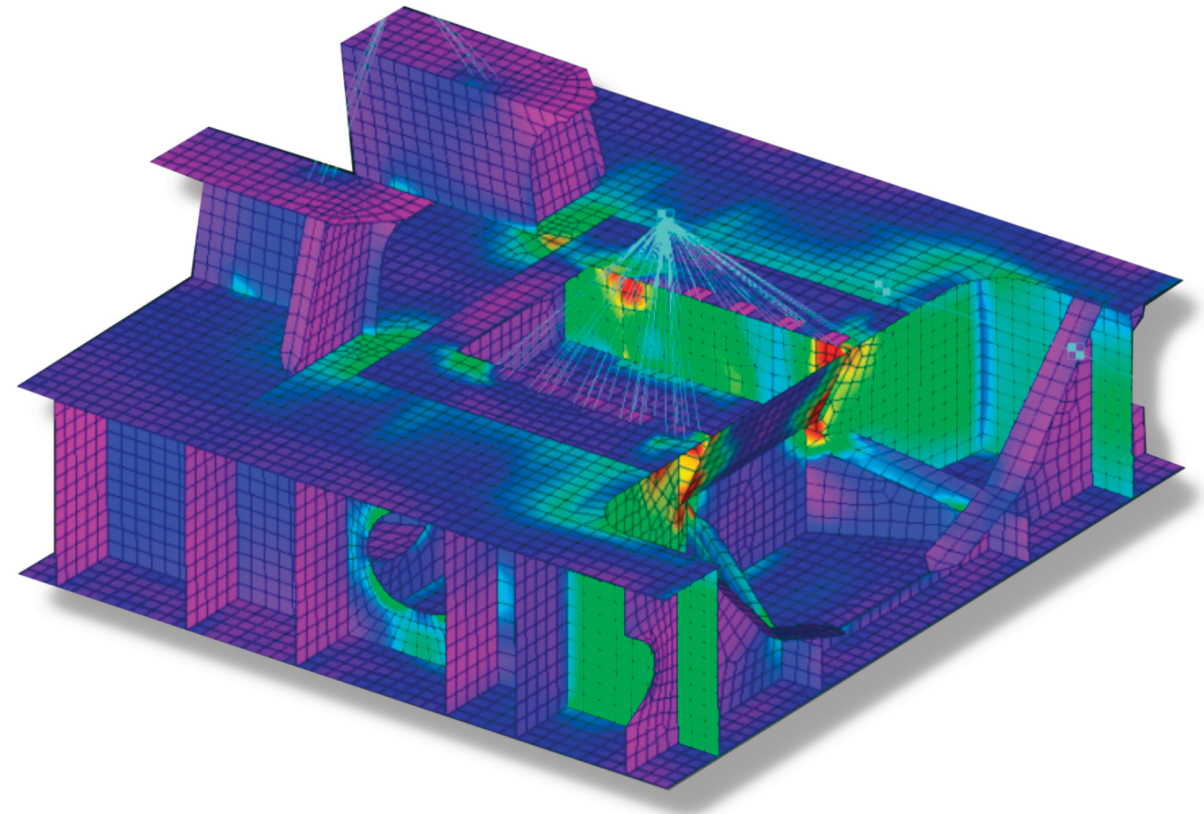
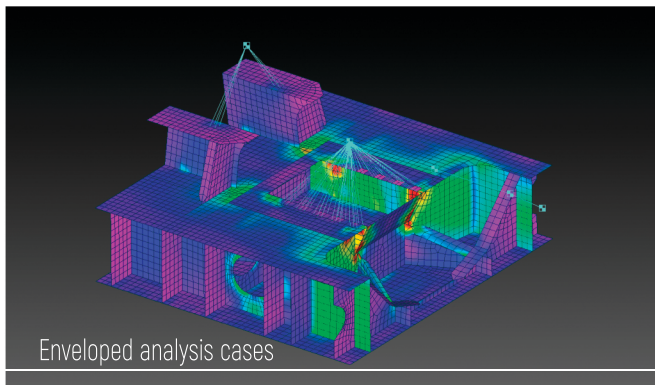
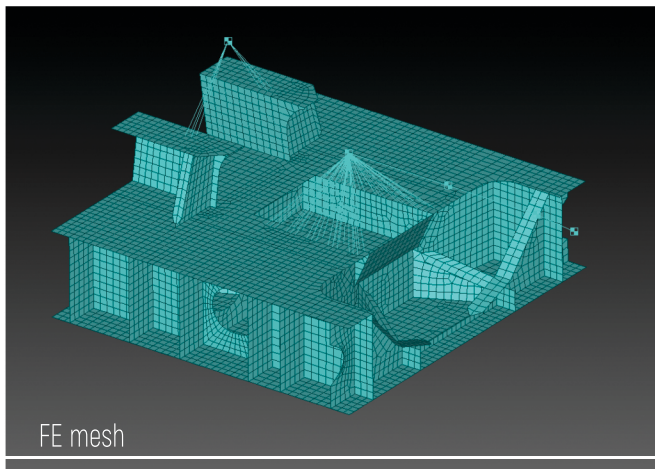
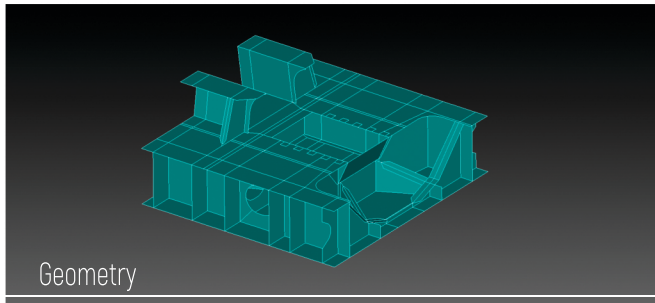
We help you with engineering tasks with different level of complexity.
We provide full-cycle engineering for machinery, gas/oil industries,
shipbuilding, architecture and robot technology



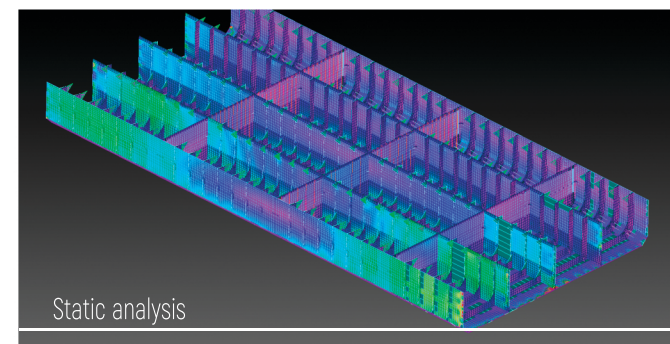
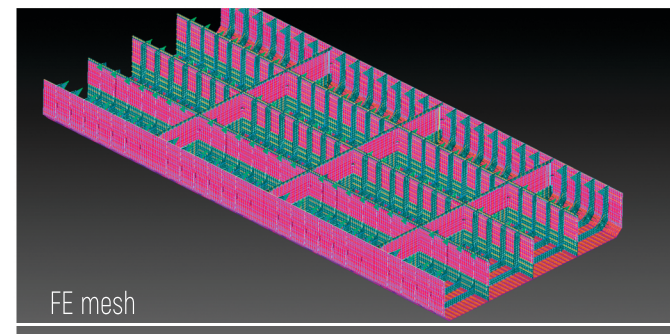
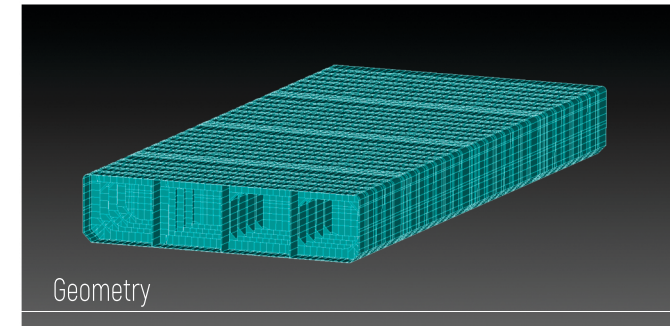
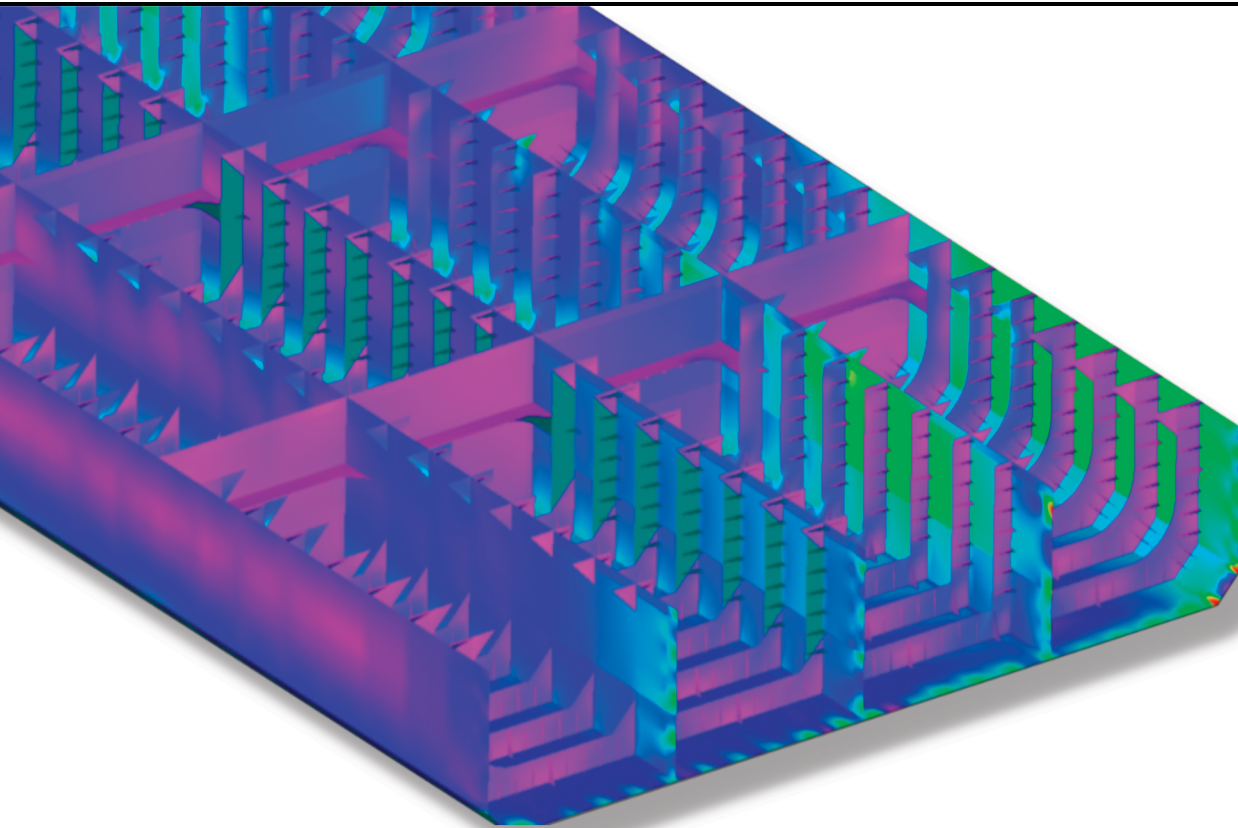
We are Ukraine local based team of engineers with wide range qualifications. Our background consist of numerous projects with different level of difficulty.

EMP specialized on:

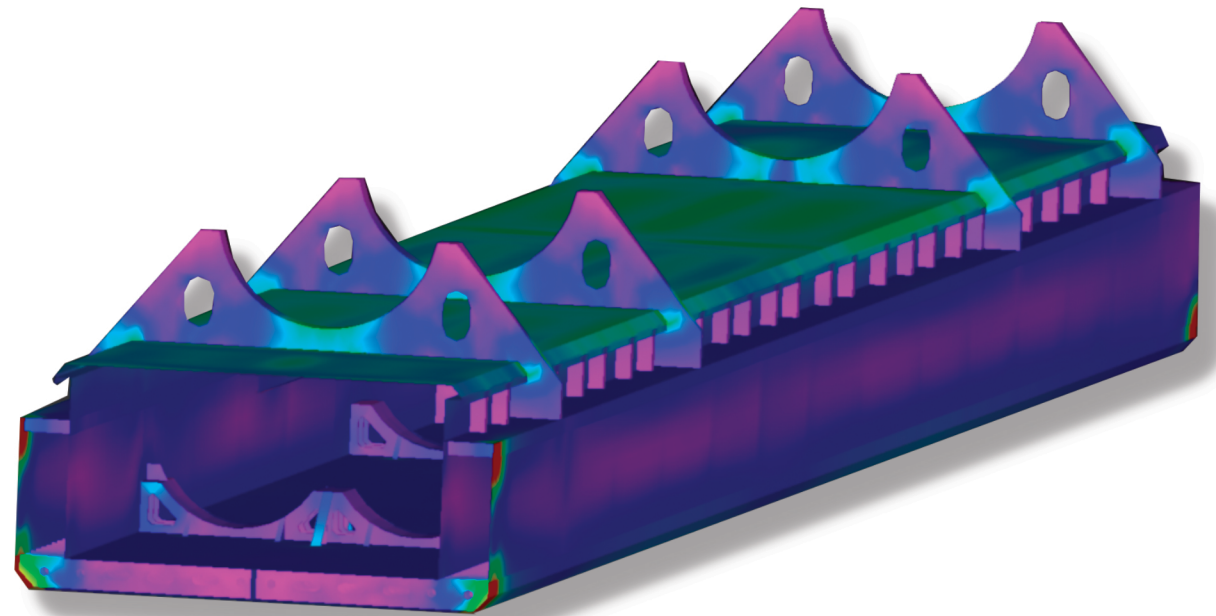
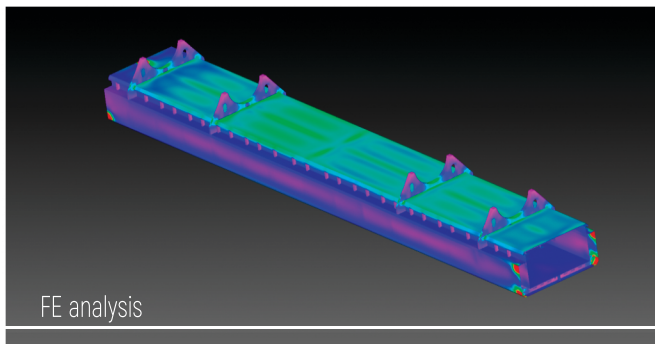
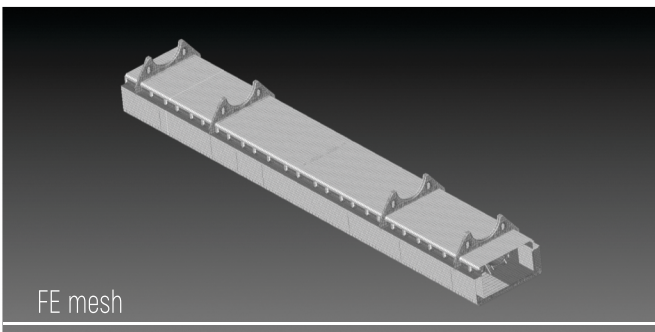
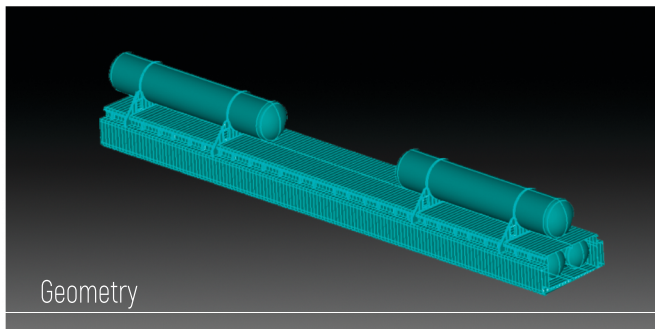
**FEA
CFD solutions
Engineering consulting
CAD modeling
Drawing**



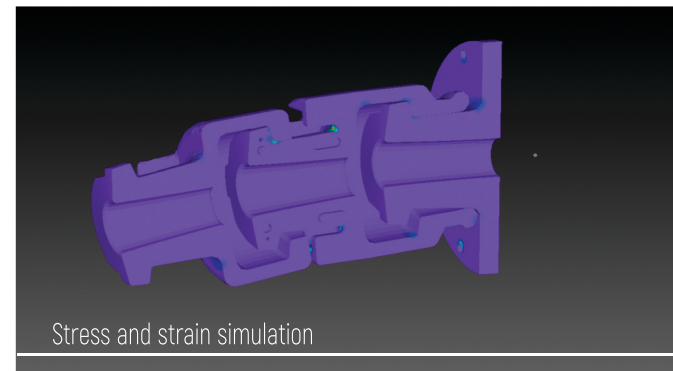
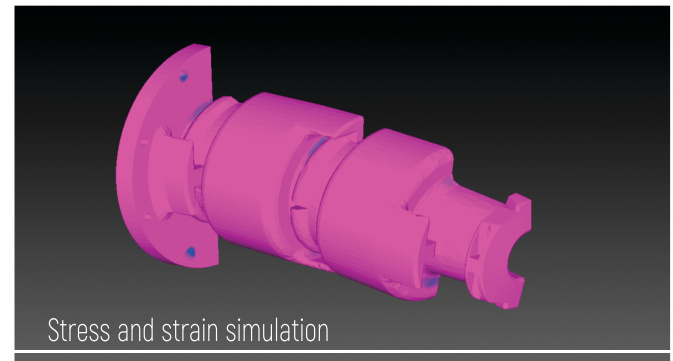
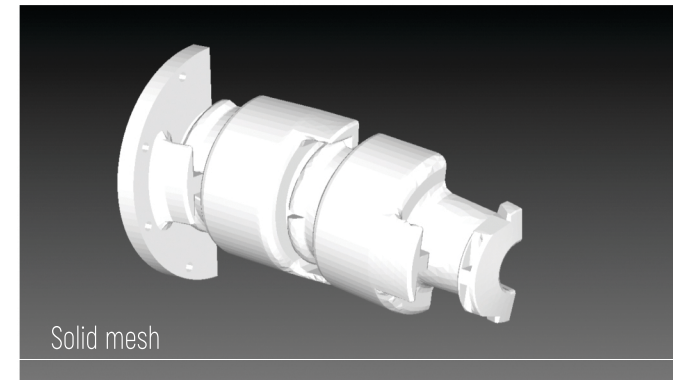
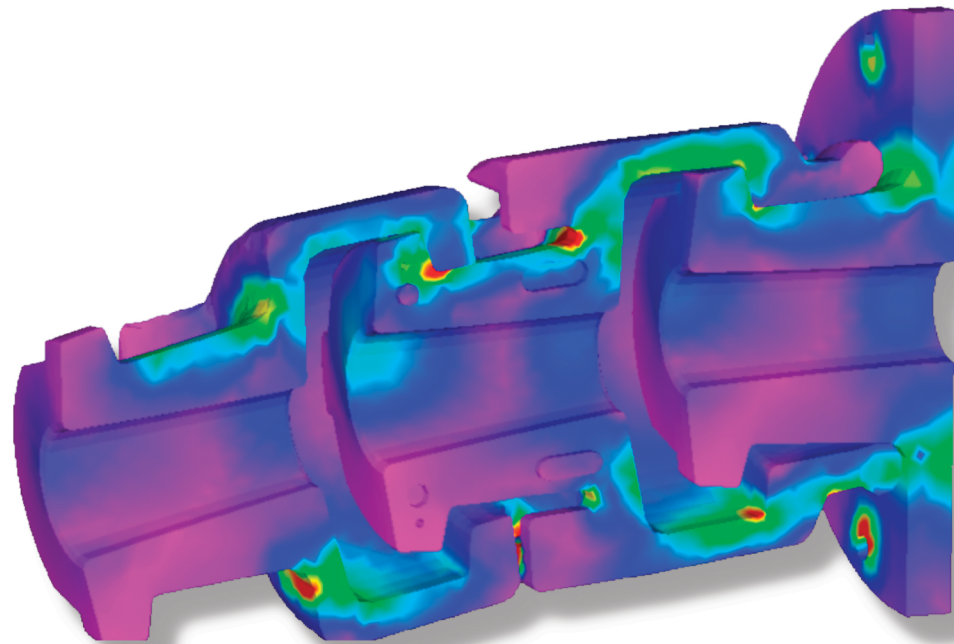
Femap/NX Nastran static simulation was performed to check the opportunity of installation new gearbox and dredge pump. The FE analysis had indicated on the need to modify the ship structure.



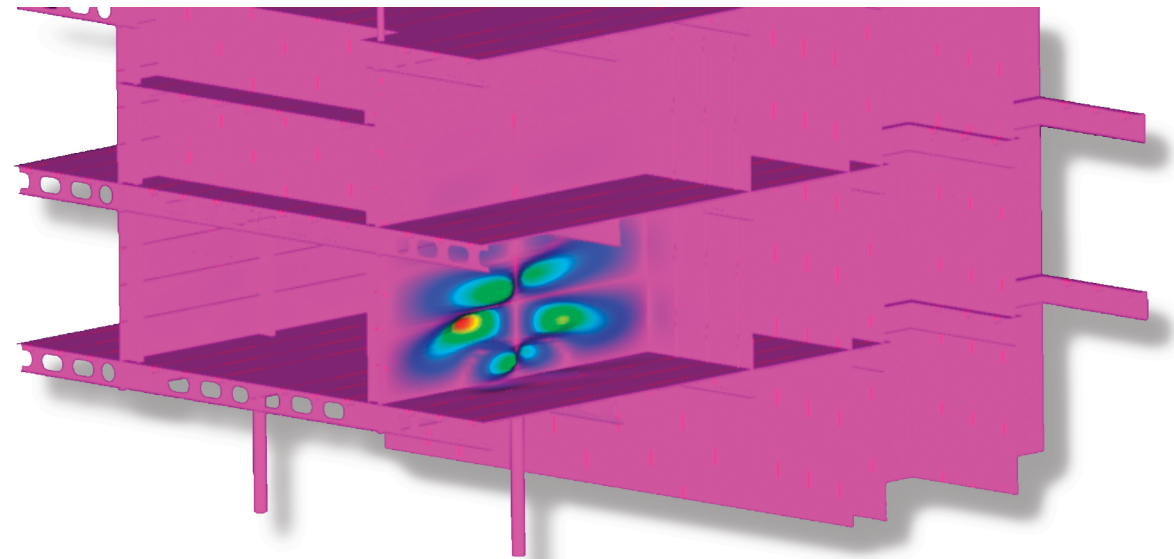
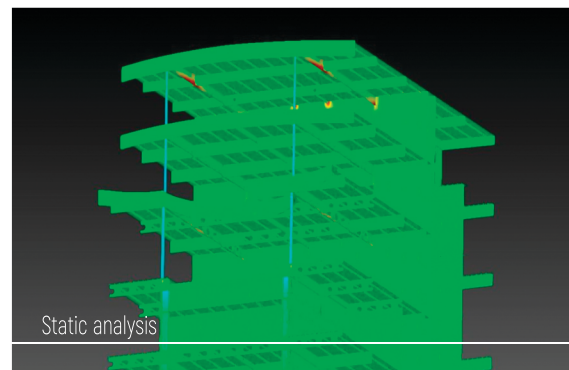
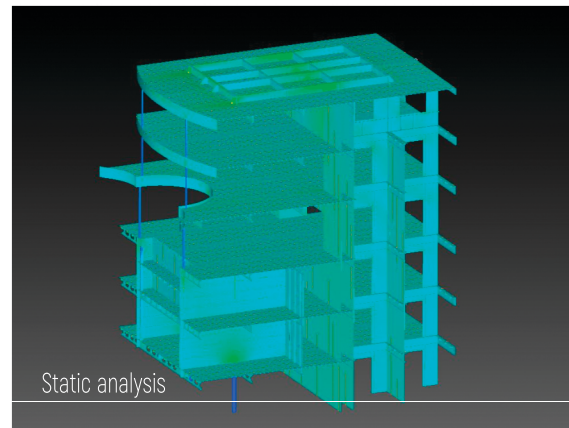
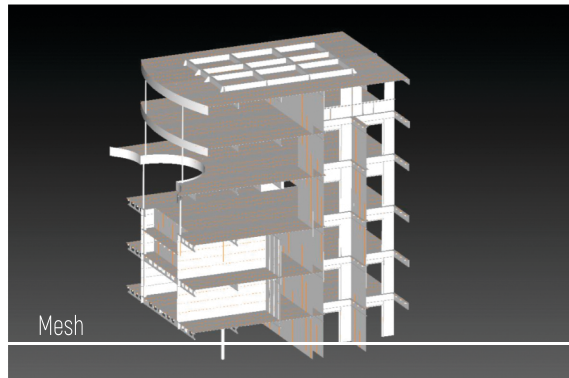
The goal of Femap/NX Nastran simulation was to confirm the performance of the gas carrier hull structure after installation of two additional gas tanks on the main deck of the ship. This model has simulated only cylindrical part of the hull, fore and aft sections has been replaced by affiliated masses. FEA simulation has showed that modified construction withstands normative bending loads.



The main goal of the Femap/NX Nastran simulation was to show the ability of heavy lifting cargo barge hull structure to withstand normative cargo load (3000 tons on 3 cargo jibs oriented to the portside of the barge). Three sections with cargo jib foundation frames placed on them have been simulated. The performance of the cargo barge construction has been totally confirmed.

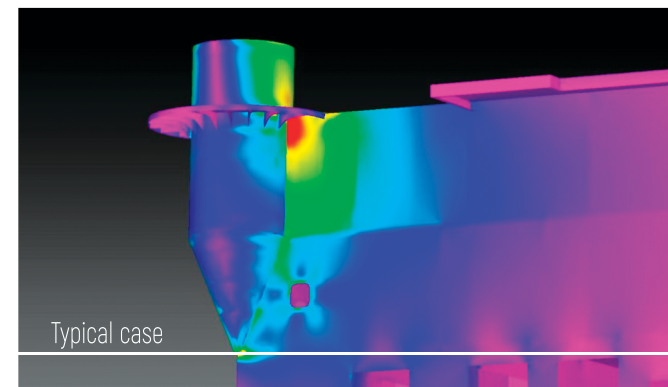
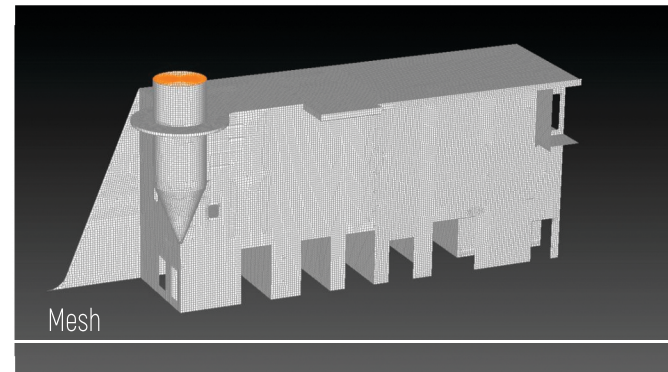
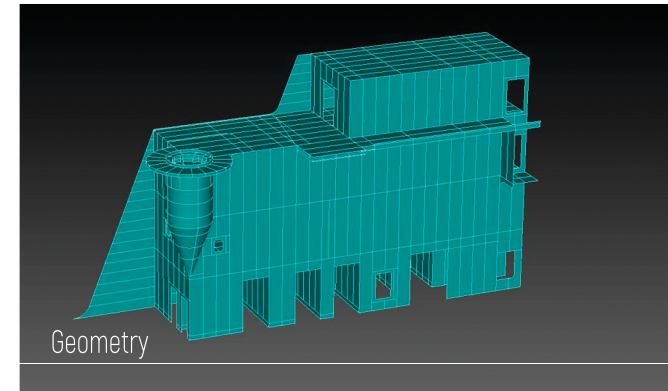
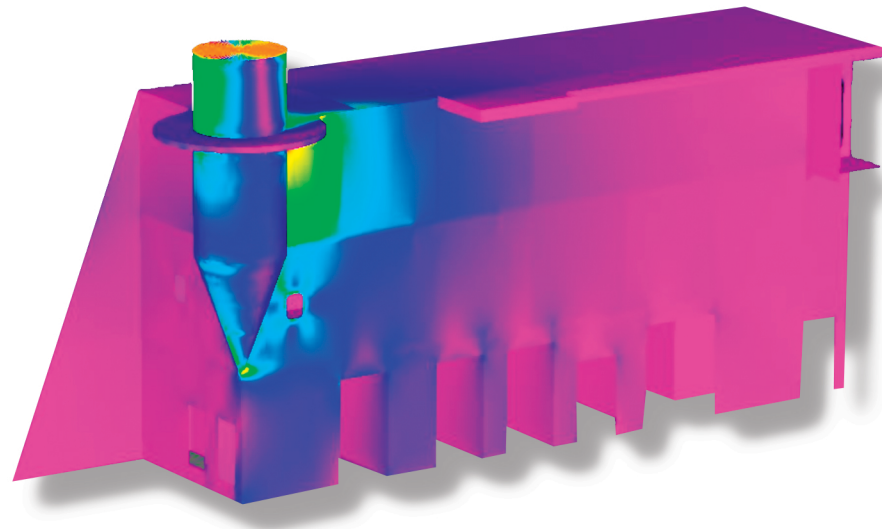


Nonlinear stress/strain simulation of the polymeric bend restriction unit installed on ocean cable layering ship. The goal of Femap/NX Nastran simulation was the solution of maximal stresses/strains in the unit under normative load. Also the bending radius of the unit has been predicted. FEA simulations has been performed for a number of typical bend restriction unit construction.



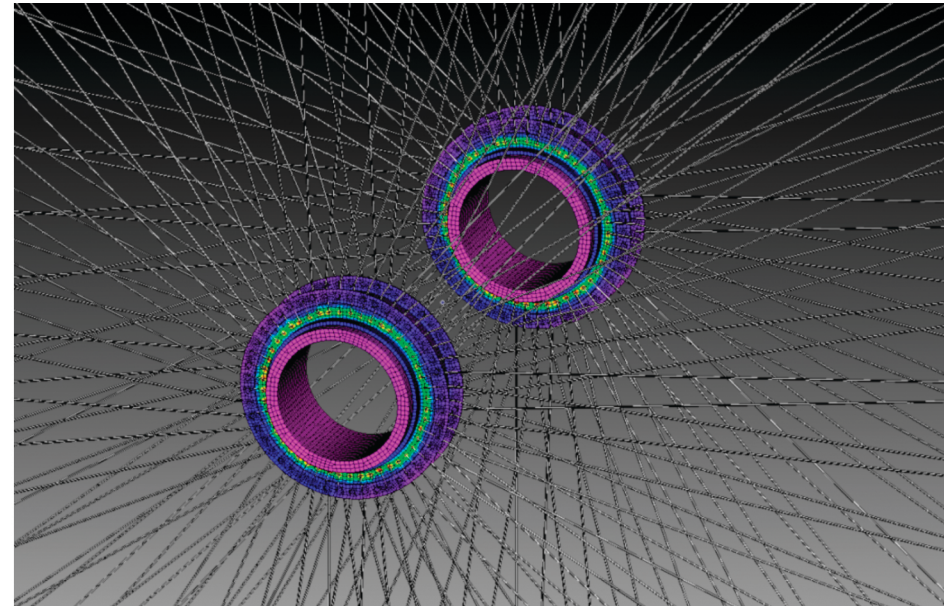
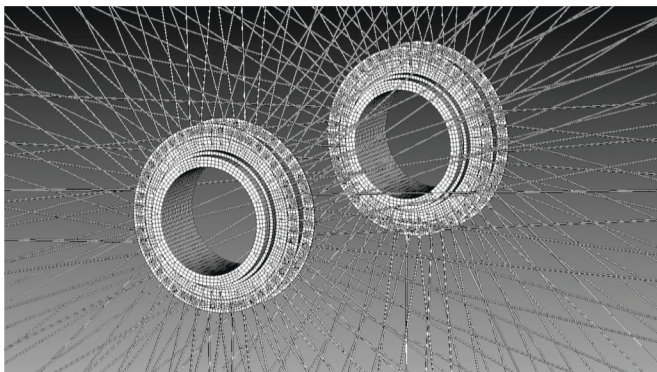
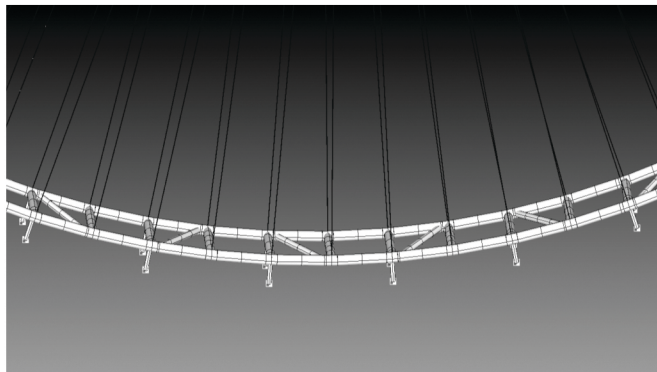
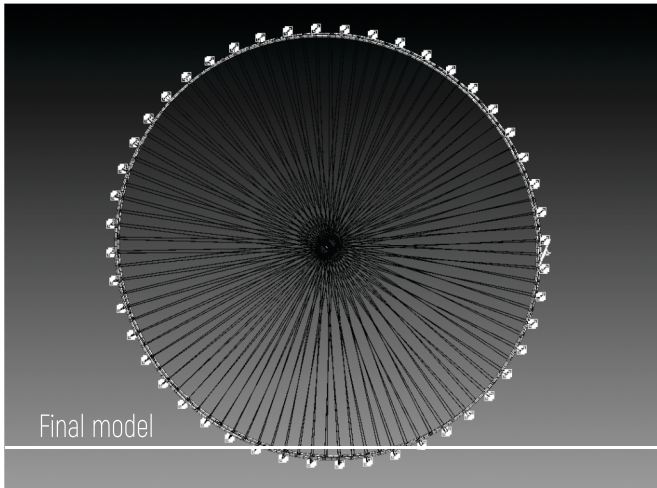
According to updated classification requirements an additional emergency power plant had to be added to the top deck of the number of cruise ships. The Femap/NX Nastran simulation had a goal to check the general performance of the modified structure.

Basing on the potential problems discovered by this simulation our team had proposed a number of ship structure modifications aimed at elimination of potential threats. Simulation series had included both initial and modified variant of the further structure.



The modification of the basic ship construction had included installation of the additional crane in the upper part of the ship superstructure.

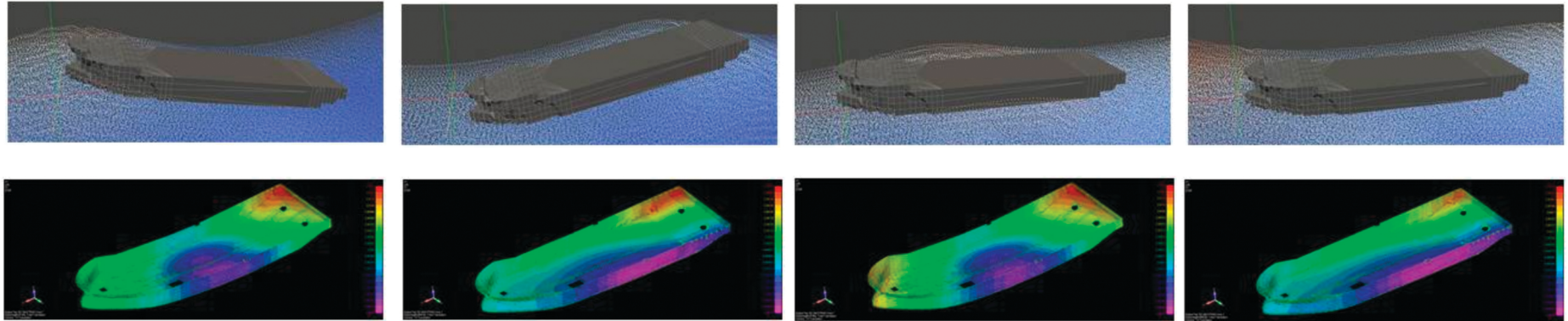
The Femap/NX Nastran simulation has included partial simulation of the ship superstructure construction with future crane foundation in it to check the general structure performance on lifting operation.



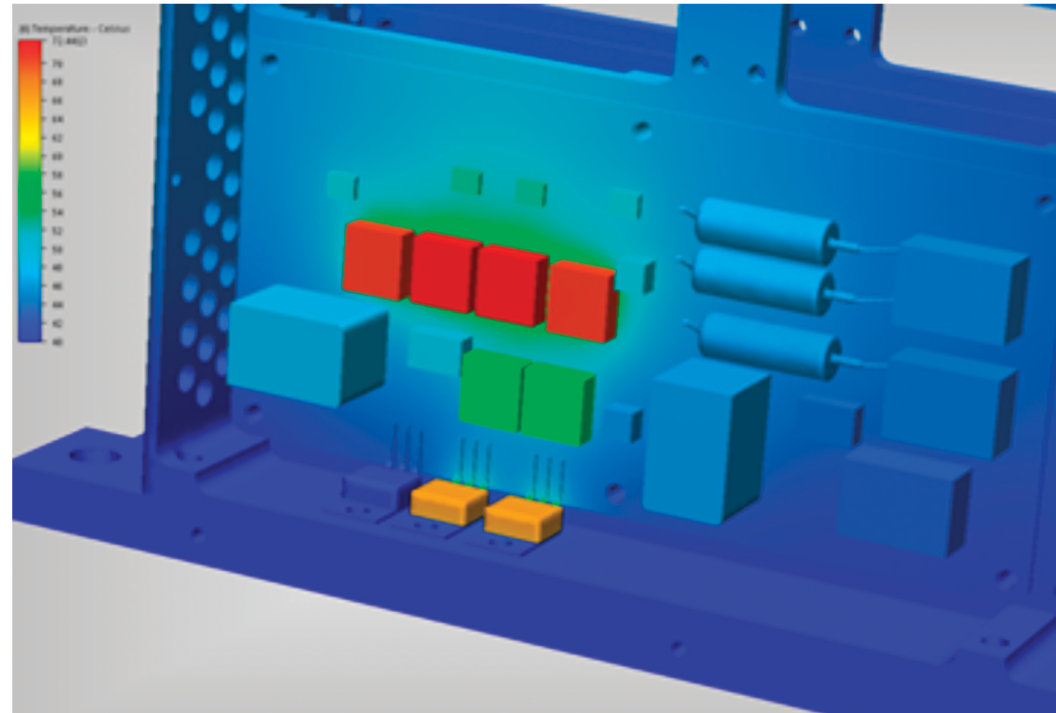
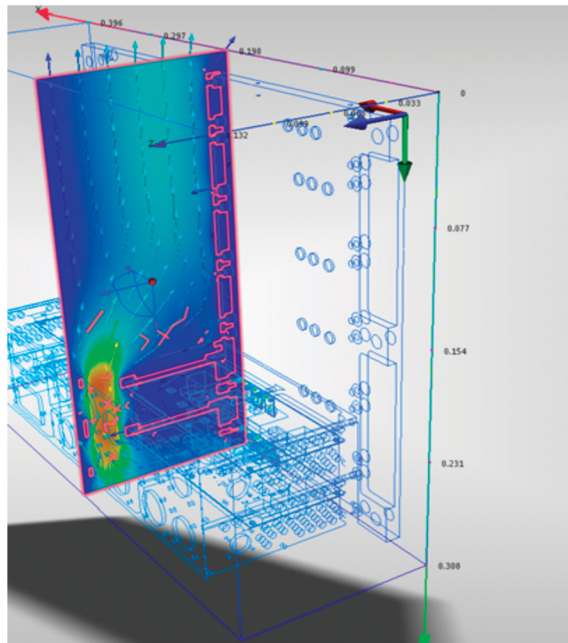
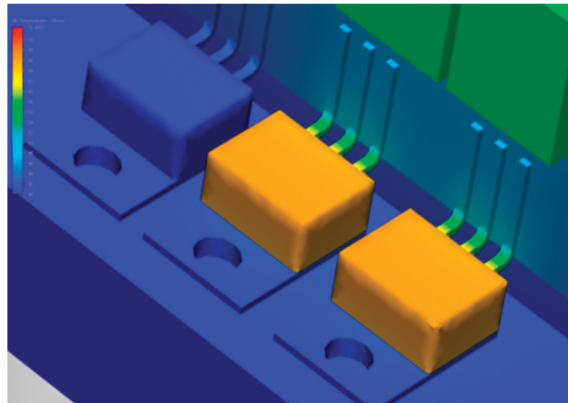
Finite element transient simulation of stick-slip effect in main bearing of Ferris wheel. This case has been studied the effect of velocity dependent friction ratio to the fluctuations of wheel rotation speed.

This case has been performed using Femap/NX Nastran simulation environment.

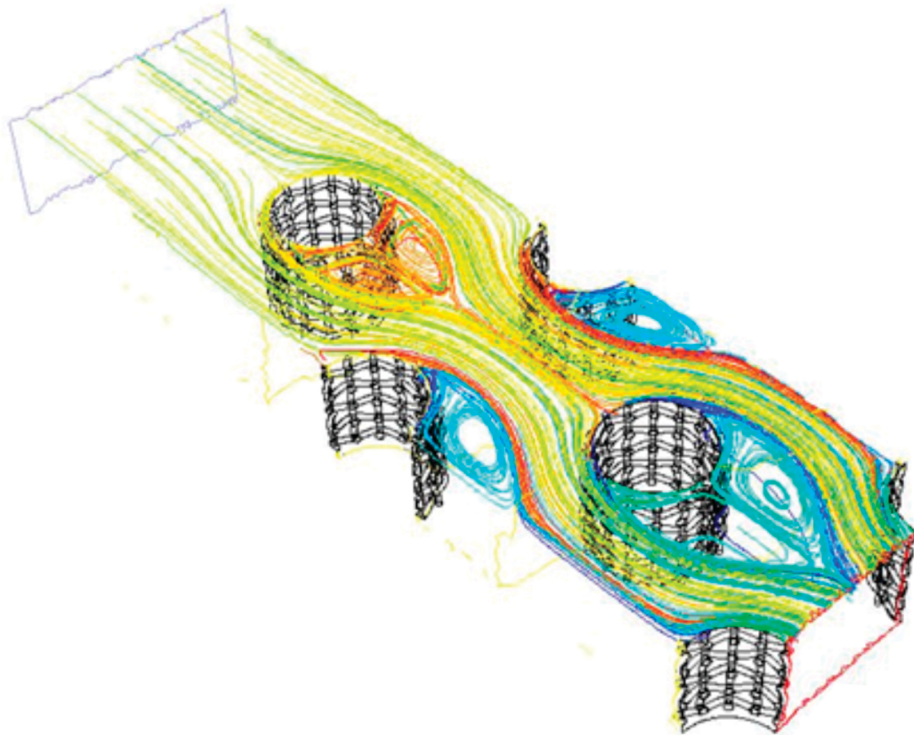
This case has been performed using Femap/NX Nastran simulation environment.



Smoothed particle hydrodynamic (SPH) simulation of ship hull / free water surface interaction. An example of non-mesh CFD method application for marine industry. DualSPHysics simulation engine/Paraview visualization has been used for this project. The water pressure field obtained by SPH solution has been used as an input for the Femap FEA simulation of the stress-strain state of the ship hull at the next stage of the simulation process.

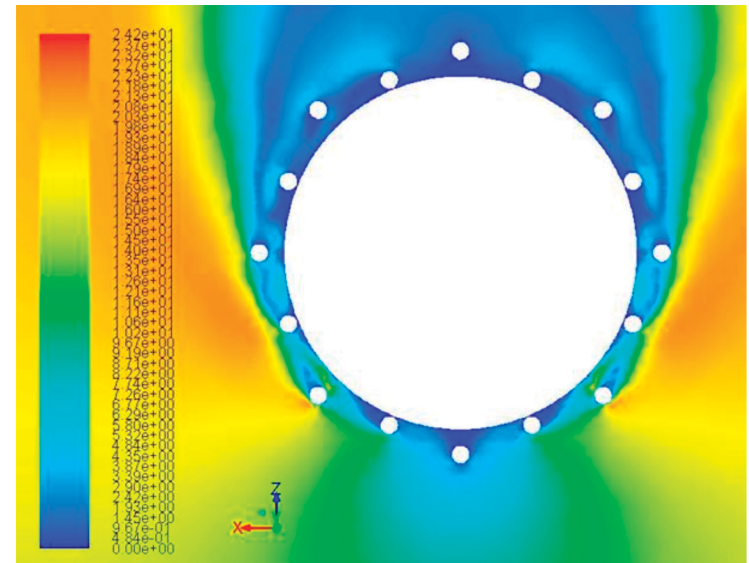


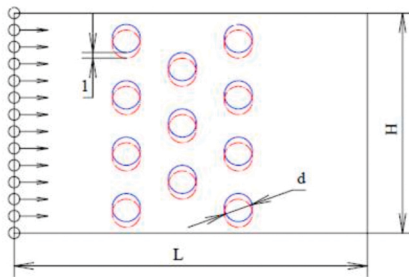
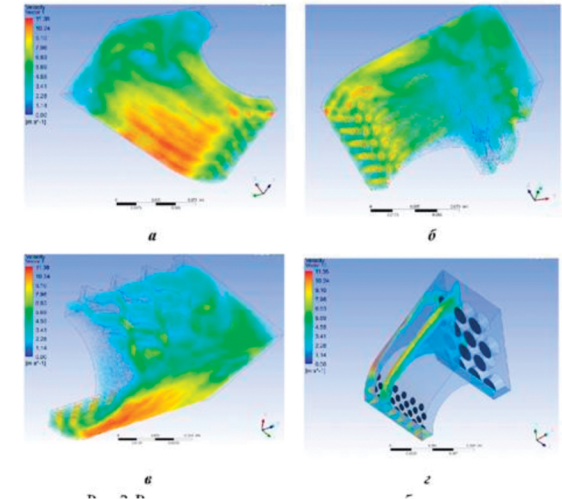
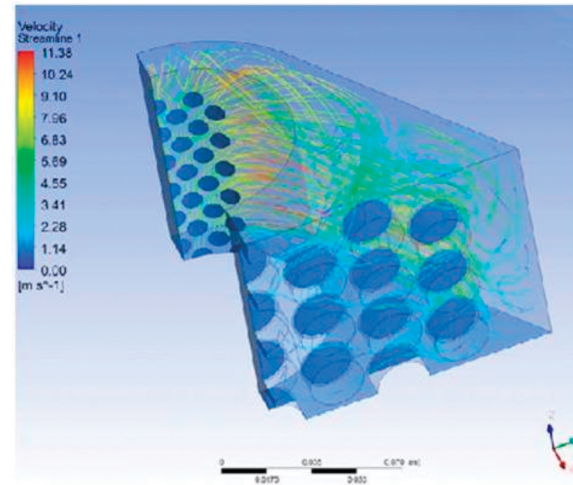
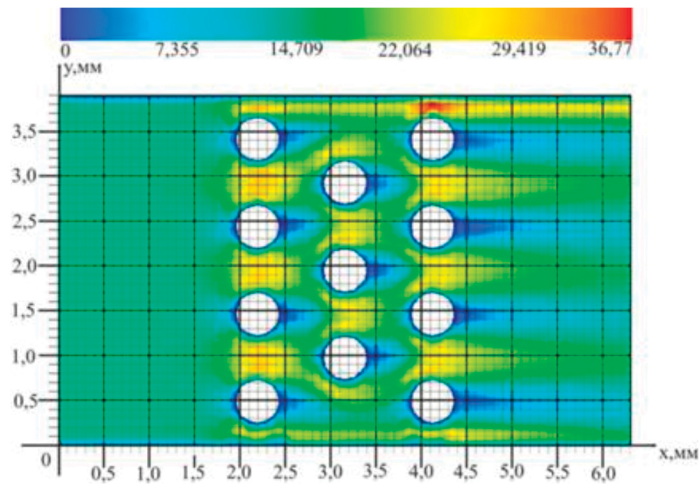
Thermal conductivity/convection heat transfer simulation for vacuum and air environment. Power supply unit PCB has been modeled as 3-layer composite (aluminum, copper, plastic) due to specialty of this equipment. Autodesk CFD Simulation has been used as a simulation tool.



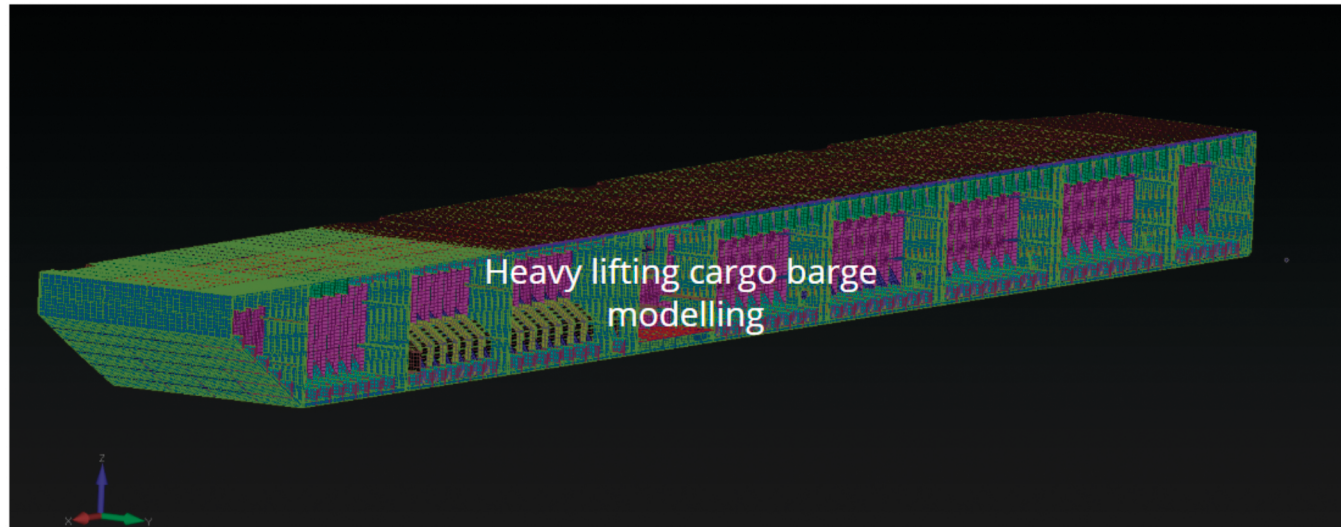
Scientific research case clarifying heat transfer ratio values for the tube heat exchanging surfaces ribbed by metal grid. The CFD models performed in Ansys Fluent simulation environment have been characterized by very high detalization of the ribbing elements.

Analytical system of heat transfer equations characterizing this type of heat exchanging surfaces has been obtained as a result of the simulation case.

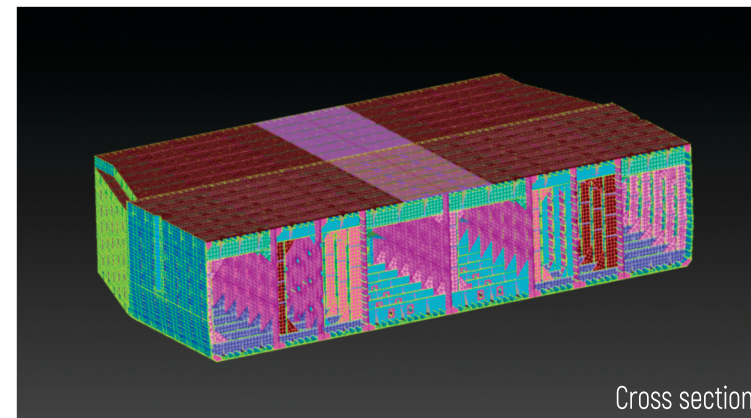
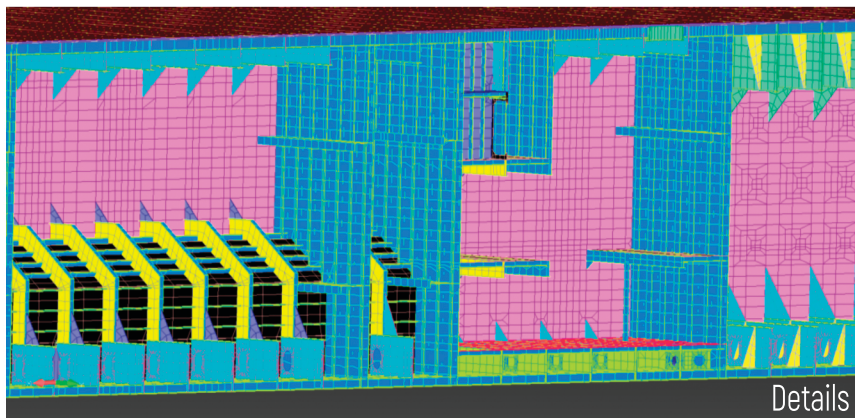




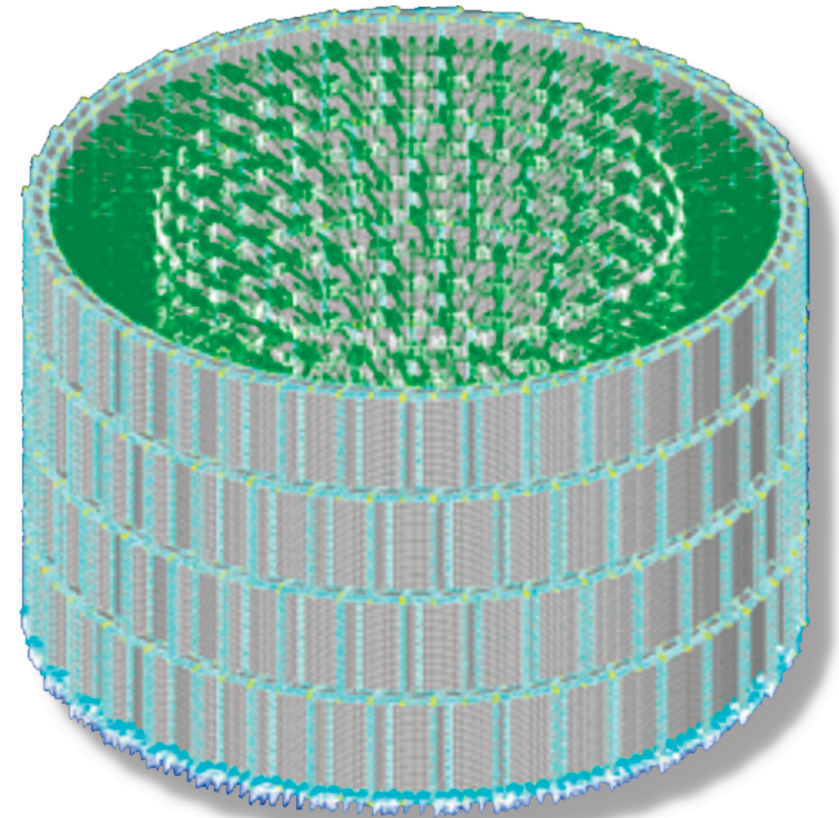
The model has been performed using Ansys Fluent tool. The model includes RANS-based steady/transient flow simulations, modeling of high frequency pulsation of the coagulation grid and particle track simulation.



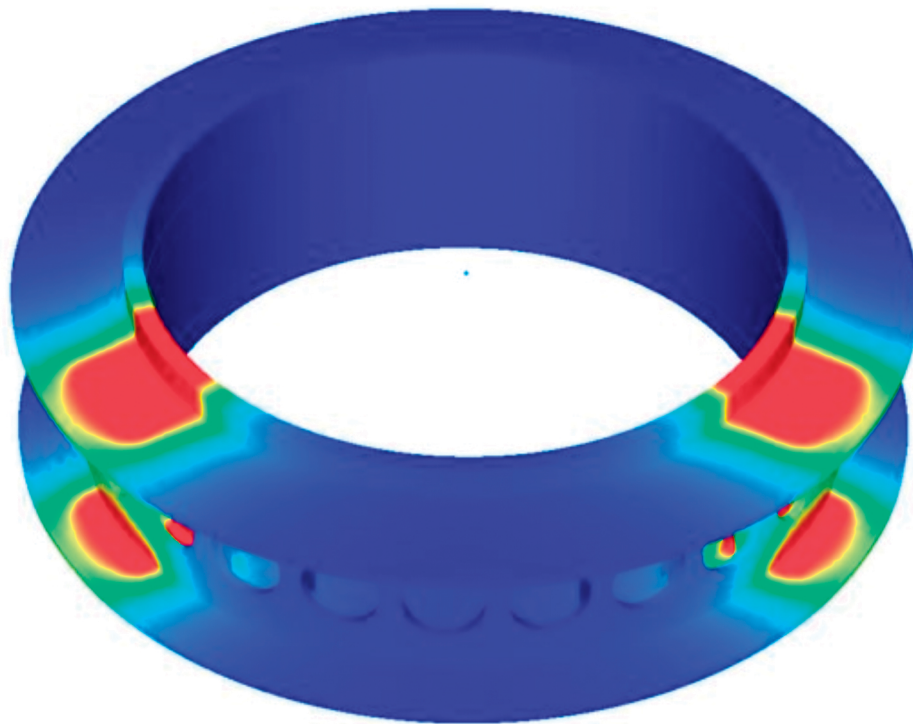
Finite element modeling of heavy lifting cargo barge (full hull plate / beam model including all internal constructive elements). The model has been performed using Siemens Femap. The project has included the ultimate level of FE-mesh optimization under customer's requirements.



Femap-based application for automatic FEA-modeling of modular plastic water tanks. Technical applications designed using Femap API is a perfect solution for serial repetitive FEA tasks like the one we had on this project. Using as input parameters only tank diameter, height, and thickness of the panel rows this application provides geometry design, mesh generation, constraint and load input, NX Nastran solution of the case and Microsoft Office formatted report (Word document or Excel spreadsheet) including generation of all necessary graphical and numerical outputs.



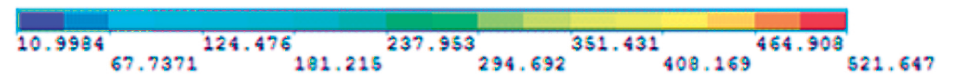
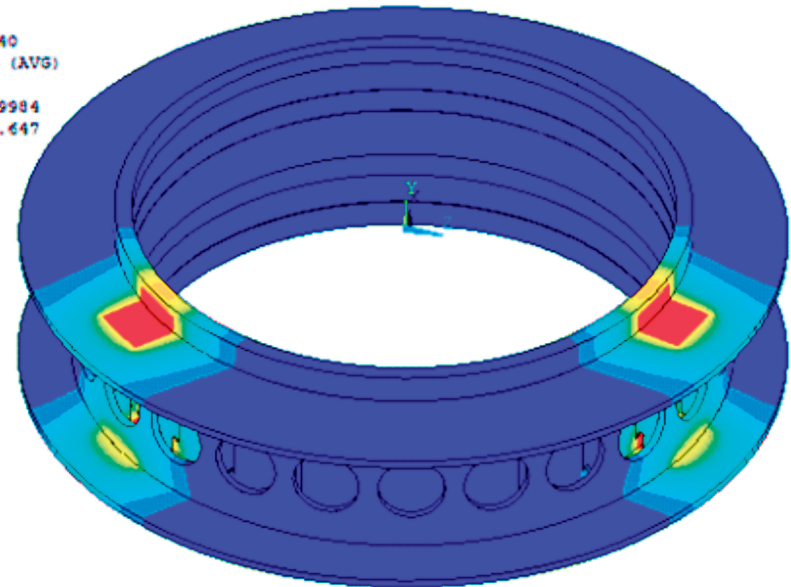
The maximal level of heat stresses has been studied in this case. The performance of the Ferris wheel main bearing after constructive modification of the hub has been checked. Current simulation has been performed using Ansys APDL and Femap tools.



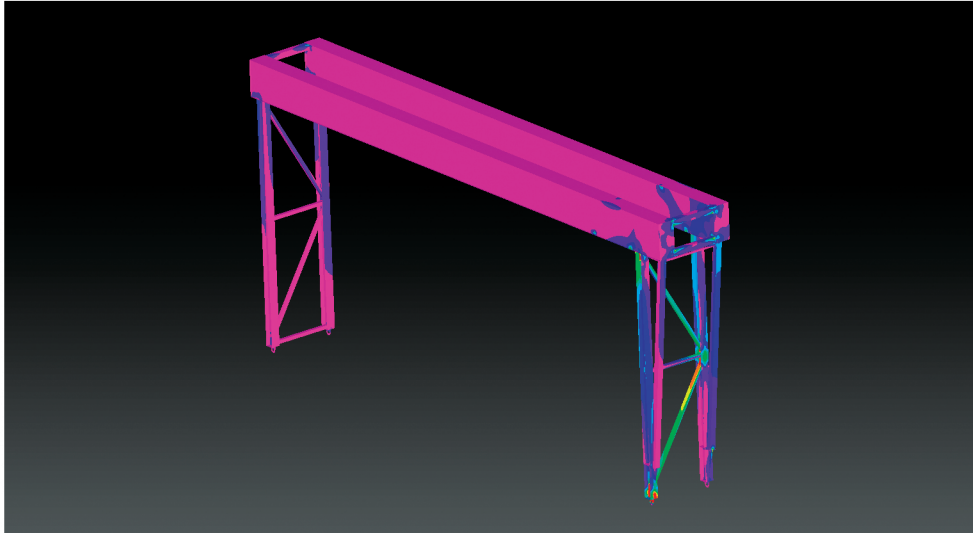
Femap analysis

```

1
NODAL SOLUTION
STEP=1
SUB =300
TIME=25740
TEMP (AVG)
RSYS=0
SMN =10.9984
SMX =521.647
    
```

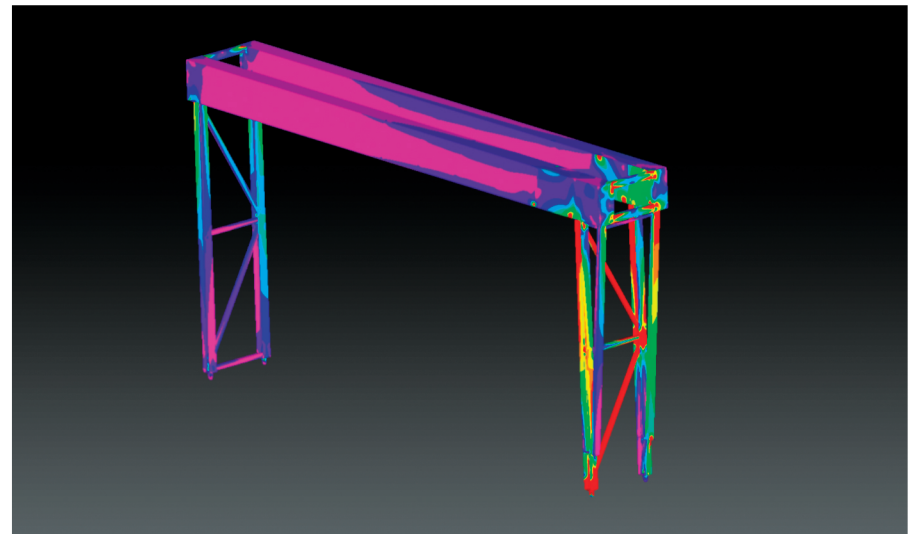


Ansys APDL analysis

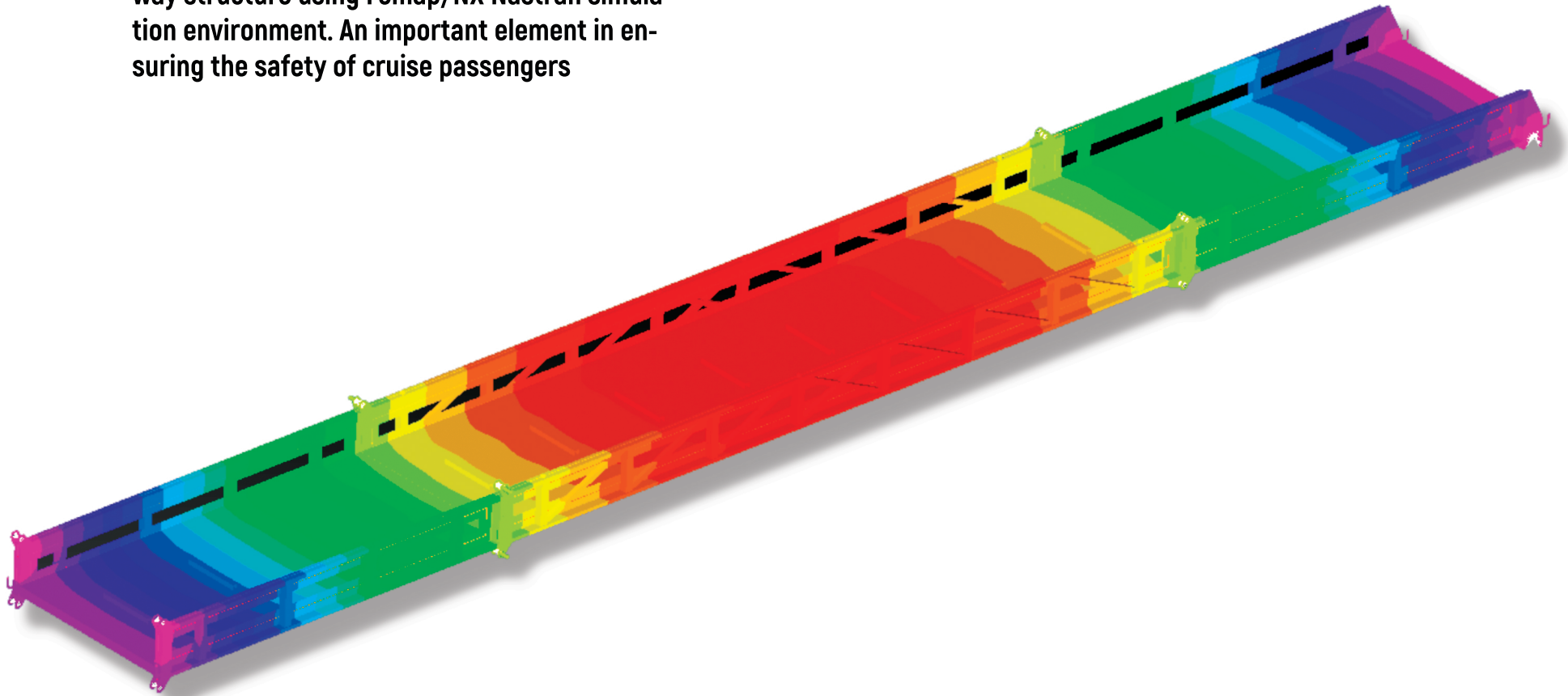


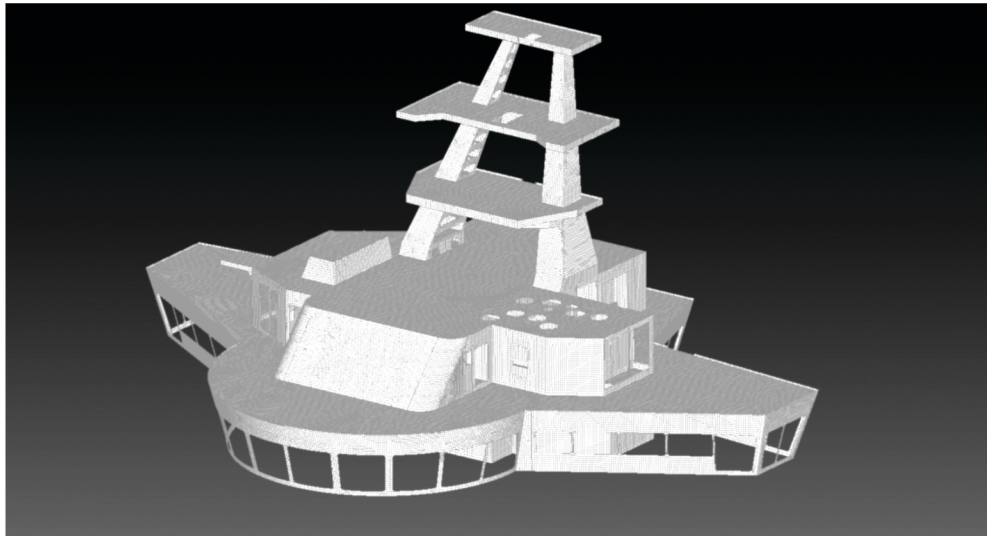
Static and buckling simulation of gentry crane structure. Finite element model built with Femap and analyzed using NX Nastran.

The main objective of the modeling was Eurocode compliance check.

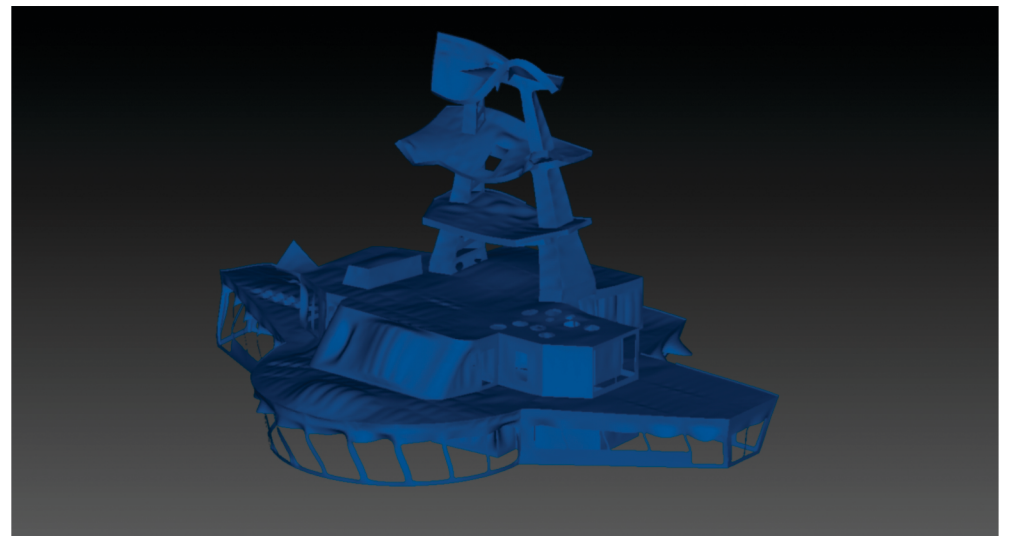
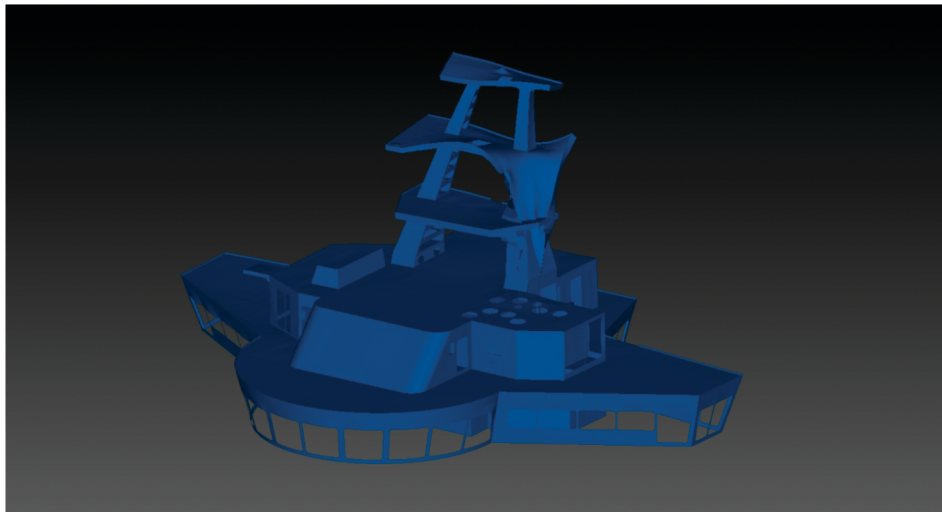


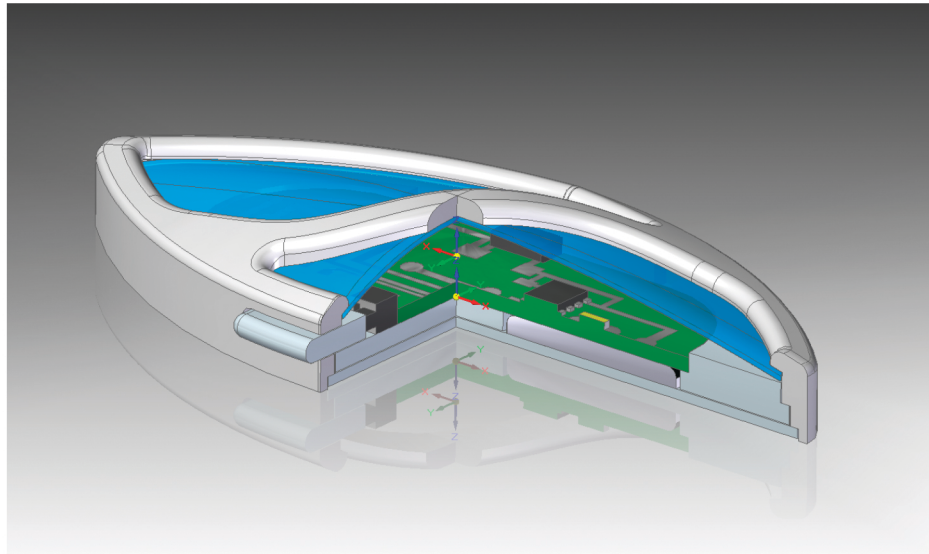
Static and buckling simulation of marine gangway structure using Femap/NX Nastran simulation environment. An important element in ensuring the safety of cruise passengers



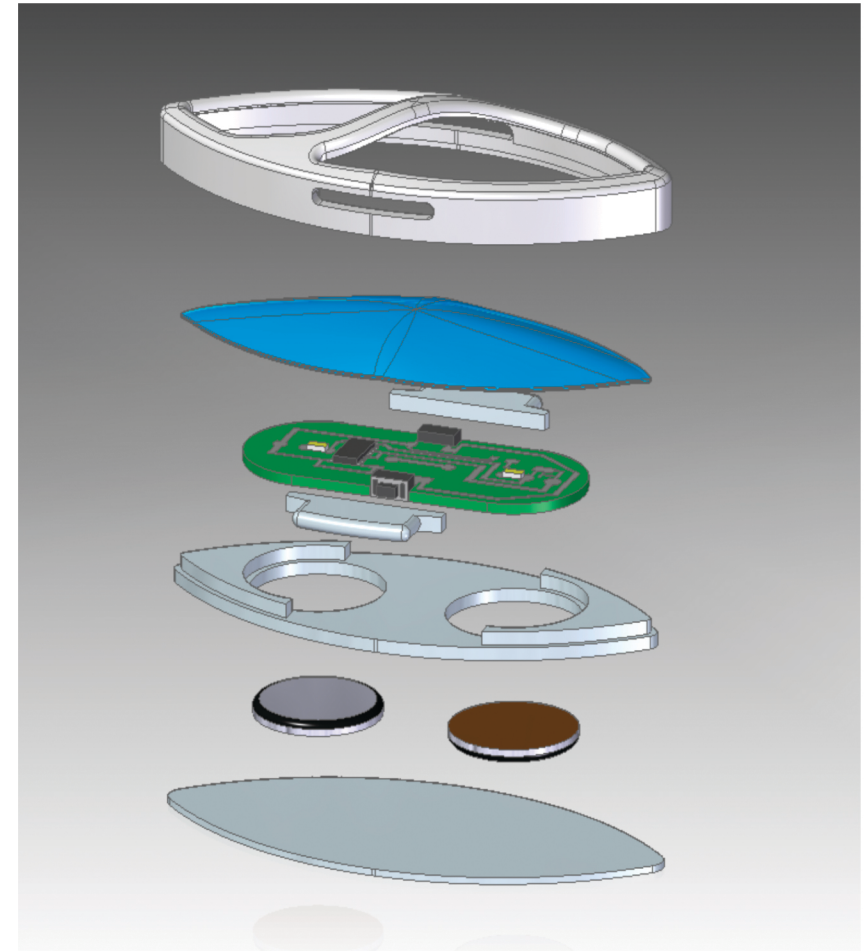


Vessel superstructure vibration analysis
Vibration analysis.
Eigenvalue analysis of ship superstructure using Femap/NX
Nastran Software. Detailed simulation of 3 top decks and mast
construction.



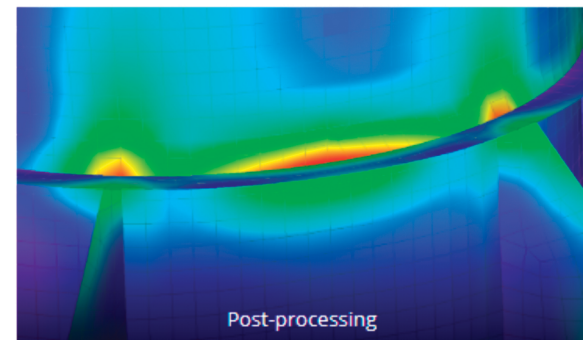
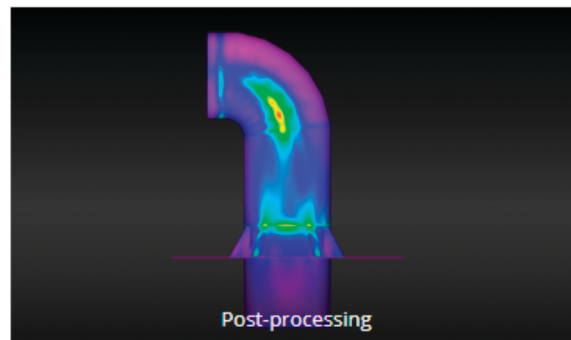
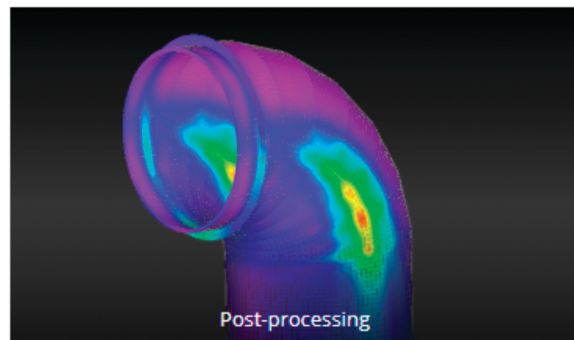
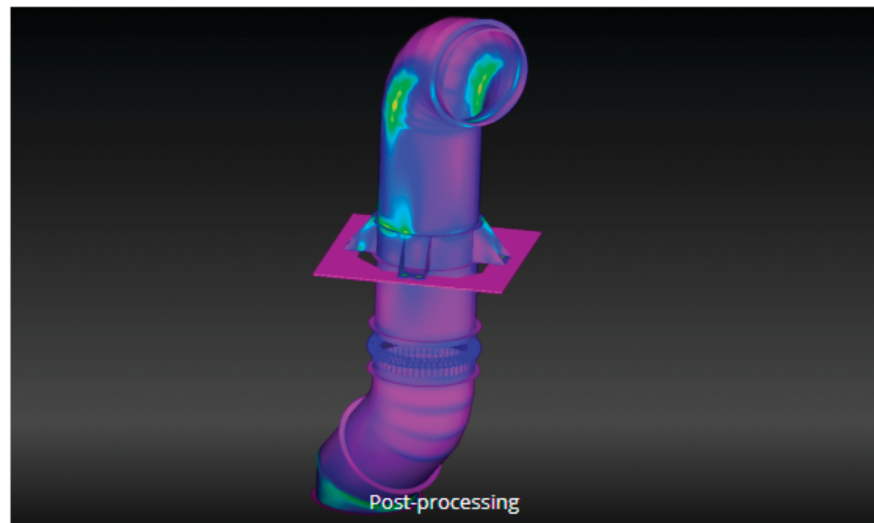
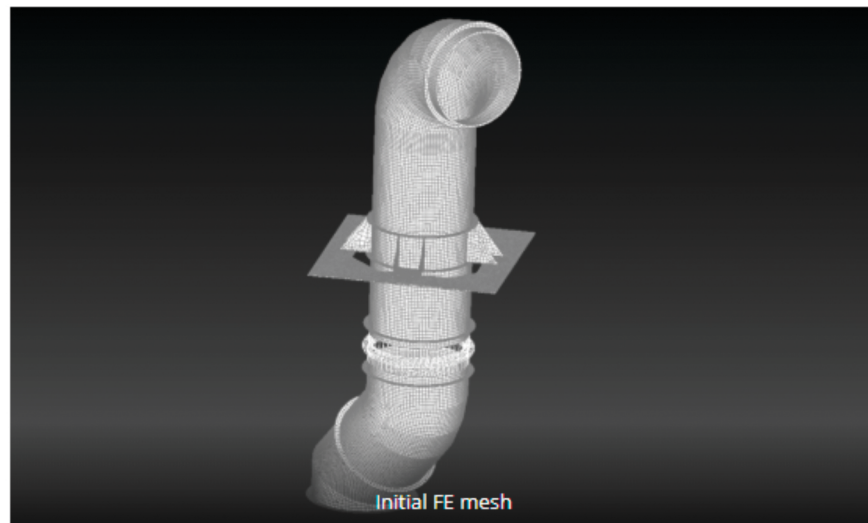


his project has included the full cycle of mass produced product design (from conceptual development to prototyping)



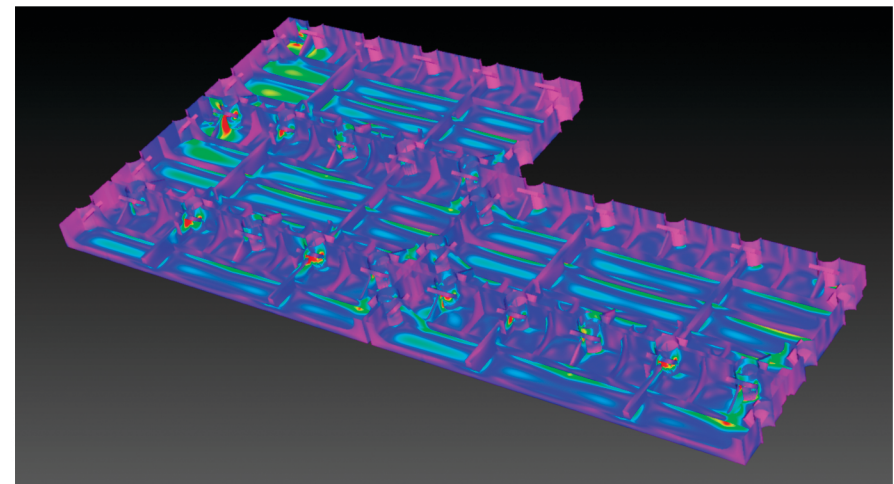
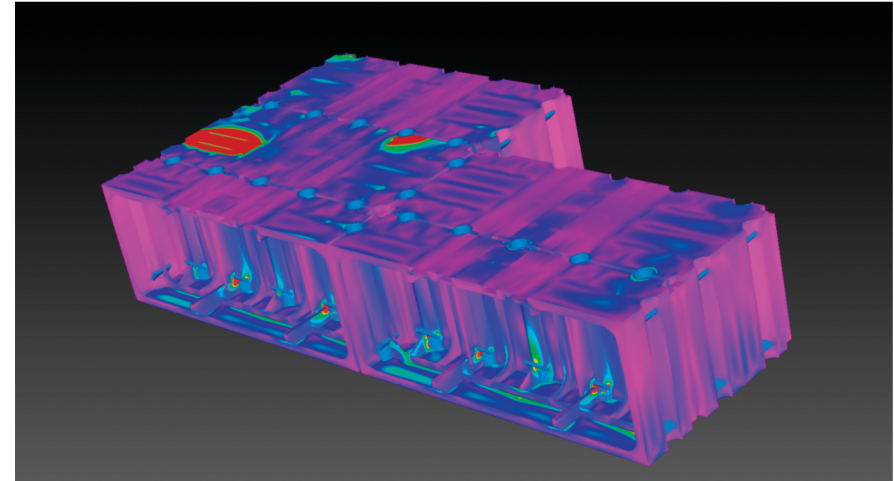
Finite element simulation of marine gas duct behavior under heeling/trim acceleration loads. Has included the simulation of flexible connection between the construction parts.

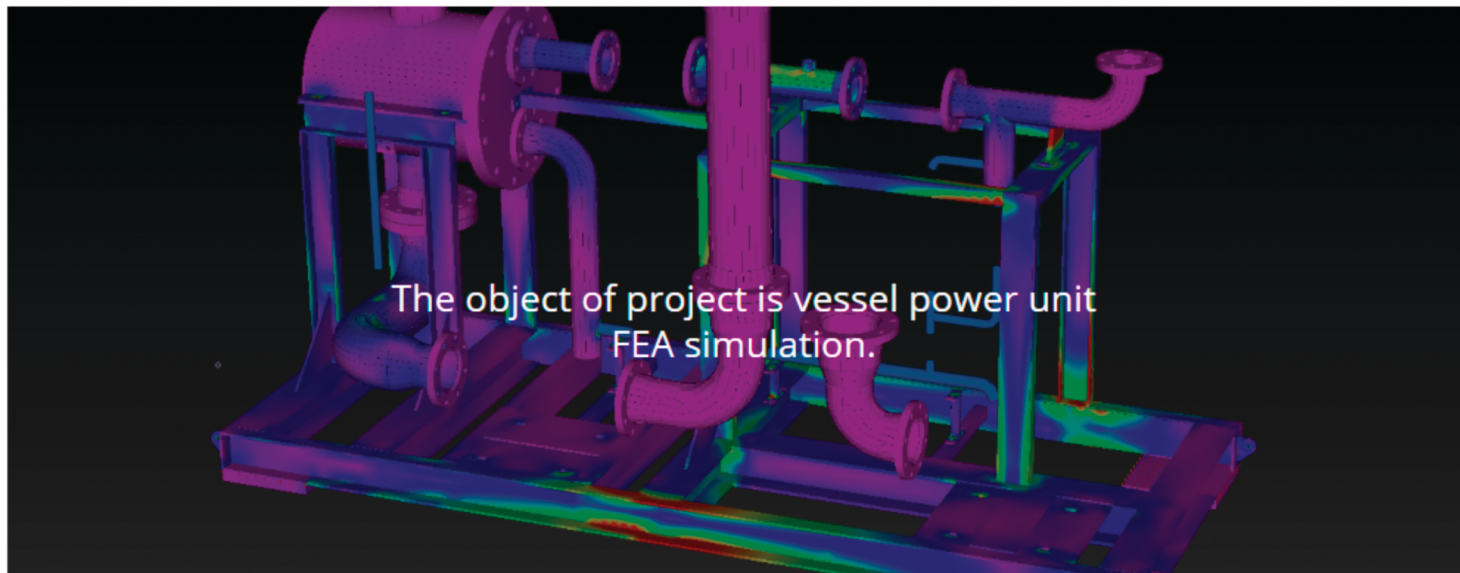
This project is the typical task of marine power plant development.



This Femap/NX Nastran model had been performed to check more effective design solution between 2 types of connecting elements (hook or screw jack connection between container-like constructive modules).

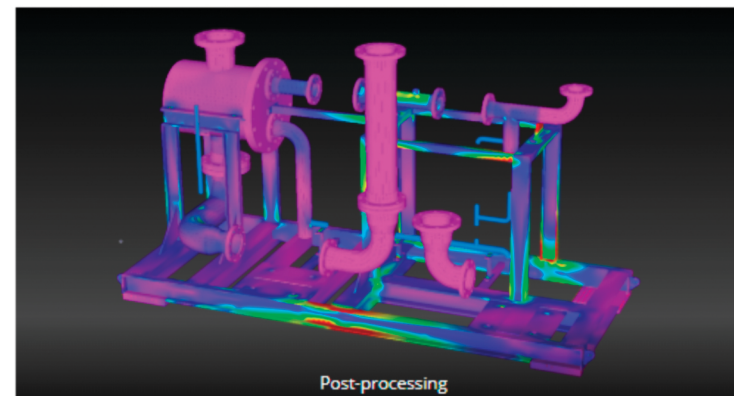
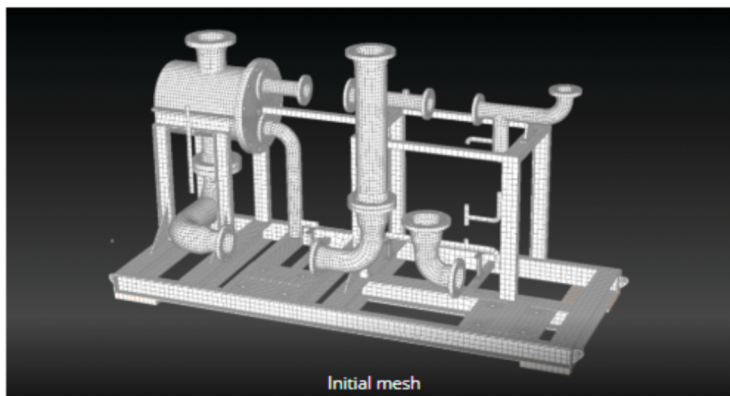
Nonlinear contact pairs behavior has been considered.





The effect of heeling/trim acceleration loads has been studied. The forces applied as X- Y- Z accelerations.

Femap/NX Nastran simulation environment has been used. The model consist of plates springs and rigid elements.





EMPUA

We help you with engineering tasks with different level of complexity.
We provide full-cycle engineering for machinery, gas/oil industries,
shipbuilding, architecture and robot technology

- Engineering
- Machinery
- Physics

 Info@empua.com

 +38 093 5513506
+38 093 1464627

www.empua.com