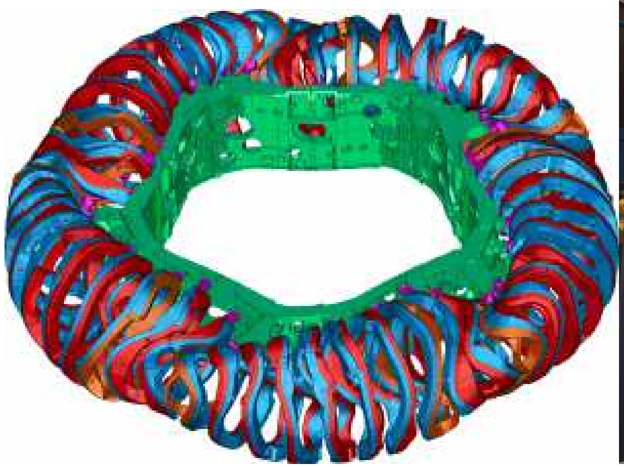
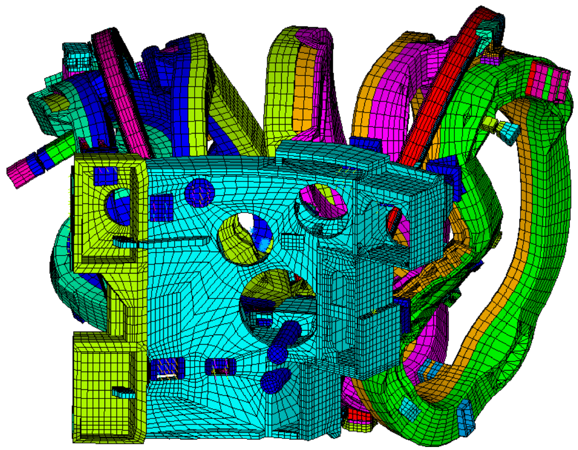
Finite Element Analysis of Critical Central Connection Elements of W7-X Stellator Coil Support System

The objective of Wendelstein 7-X project is the stellator-type fusion reactor. In this device plasma channel is under control of magnetic field coming from magnet system of complex shape, made of 70 superconducting coils symmetrically arranged in 5 identical sections. Every coil is connected to central ring with two extensions which transfer loads resulting from electromagnetic field and gravity.
During operation at a service temperature (ST) of 4K the superconducting coils of the W7-X magnet system exert high electromagnetic loads. Therefore, the detailed analysis of the coil - central support connections, the so called Central Support Elements (CSE), is a critical item for W7-X. Each coil is fastened to the CSS by two central support elements (CSE).
The aim of this work was to analyse mechanical behaviour of the bolted connections using detailed 3D finite element models (including bolts , washers, welds etc). The Global Model of the structure, analysed by Efremov Institute in Russia, provided information about the loads acting on the connections.

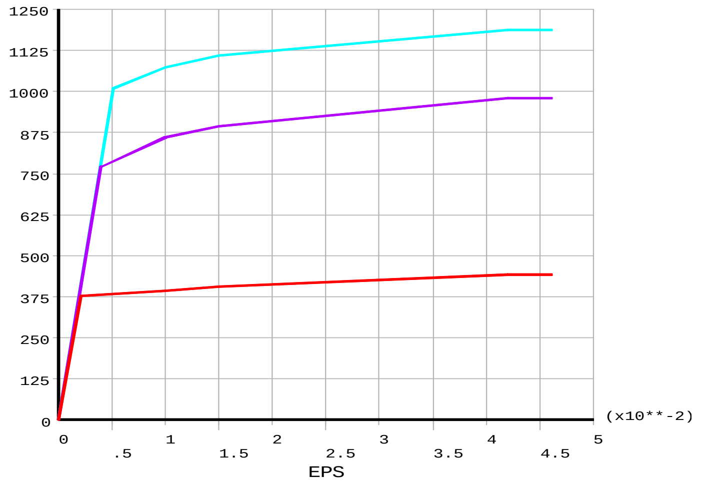


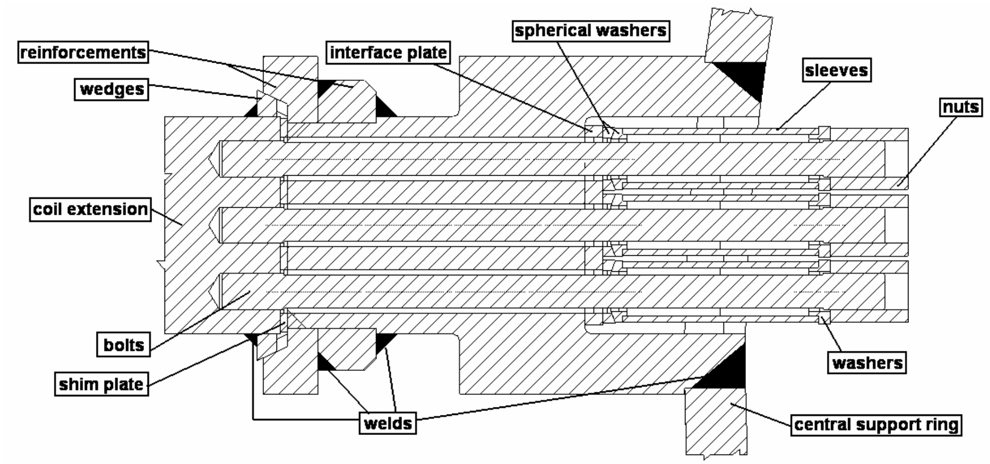


Design analyses of the support structure elements: nonlinear simulations including contact with friction, plasticity,assembly stresses, submodelling technique and using parametric models (14 bolted connections) . The work performed for Institute of Plasma Physics, Greifswald , Germany.

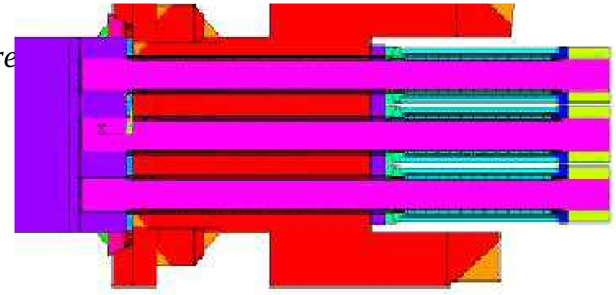
The results of the numerical simulation help to check the magnitudes of displacements and stresses for different loading scenarios and some modifications of the considered structures.

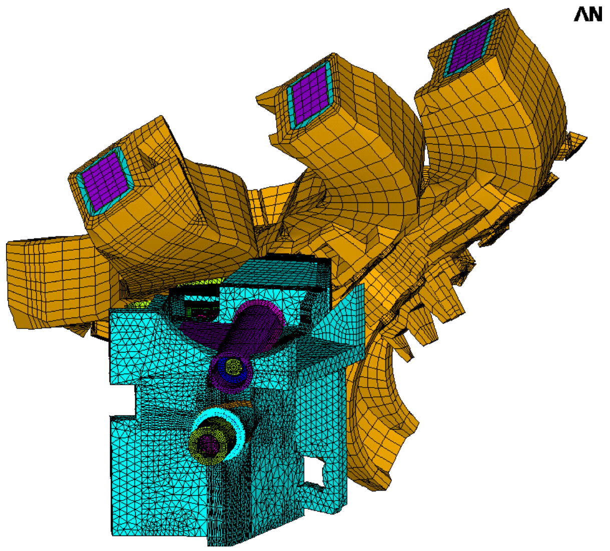
material 1 --- 1.4429 steel: shim, wedges, ring,

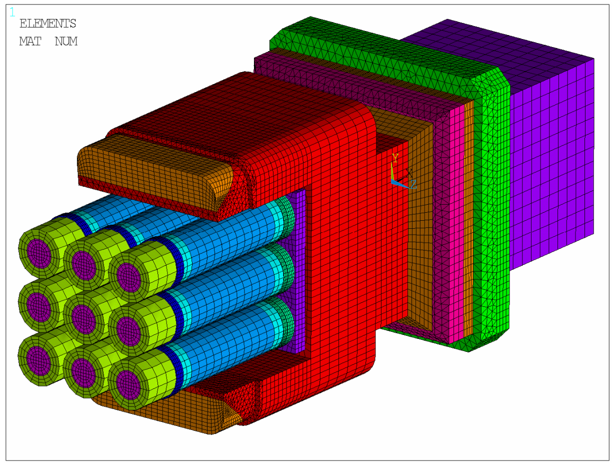




Stress-strain curve for material 1 (1.4429 steel) corresponding to the temperature
293 K (red graph)
77K (violet)
4K (blue)



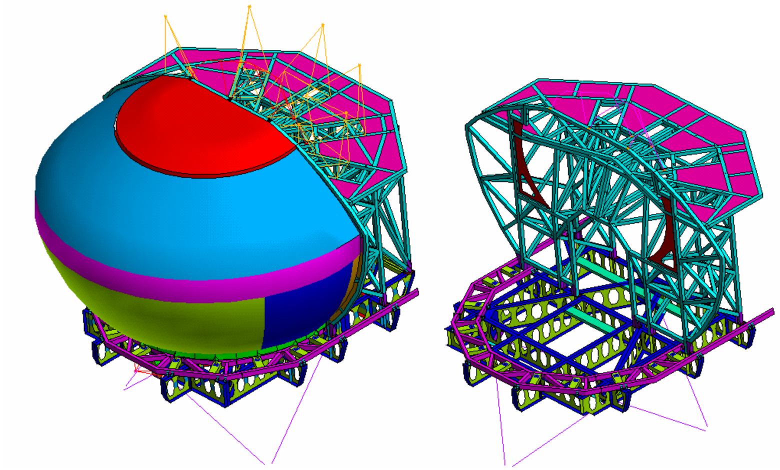




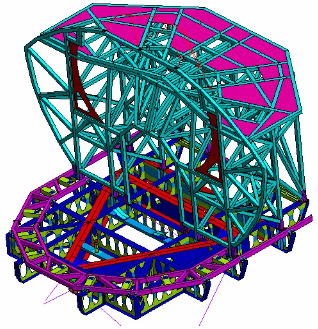
Structural analysis and design of the "KLAUDIA" flight simulator

The FE model of the initial platform design has showed the structure to be too flexibile. To find better solution the simplified FE model has been built, easy for modifications. The model has enabled quick verification of new concepts. The final detailed FE model has confirmed the improvement of the design. The fully nonlinear FE submodels have been built to check the stress level in the main joints. Vibration characteristics (natural frequencies and mode shapes) of the structure have been found

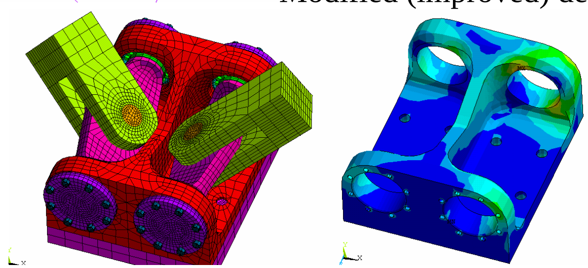
The FE model was built using shell, solid, beam, mass and link elements**.** The project was done for MP-PZL Aerospace Industries , Poland



Initial FE model



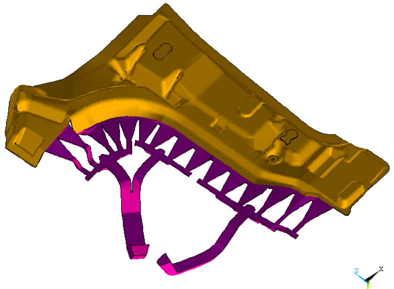
Modified (improved) design



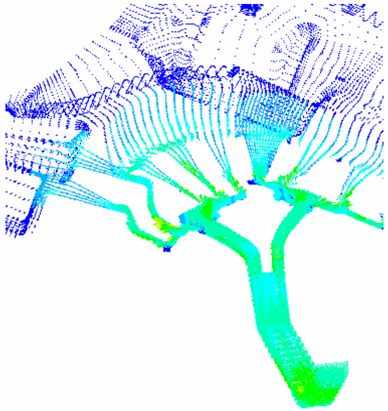
submodel of the joint

FE analysis of thin-walled elements' deformation during aluminium injection moulding

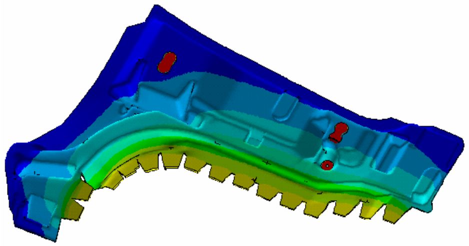
Numerical simulations have been performed to model the process of filling the mould by hot aluminium alloy. The analysis has enabled improvements of the element stiffness diminishing geometrical changes caused by the process. Fluid flow simulation with transient thermal analysis including phase change have been performed, followed by the structural elasto-plastic calculation of residual effects. The project performed for Alusuisse Technological Center, Sierre, Switzerland.

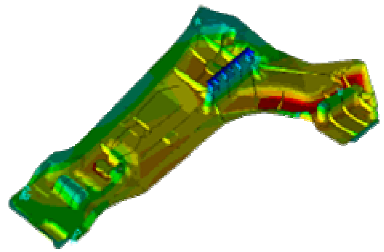


FE model

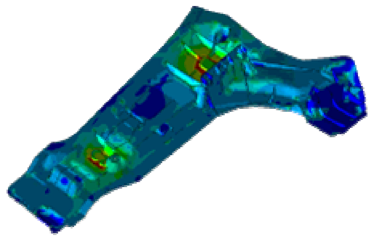


Velocity field during injection

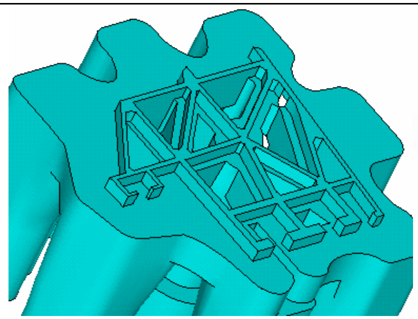


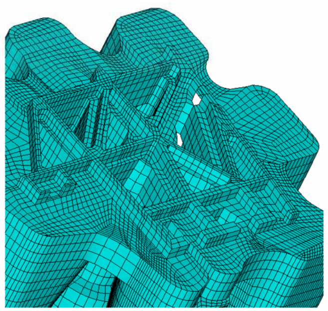


Temperature distribution (cooling effect) and displacements

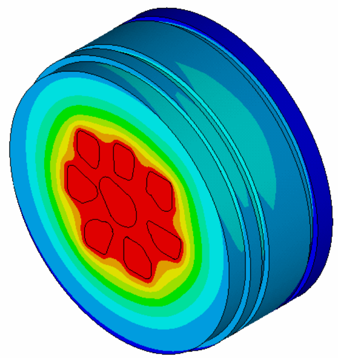


Residual stress distribution





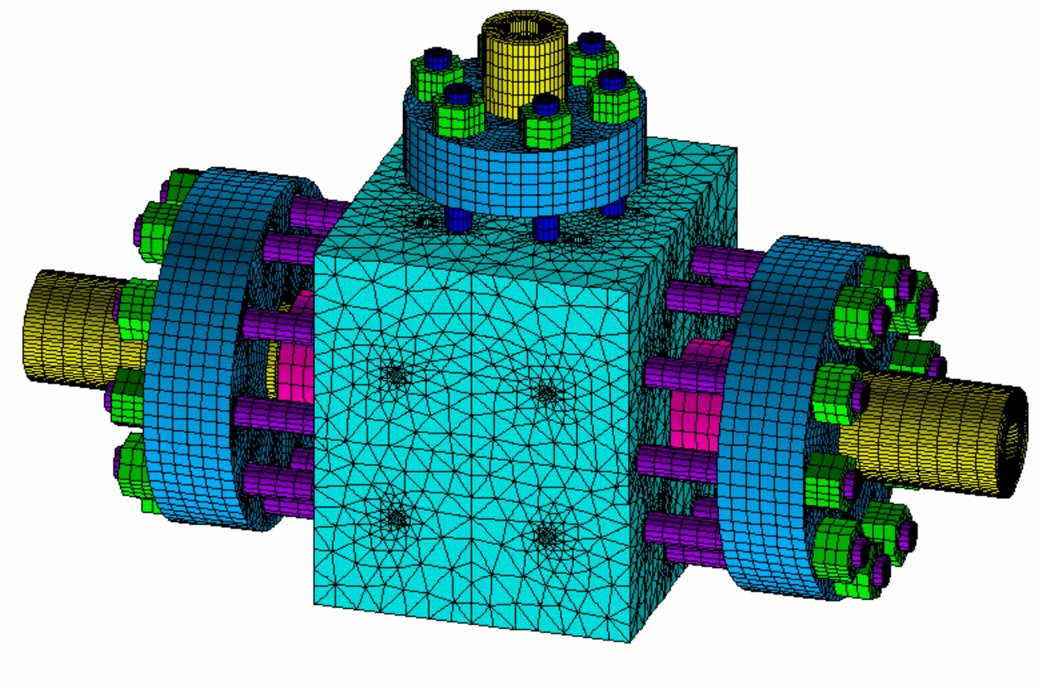
FE model of the die



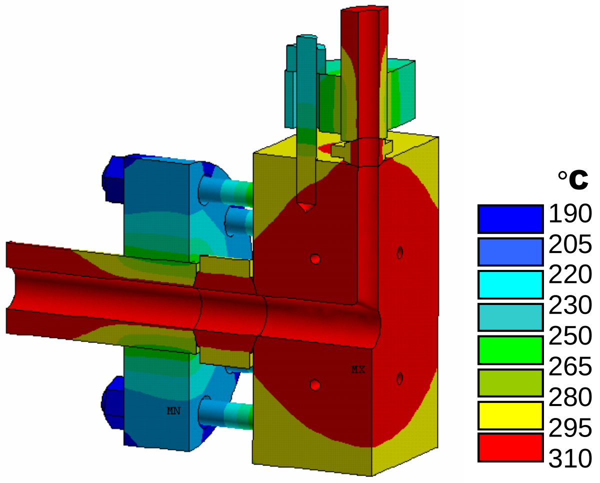
Velocity and temperature distribution

FE analysis of a high pressure T-connection

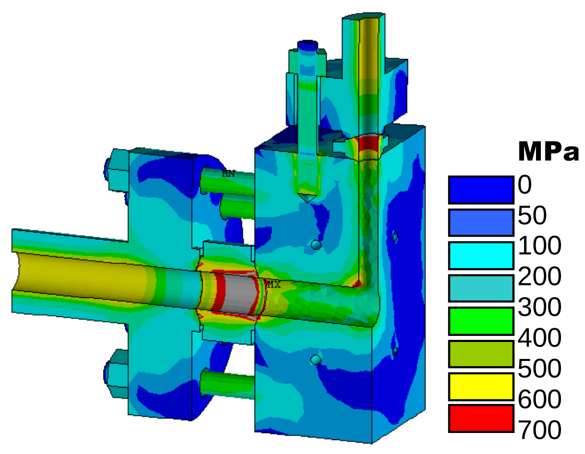
The aim of the analysis was to find out stress and strain distribution in a T-connection caused by high internal pressure (2600 at) and temperature gradients. External cooling, assembly procedure (screw pretension), contact and plasticity effects have been included. The project done for ORLEN petrochemical company



FE model



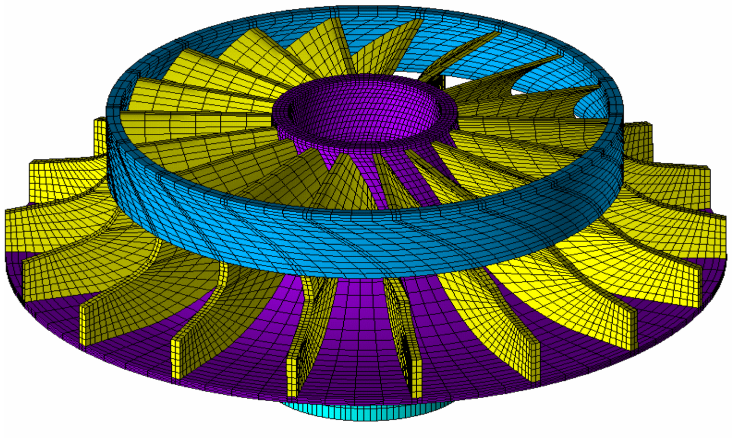
Temperature distribution



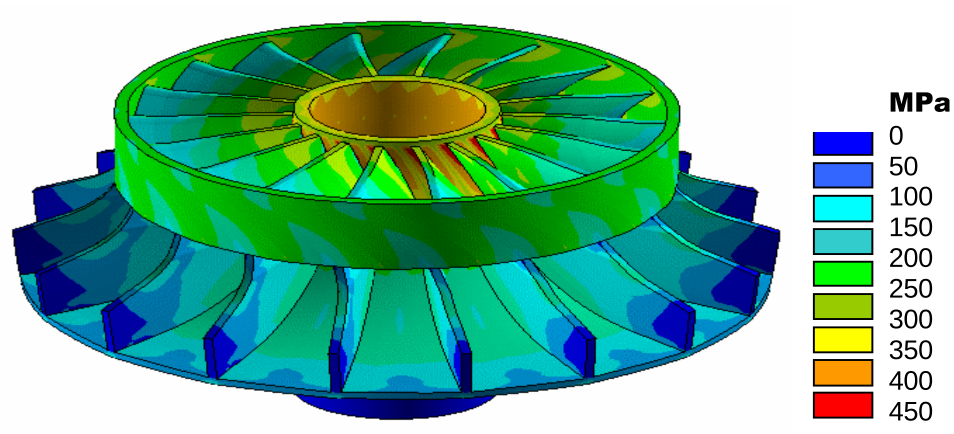
Von Mises stress

FE analyses of rotor disks

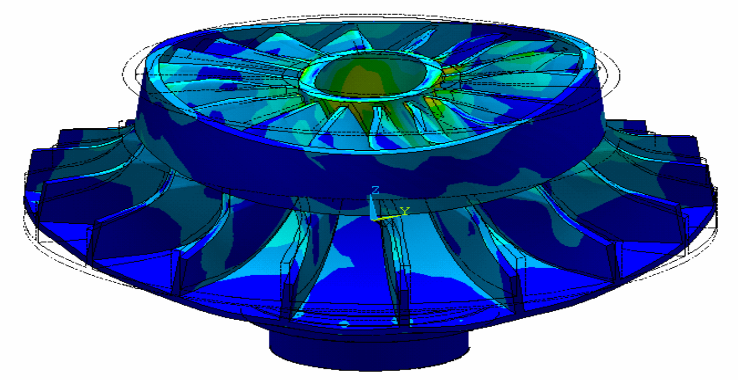
The aim of the analysis was to asses the right shape details of the rotor to avoid high stresses and to find its vibration characteristics.



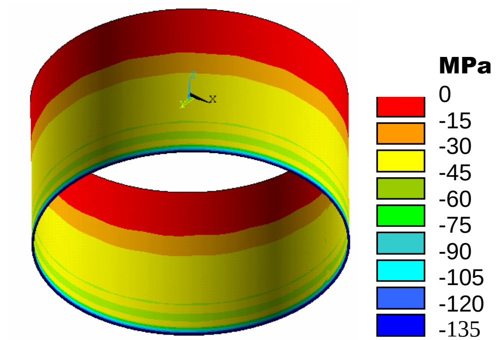
FE mesh



Von Mises stress distribution



The mode shape for the natural frequency of 2203Hz

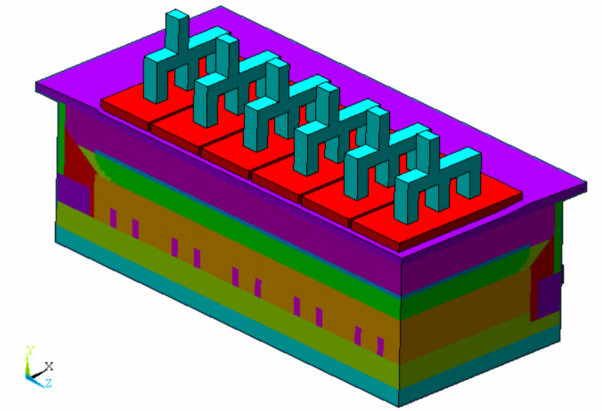


Contact pressure between the shaft and the rotor disk

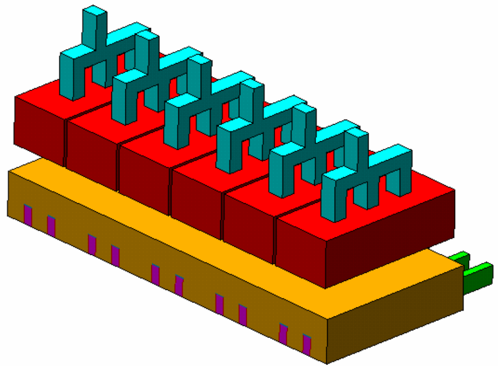
Thermo-electrical analysis of aluminium reduction cells

The analyses were performed to find temperature field and electrical potential distribution inside the reductant cell used in the process of aluminium production. The project done for Alusuisse Technological Center, Sierre, Switzerland.

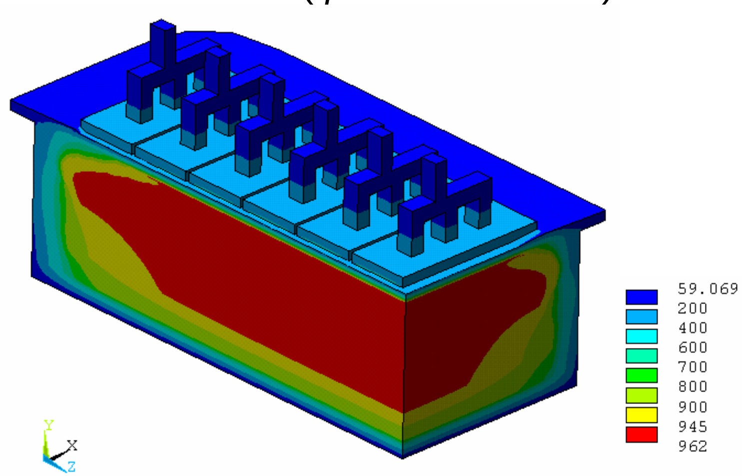
The influence of geometry, material properties and boundary conditions on the phenomena that take place in the bath and liquid aluminium is investigated. The analysis enabled to correct the design and to improve efficiency of the processes.



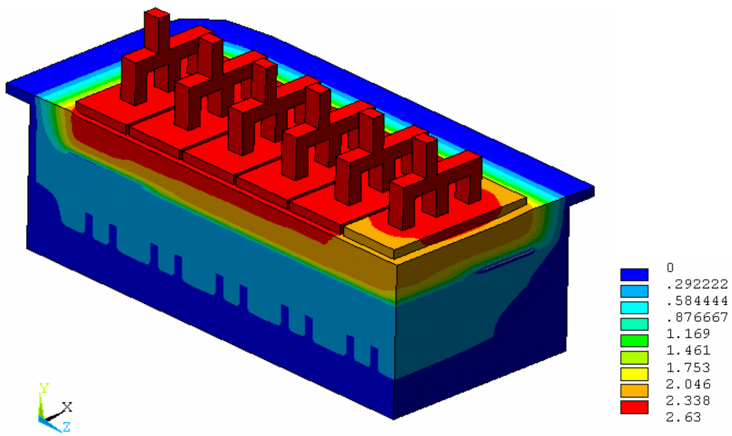
FE model (*quarter of the cell*)



Anode and cathode blocks



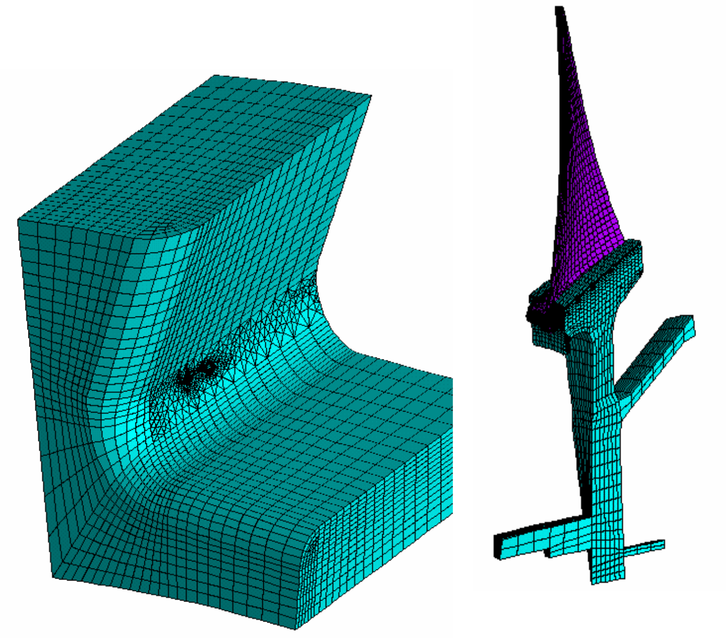
Temperature distribution



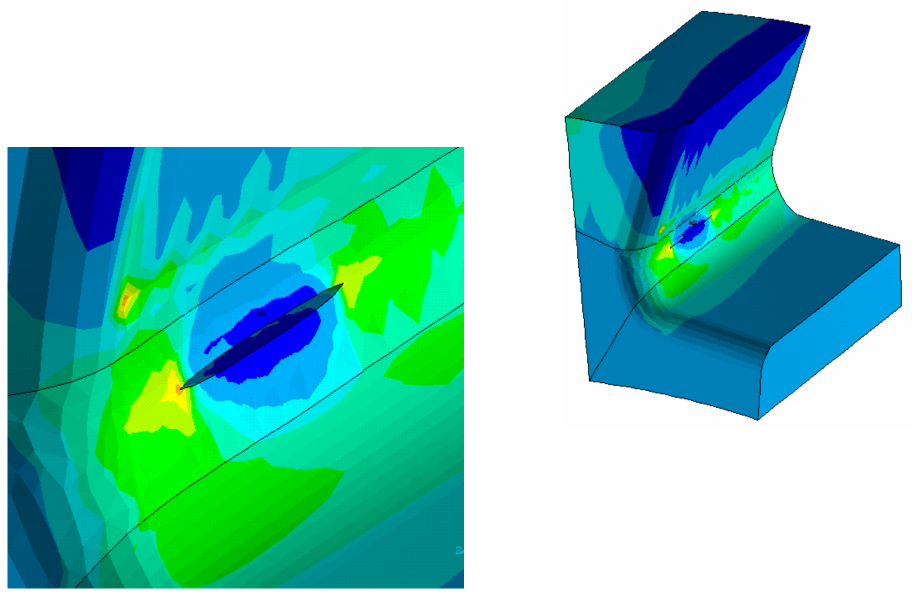
Electric potential distribution

FE analysis of the turbine blade locking part defects (imperfections)

Experiments show the presence of defects like surface scratch, or micro-crack in the region of blade locking part of the turbine disks. Such imperfections may result in crack initiation and propagation. A segment of the turbine disk together with a blade has been modelled (including contact). Half-elliptical crack has been introduced in the sub-model. Stress intensity factors and Rice integral values have been calculated.



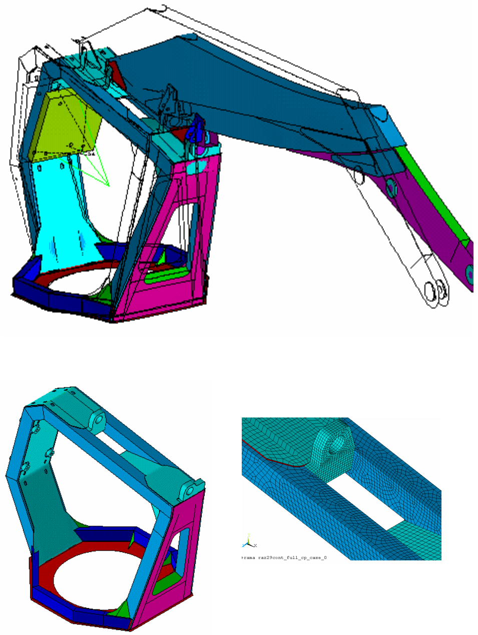
FE model



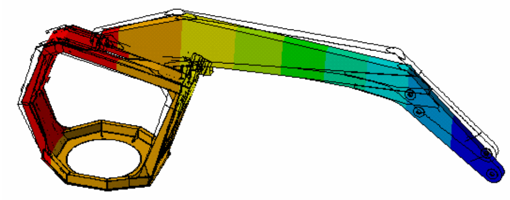
Von Mises stress distribution
in the vicinity of the crack

FEM Analysis of the Winch Frame and Boom of the snow groomer

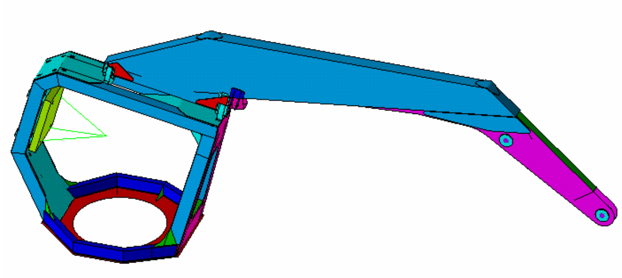
The aim of the analysis was to check the stiffness and stress level of the new design of the structure. Numerical model consisted of FE shell elements supplemented by brick, beam, link and mass elements. In regions of special care sub-models were used involving contact elements. The results suggested essential changes of design. The project done for PLUMETTAZ S.A., Bex, Switzerland



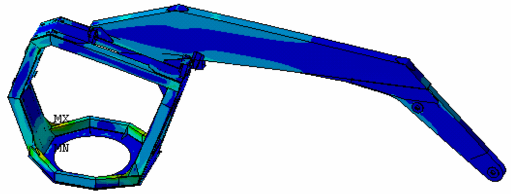
FE model



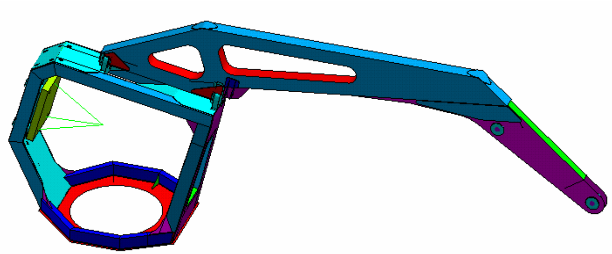
Displacements



Initial model

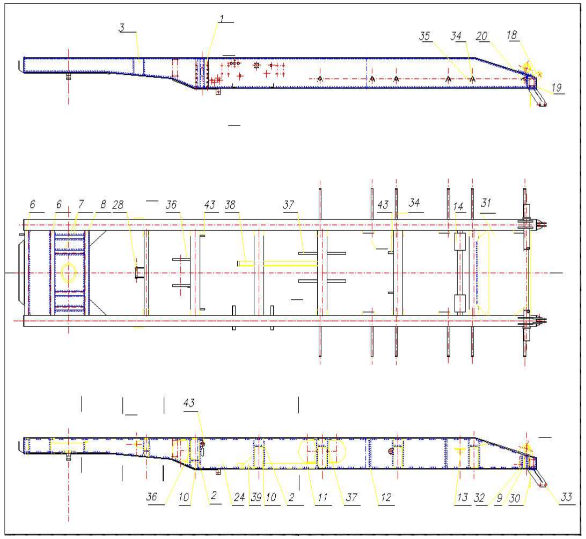


Von Mises stress

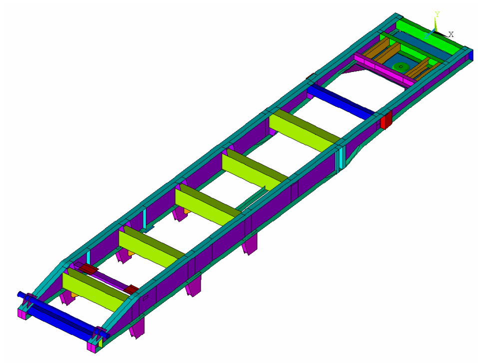


Modified design

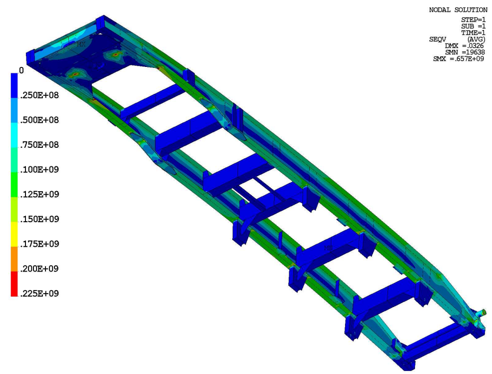
CAD/CAE study of a New Design of Truck Frame



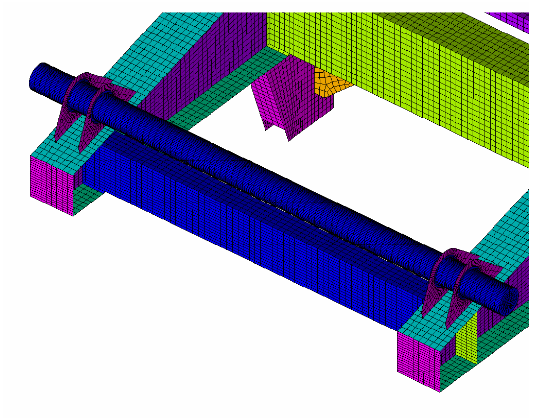
CAD project

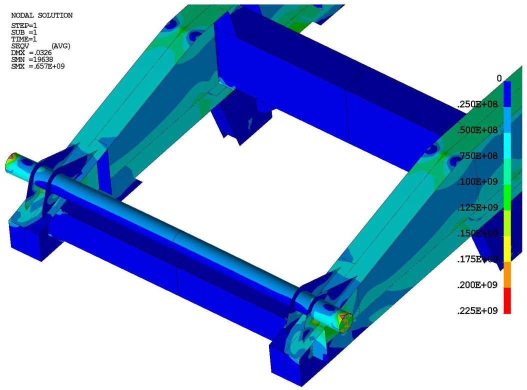


FE model



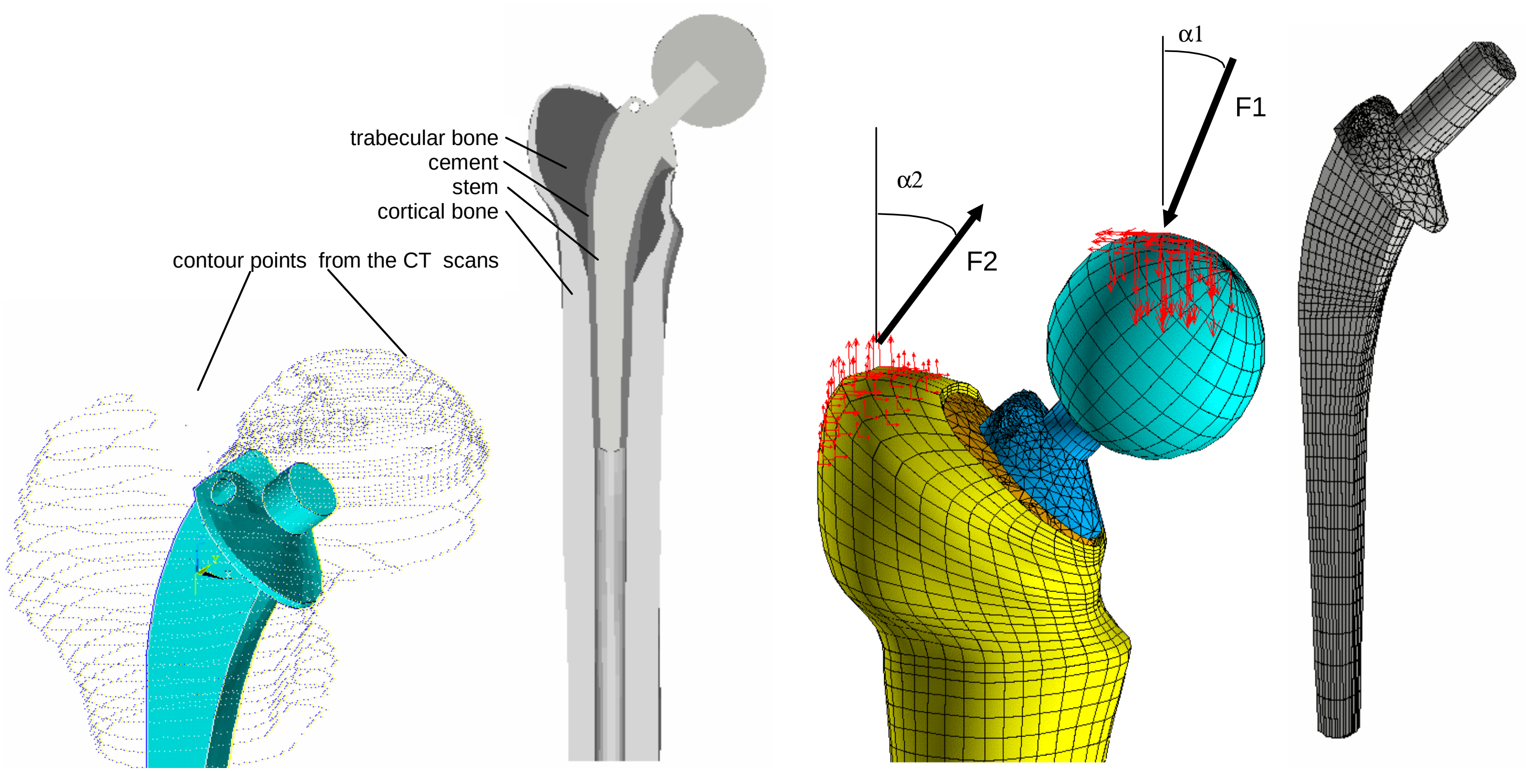
Von Mises stress



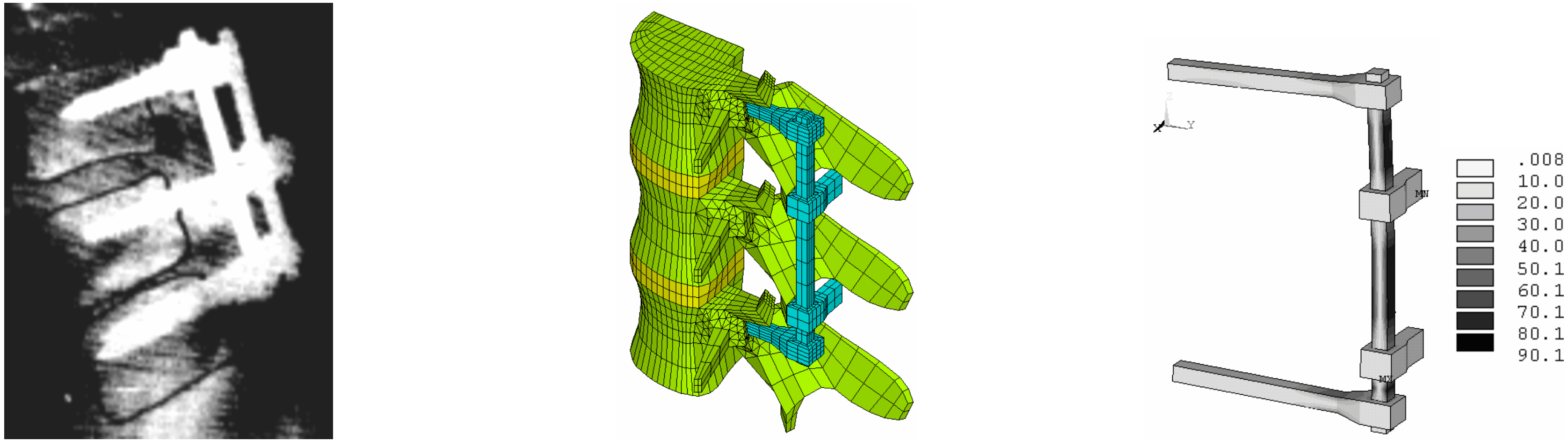


Finite element method in bone-implant system strength analysis

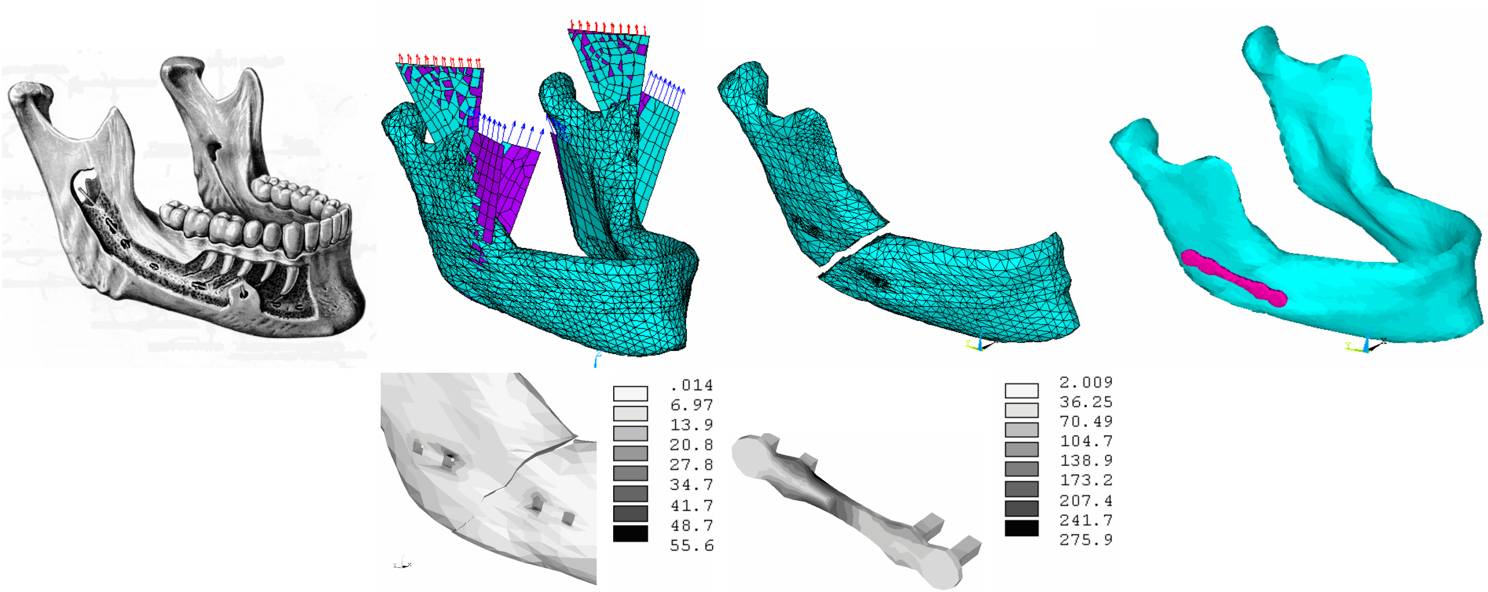
The three-dimensional FE models of the living tissues-implant systems can deliver the valuable information about mechanisms of stress transfer and healing processes after the orthopaedic surgery. In the presented example some different variants of the hip stem were considered to find the best solution, which should reduce stress concentration within the bone tissues. The model of the femur was built using the data obtained from CT scans. The considered load corresponds to one leg stance of a man weighting 800N.



Finite element model of the femur endoprosthesis *: body weight BW=800N, F1=2.47 BW, F2=1.55BW, α1=28°, α2= 40°*



FE model of the spine stabilizer and the von Mises stress distribution within the frame



Von Mises Equivalent stress (MPa)

FE model and selected results of numerical simulation of mandibular ostheosynthesis