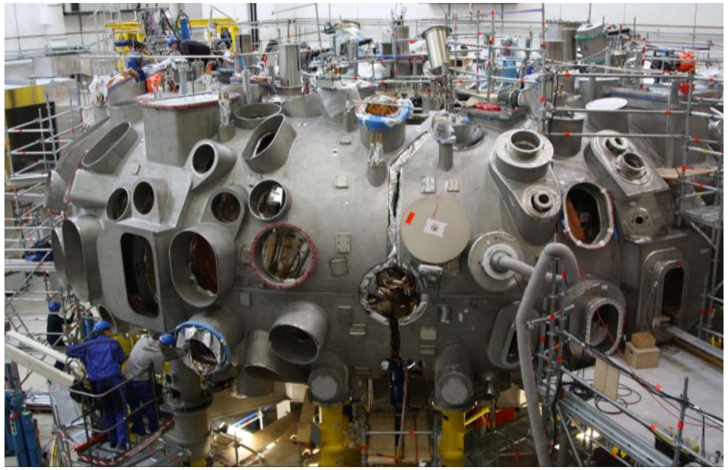
Finite Element Analysis of Critical Central Connection Elements of W7-X Stellator Coil Support System

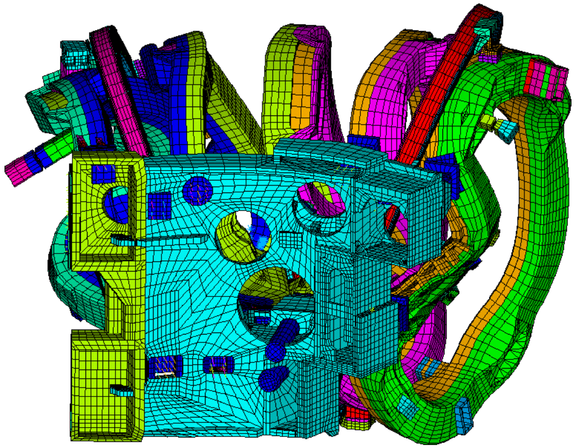
The objective of Wendelstein 7-X project is the stellator-type fusion reactor. In this device plasma channel is under control of magnetic field coming from magnet system of complex shape, made of 70 superconducting coils symmetrically arranged in 5 identical sections. Every coil is connected to central 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 forces and moments against each other and against the central support structure. Therefore, the detailed analysis of the coil to central support connections, the so called Central Support Elements (CSE), is a critical item for W7-X. Each NPC and PLC is fastened to the CSS by two central support elements (CSE).

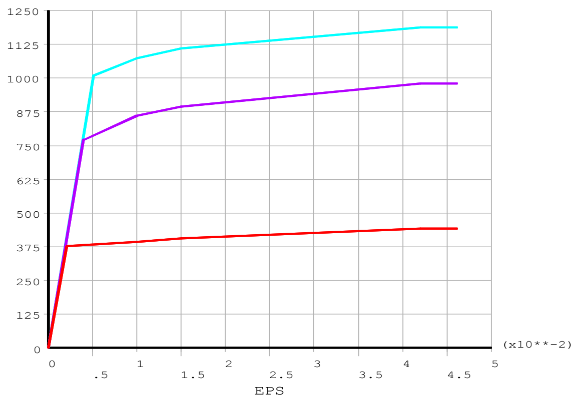
The aim of this work was to analyse mechanical behaviour of the bolted connections using detailed 3D finite element models (including bolts , washers, welds etc). The Global Model, analysed by Efremov Institute in Russia, provided information about the loads acting on the connections. All simulations were performed assuming elasto-plastic behaviour of the materials, assembly stresses and friction contacts between different parts of the connections. Stress distributions, displacements, forces acting on the bolts and welds were studied using standard and submodeling routines.

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.

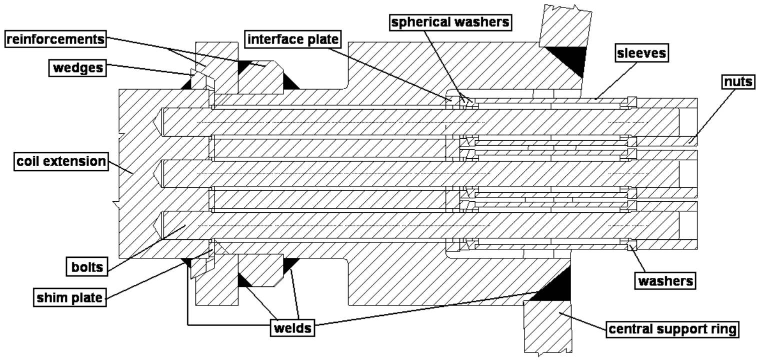


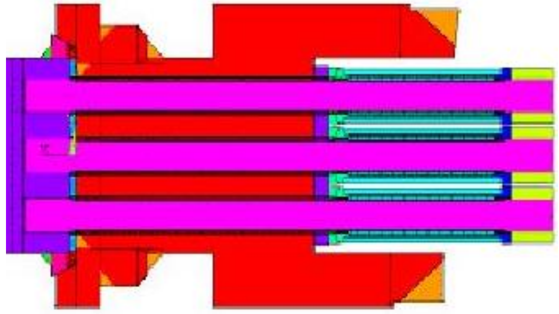


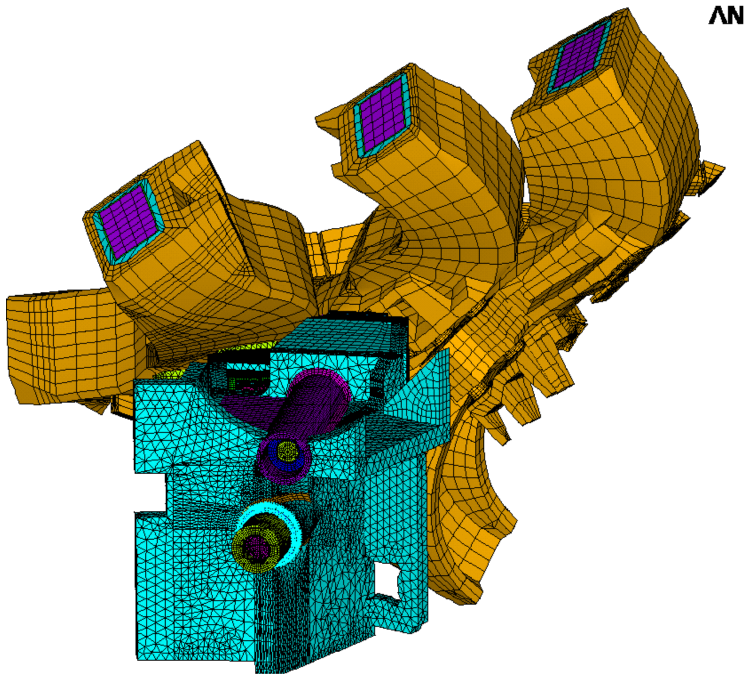
Design analyses of the support structure elements: nonlinear simulations including contact with friction, plasticity, submodelling technique using parametric models (14 bolted connections). ***The work performed for Institute of Plasma Physics, Greifswald , Germany***.

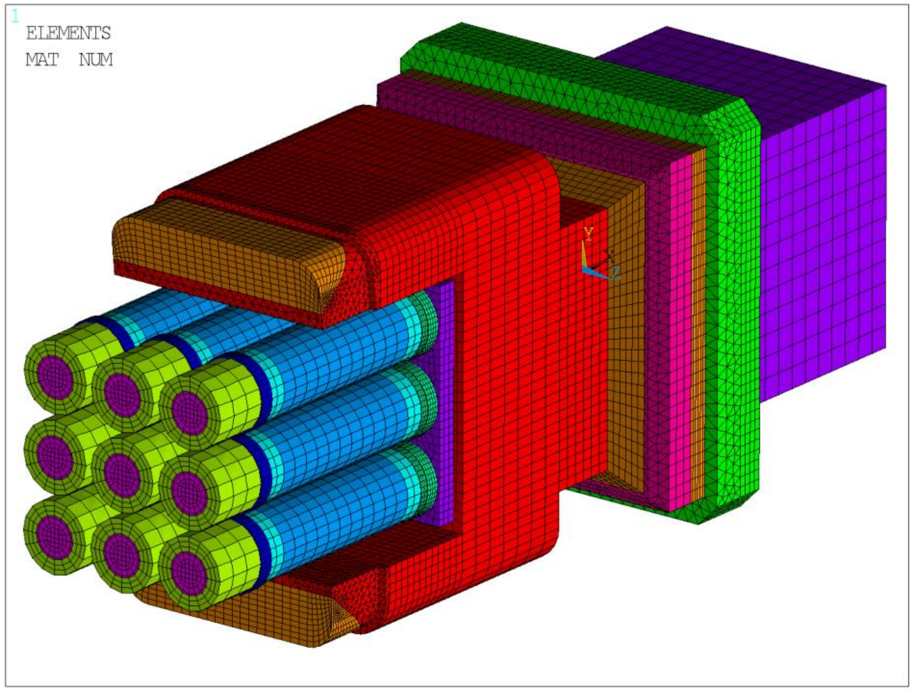


Stress-strain curve for material 1 (1.4429 steel)- shim, wedges, ring



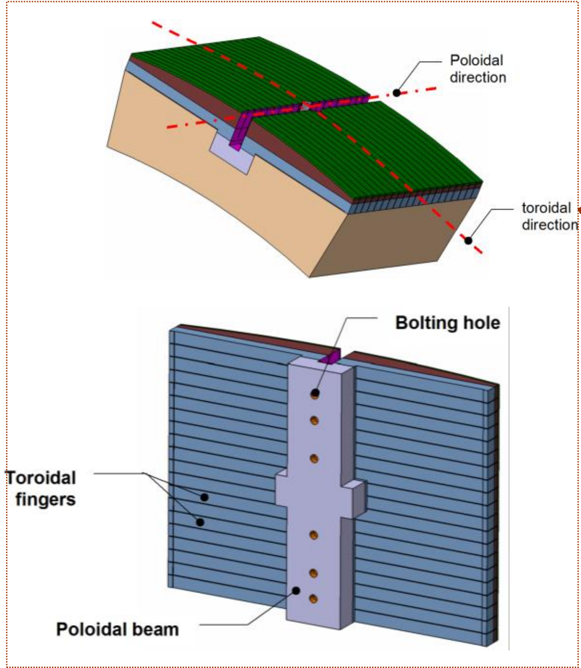




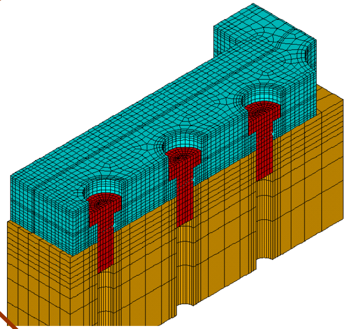


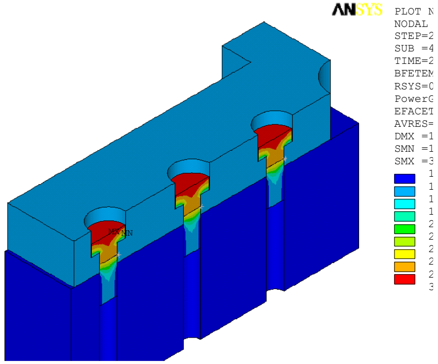
FE modelling of the mechanical behaviour of separable first wall elements for ITER (2008)

Thermo-structural analyses of the First Wall (FW) protection panel (on the shield module) has been solved ***in cooperation with IPP, Garching, Germany***. The aim of the analyses was to find the optimum design parameters of the first wall panel components.

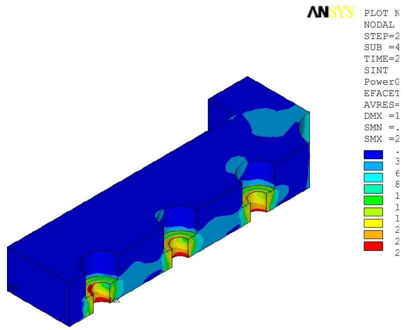


TASK 1 (Poloidal beam)



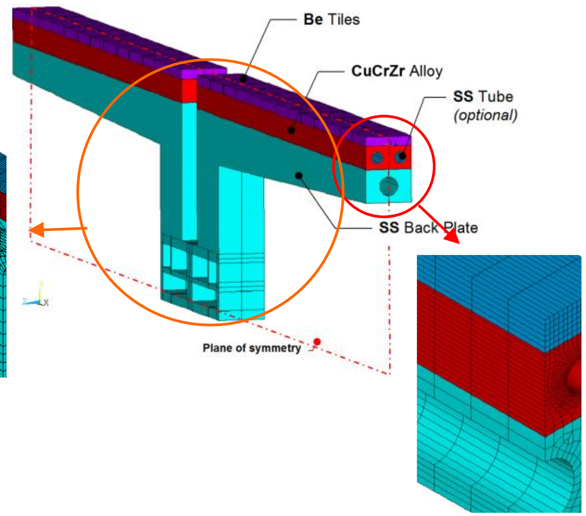


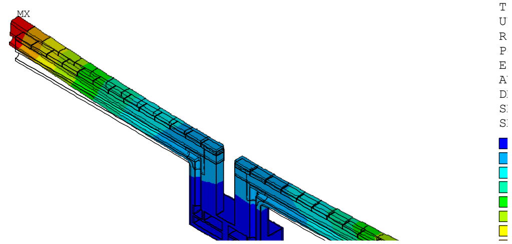
Temperature distribution



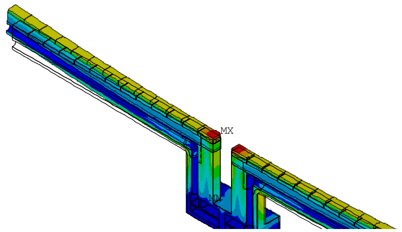
Stress intensity

TASK 2 (toroidal finger)

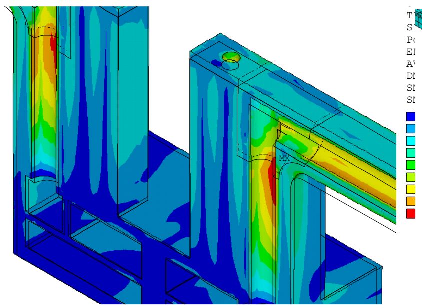


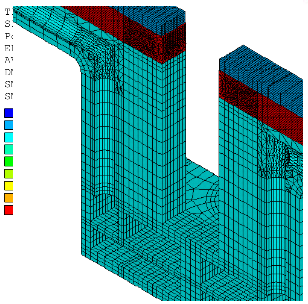


Vertical displacements



Temperature distribution

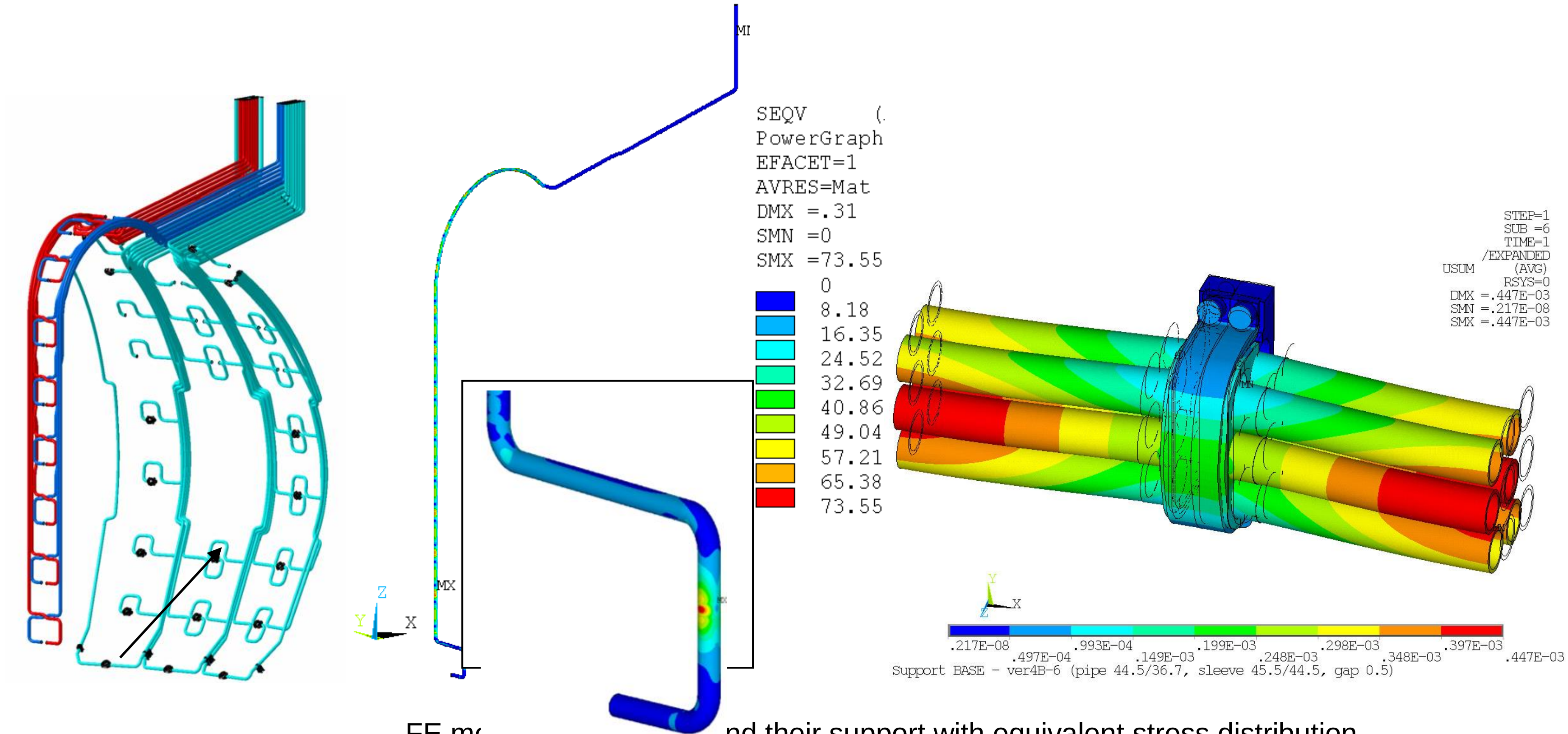




Stress intensity

Preliminary mechanical analysis of blanket manifold concept for ITER reactor

The aim of this work was to perform the 3-D finite element stress analysis of the manifold concept proposal (pipe option)
The work included the following analyses:
- Calculation of the displacements and stresses in the pipes caused by EM loads, pressure, temperature and assembly stresses
- Parametric analysis of the stresses in the supports corresponding to different design assumptions.



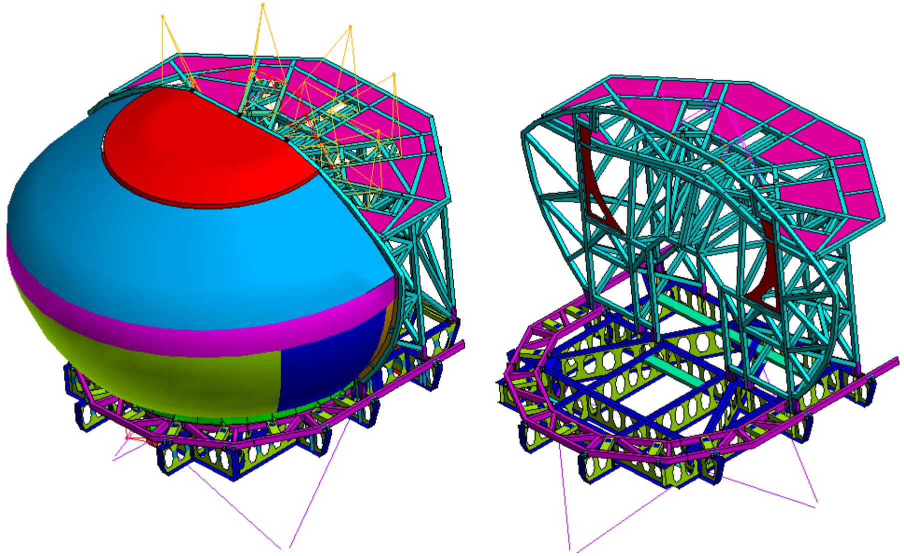
FE models of the pipes and their support with equivalent stress distribution

Structural analysis and design of the "KLAUDIA" flight simulator

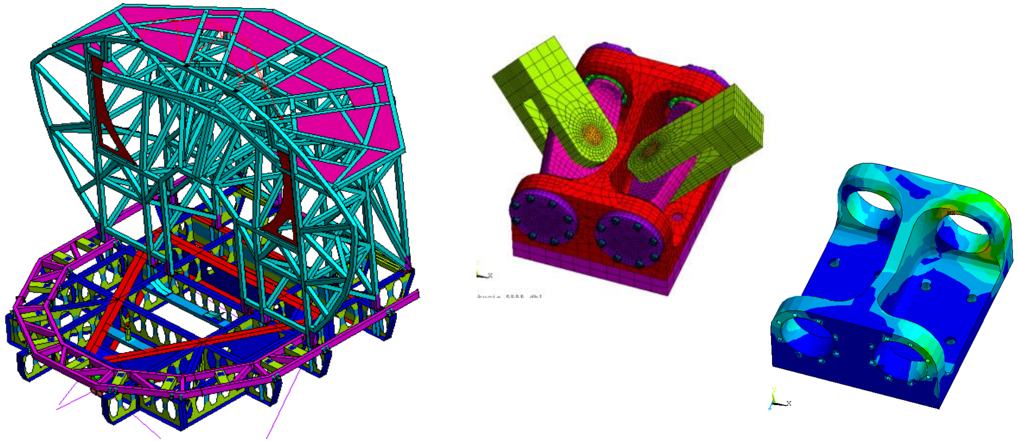
The FE model of the initial platform design has showed the structure to be too flexibile. To find better solution the simplified FE model has been built, easy for modifications. The model has enabled quick verification of new concepts. The final detailed FE model has confirmed the improvement of the design. The fully nonlinear FE submodel has 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



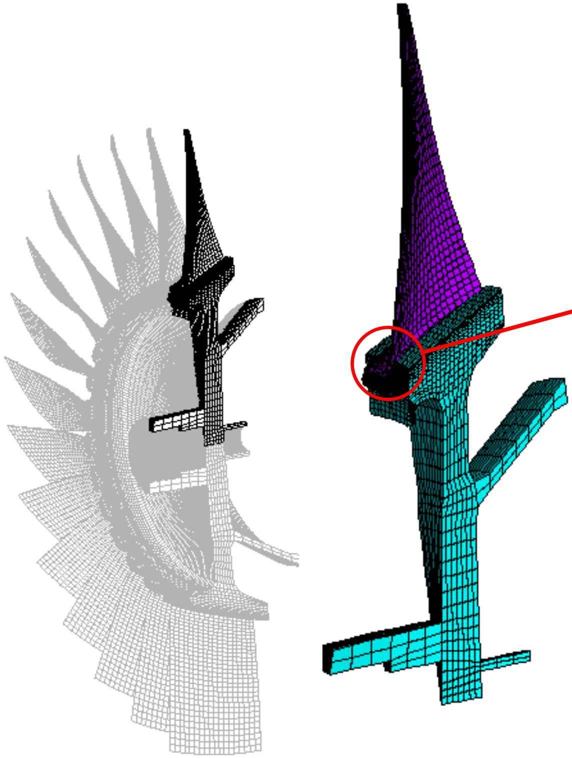
Initial FE model



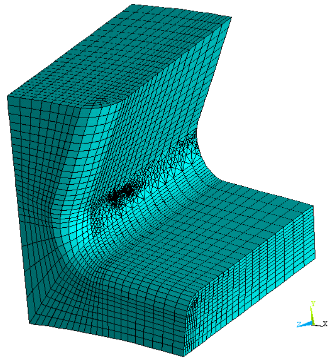
modified structure and submodel of the joint

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

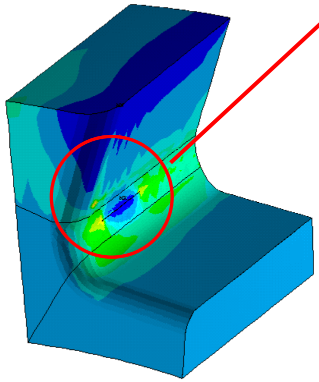
Experiments show the presence of defects like surface scratch, or micro-crack in the region of blade locking piece of the turbine disks. Such imperfections may result in crack initiation and propagation. A segment of the turbine disk together with a blade has been modelled (including contact). Half-elliptical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



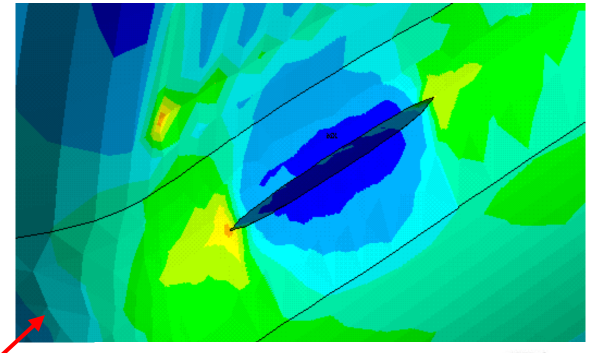
FE global model

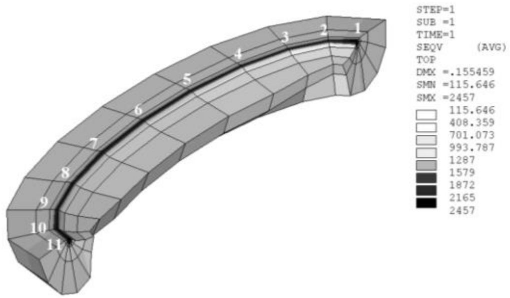


submodel of the vicinity of the crack



Von Mises stress distribution in the vicinity of the crack





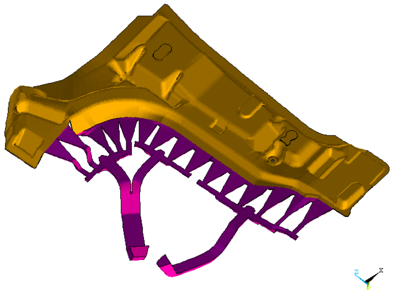
submodel of the crack

Stress intensivity factors (*N/mm*$^{3/2}$) along the front of the crack

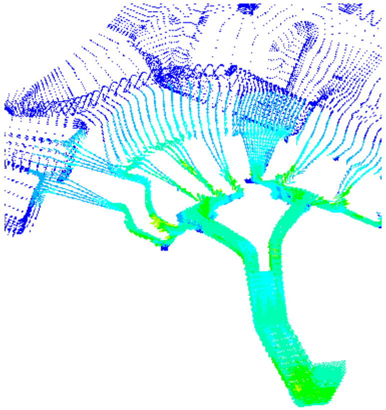
Pkt.	1	2	3	4	5	6	7	8	9	10	11
K_I	410	421	442	453	465	472	476	462	455	455	450
K_{II}	81	59	38	8	12	28	44	50	52	66	76
K_{III}	30	42	64	73	78	78	72	62	48	29	25

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

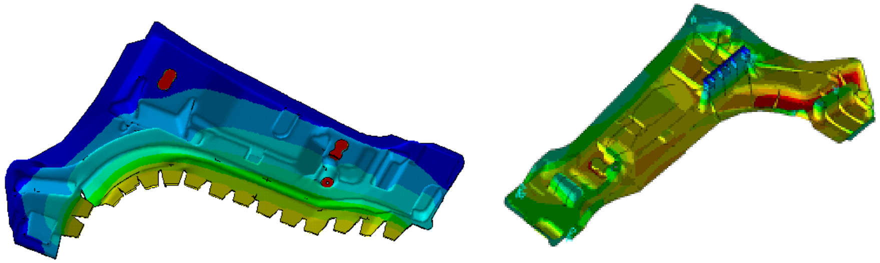
Numerical simulations have been performed to model the process of filling the mould by hot aluminium alloy. The analysis has enabled improvements of the element stiffness diminishing geometrical changes caused by the process. Fluid flow simulation with transient thermal analysis including phase change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



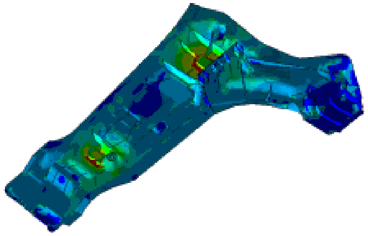
FE model



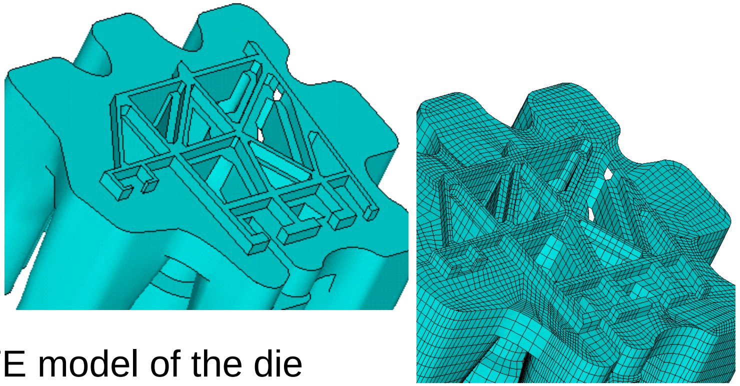
Velocity field during injection



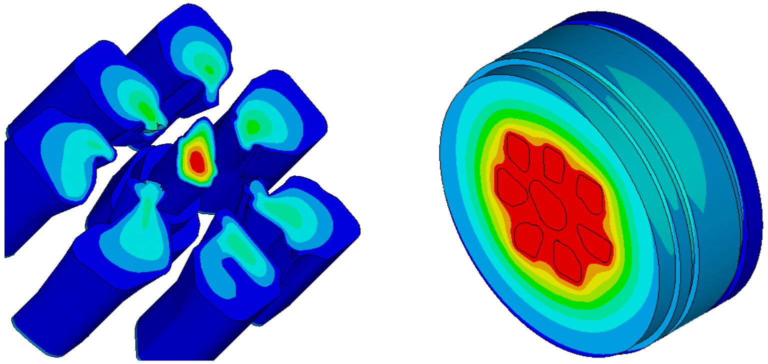
Temperature distribution (cooling effect) and displacements



Residual stress distribution



FE model of the die

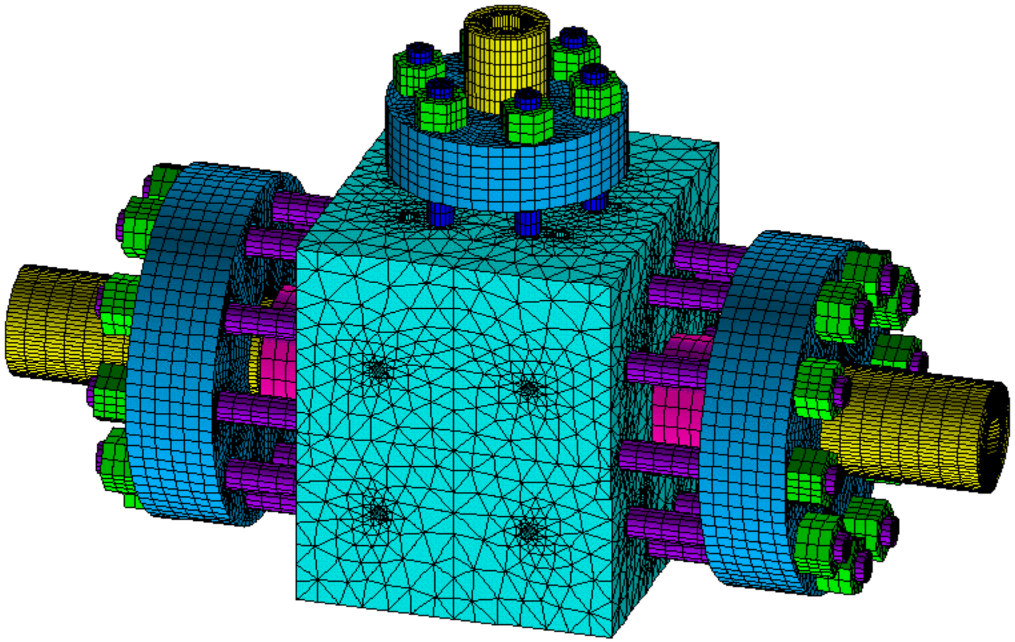


Velocity and temperature distribution

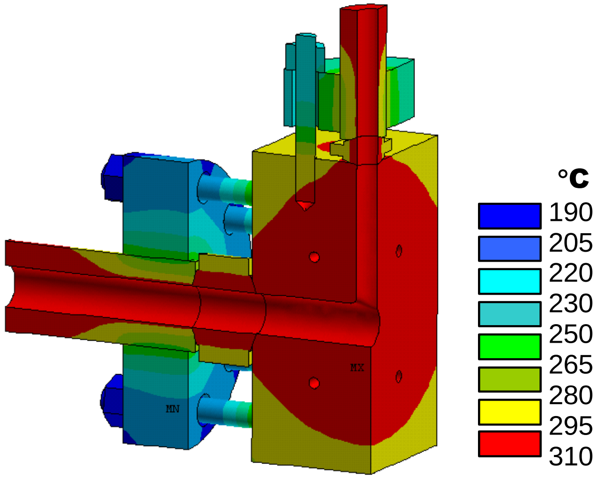
FE analysis of a high pressure T-connection

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.

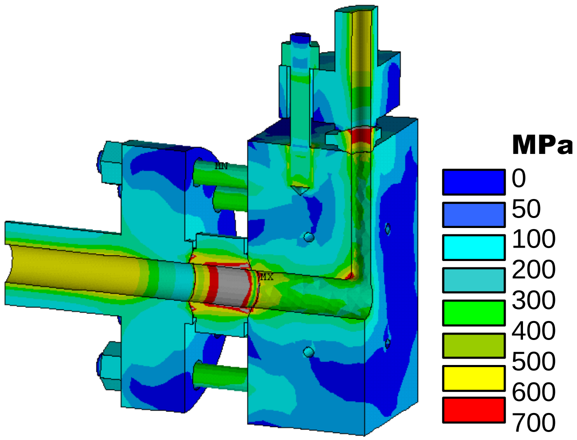
The project done for ORLEN petrochemical company



FE model



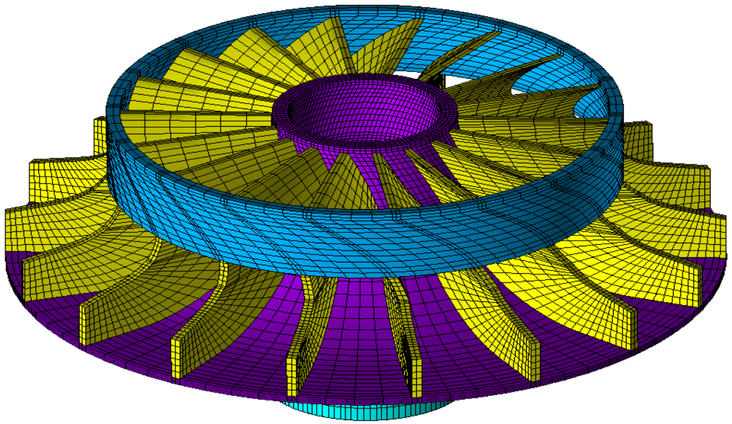
Temperature distribution



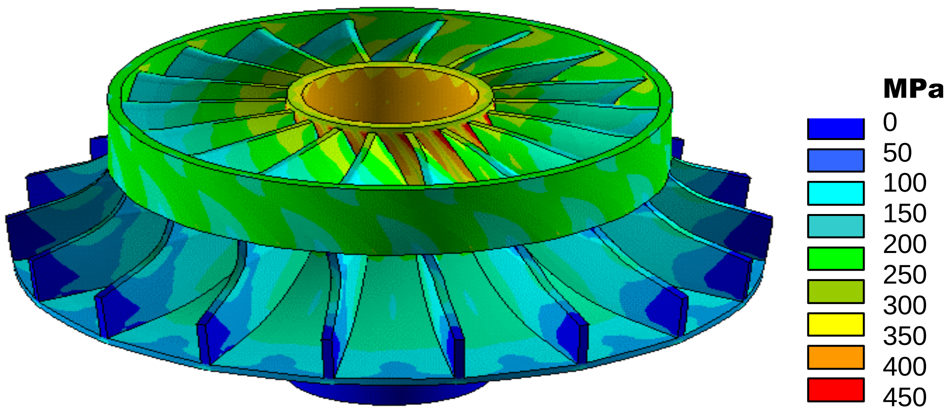
Von Mises stress

FE analyses of rotor disks

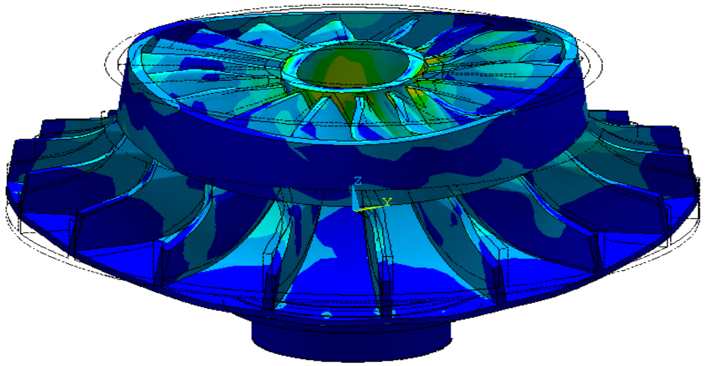
The aim of the analysis was to asses the right shape details of the rotor to avoid high stresses and to find its vibration characteristics.



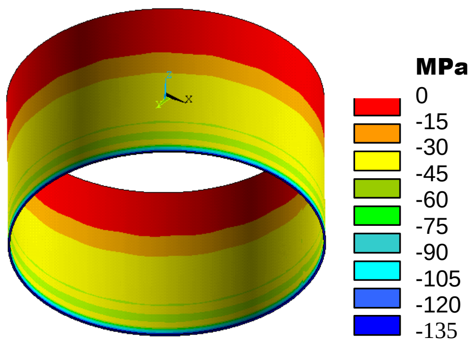
FE mesh



Von Mises stress distribution



The mode shape for the natural frequency of 2203Hz

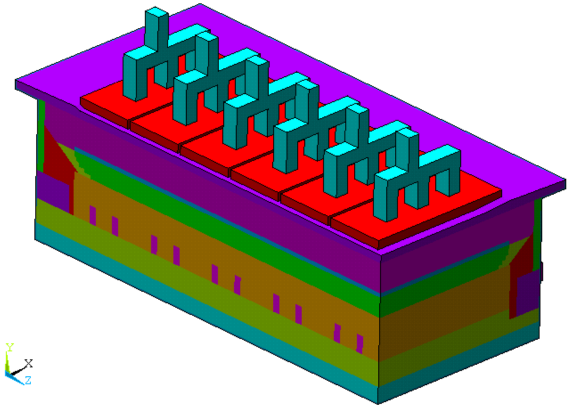


Contact pressure between the shaft and the rotor disk

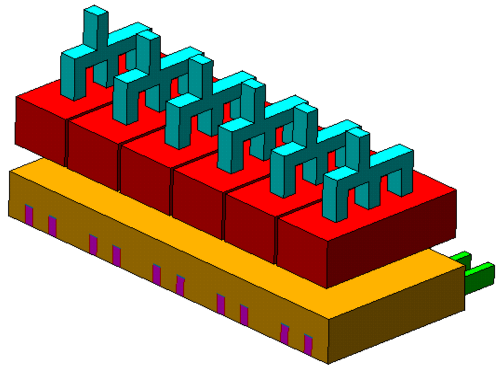
Thermo-electrical analysis of aluminium reduction cells

The analyses are 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.***

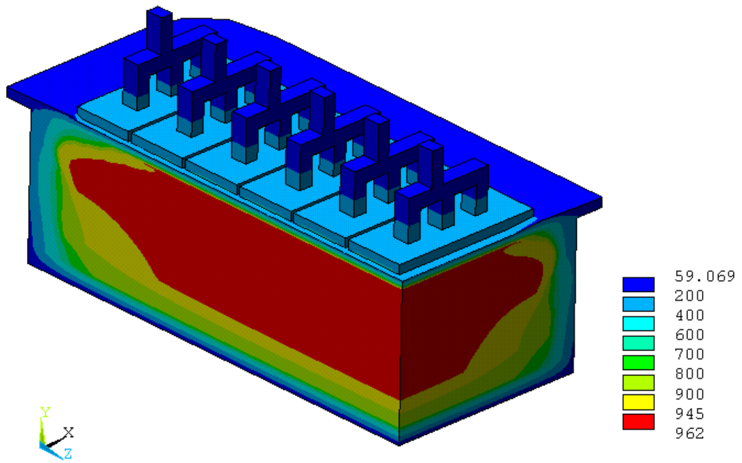
The influence of geometry, material properties and boundary conditions on the phenomena that take place in the bath and liquid aluminium is investigated. The analysis enabled to correct the design and to improve efficiency of the processes.



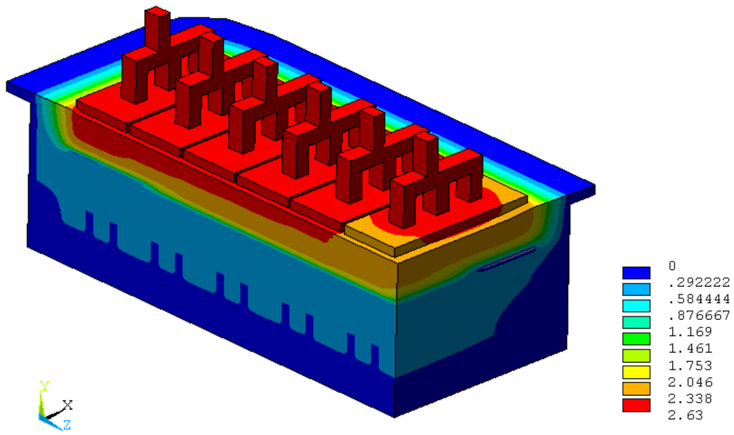
FE model (*quarter of the cell*)



Anode and cathode blocks



Temperature distribution

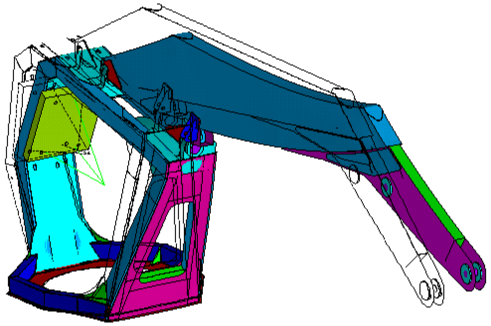


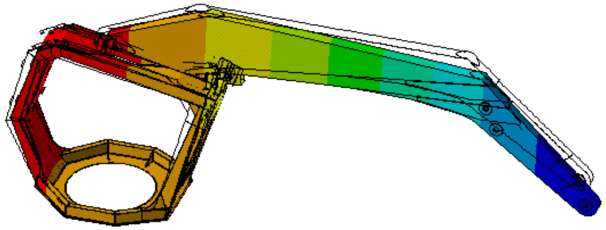
Electric potential distribution

FEM Analysis of the Winch Frame and Boom

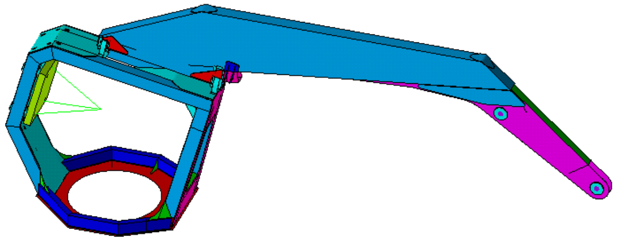
The aim of the analysis was to check the stiffness and stress level of the new design of the structure. Numerical model consisted of FE shell elements supplemented by brick, beam, link and mass elements. In regions of special care sub-models were used involving contact elements. The results suggested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

The project done for PLUMETTAZ S.A., Bex, Switzerland

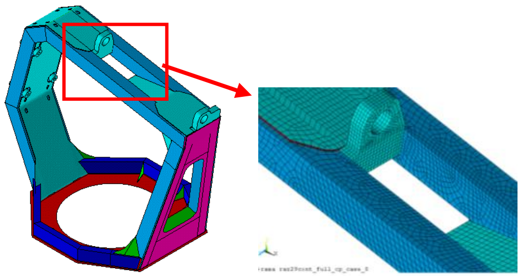




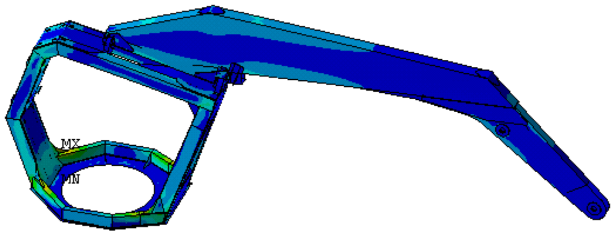
Displacements



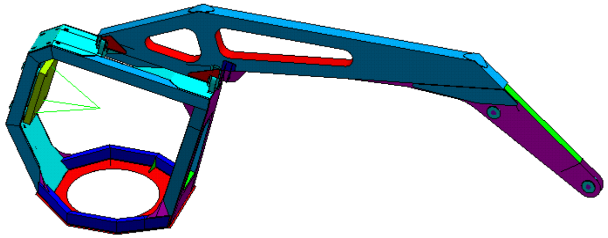
Initial model



FE model

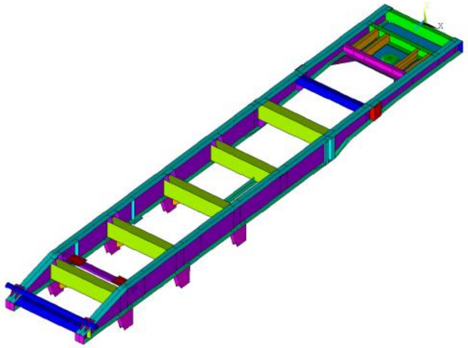


Von Mises stress

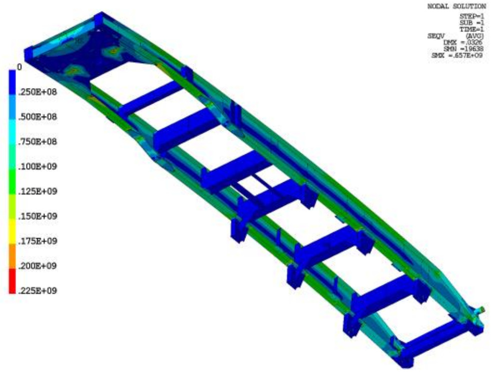


Modified design

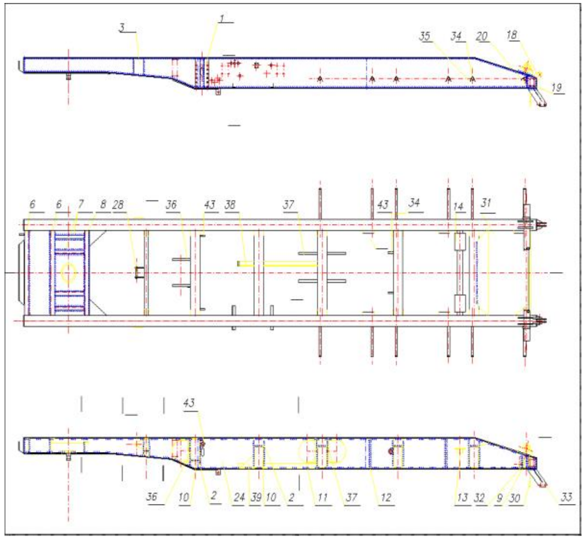
CAD/CAE study of a New Design of Truck Frame



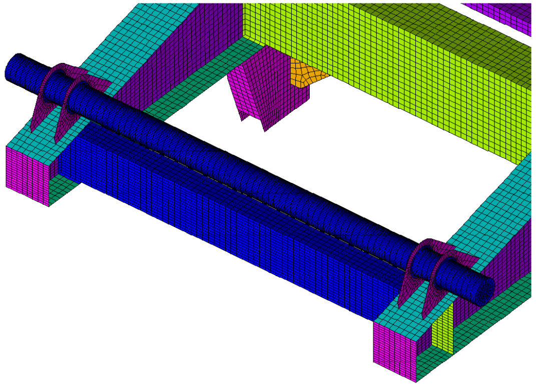
FE model

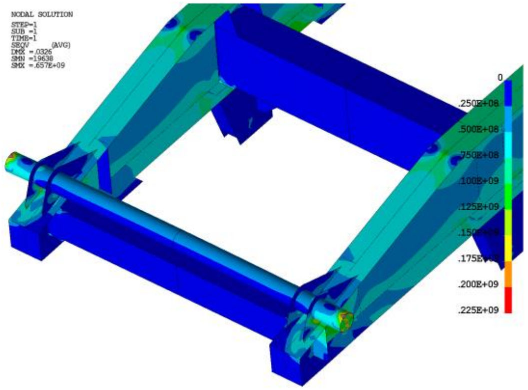


Von Mises stress



CAD project





Stress and strain analysis of the composite wing structure of PW-5 glider

FEDERATION AERONAUTIQUE INTERNATIONALE

FAI

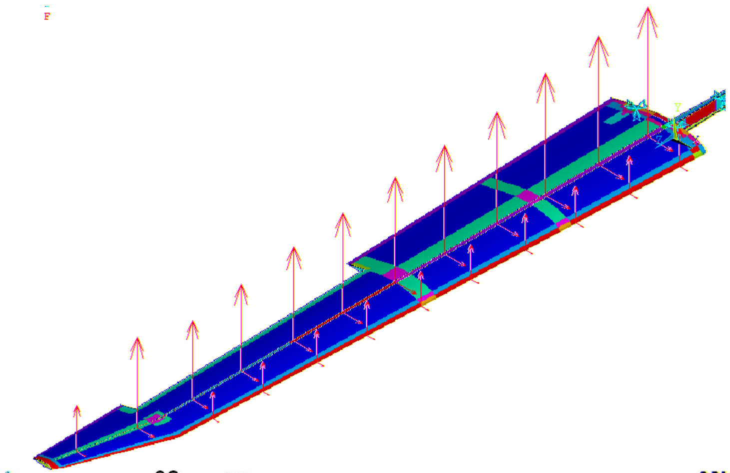
Diplôme d'Honneur

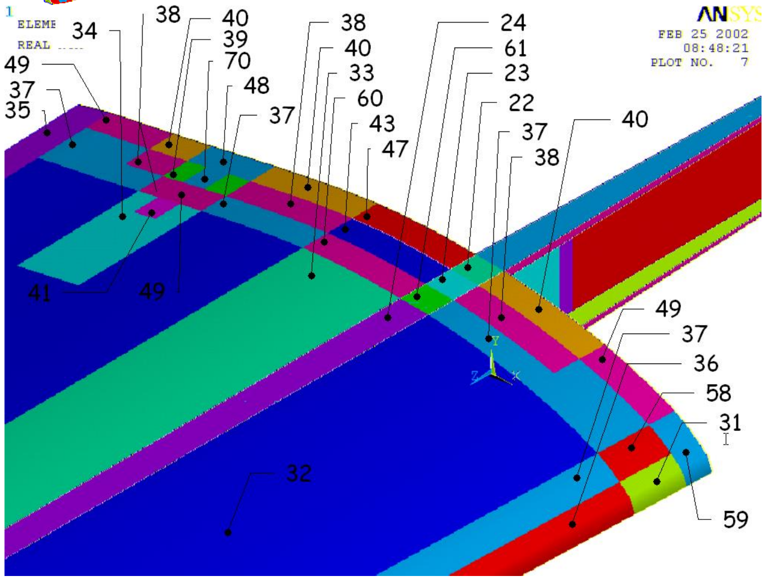
This is to certify that the PW-5 "Smyk" designed by the Warsaw University of Technology is the winner of the FAI's World Class Glider Competition. The PW-5 is therefore hereby declared FAI WORLD CLASS GLIDER.

Paris, le October 25, 1994

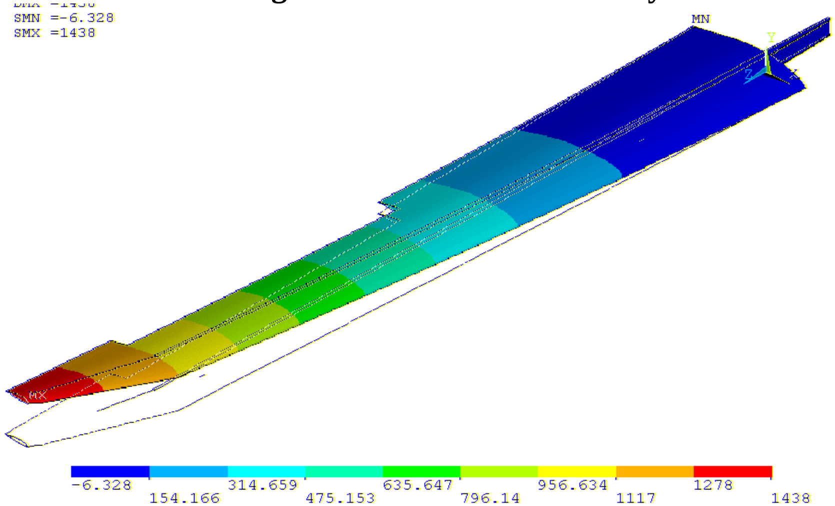
Le Secrétaire Général de la FAI

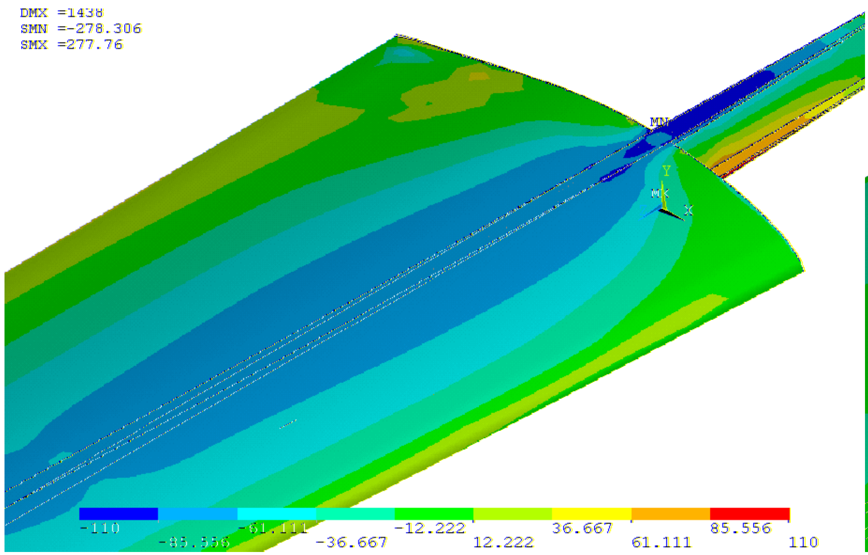
Le Président de la FAI

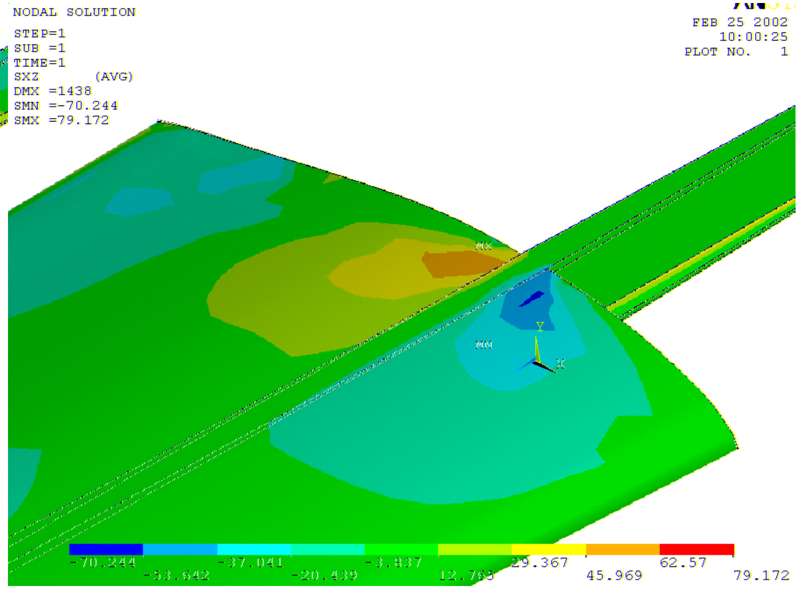




Deflection of the wing structure under the aerodynamical loads





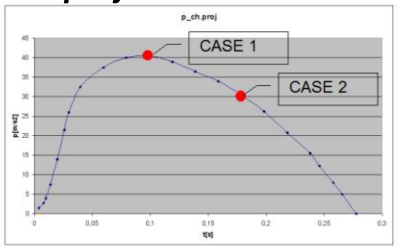


Stress tensor components : σ_z bending stress distribution **and shear stress distribution**

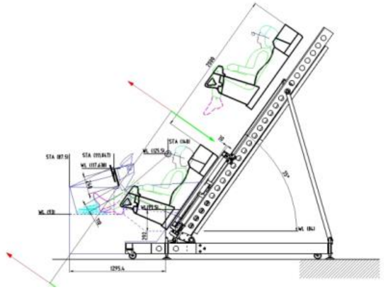
FE analysis of the EPT device (2005)

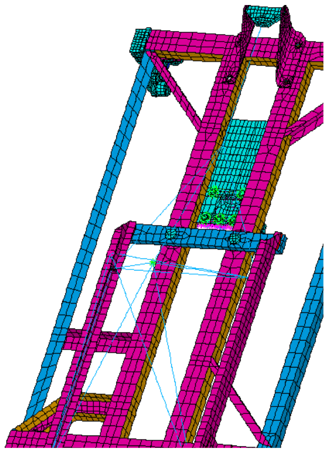
The FE model of the EPT device has been investigated. Two separate FE models have been prepared: FE model of the sled complete and FE model of the base frame with launching guide rails. The FE models have been built using shell, solid, beam, mass and link elements.

The project was done for MP-PZL Aerospace Industries , Poland

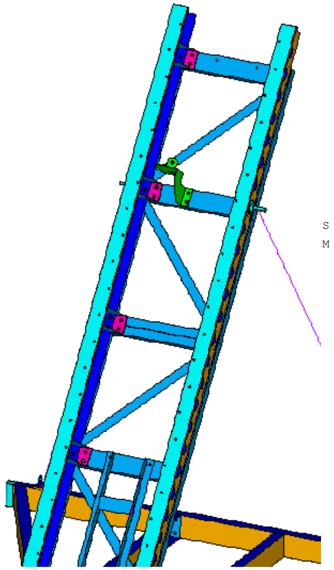


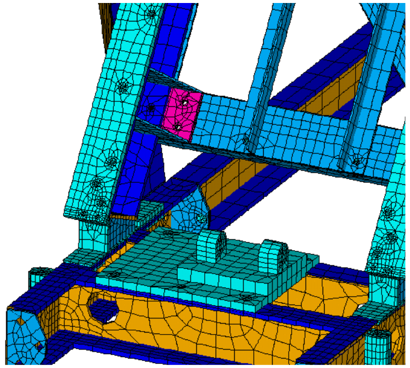
Acceleration characteristics.



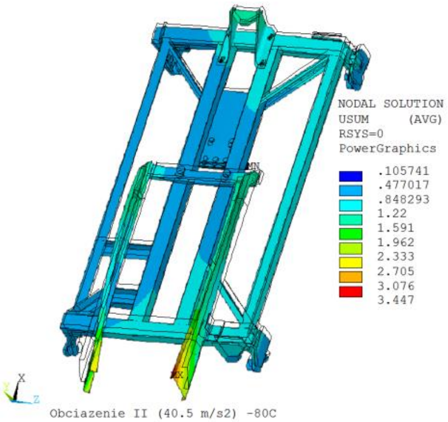


FE model of the sled complete

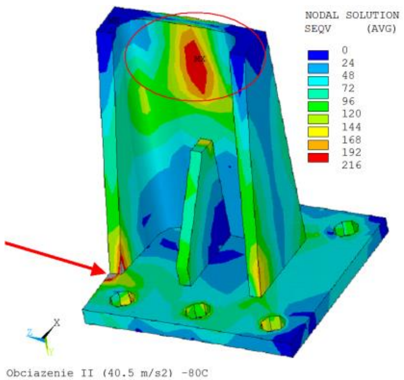




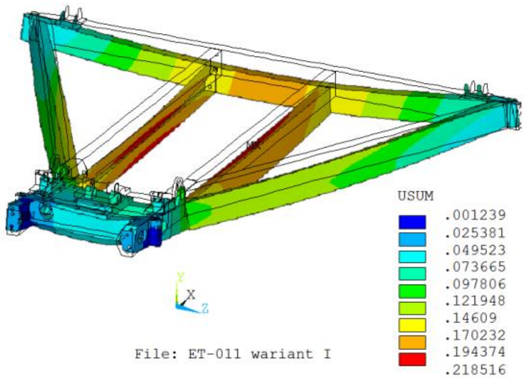
FE model of the base frame with launching guide rails



Total displacements in the sled



Von Mises stress in the the actuator hook up

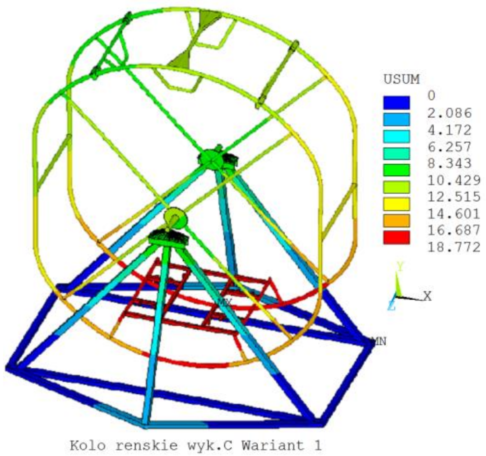


Total displacements in the base frame

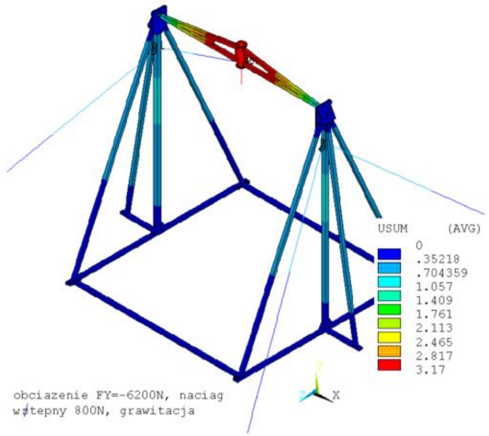
Training Devices for pilots (2009)

The FE models of the training devices for pilots have been analyzed. Several FE models have been studied. The FE models have been built using shell, solid, beam, mass and link elements.

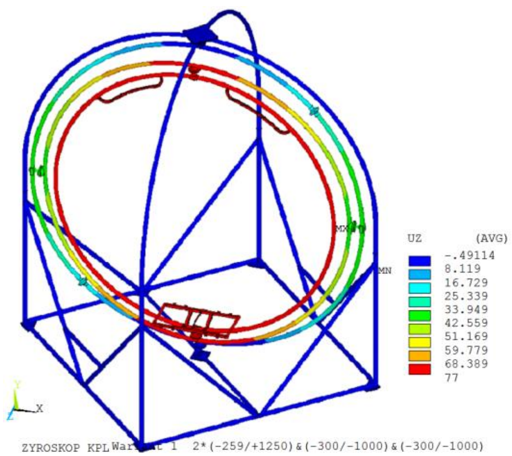
The project was done for MP-PZL Aerospace Industries , Poland



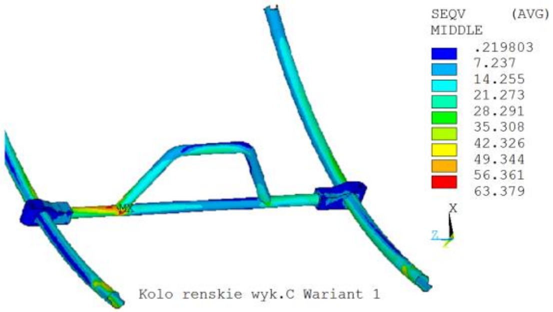
Total displacements in the wheel



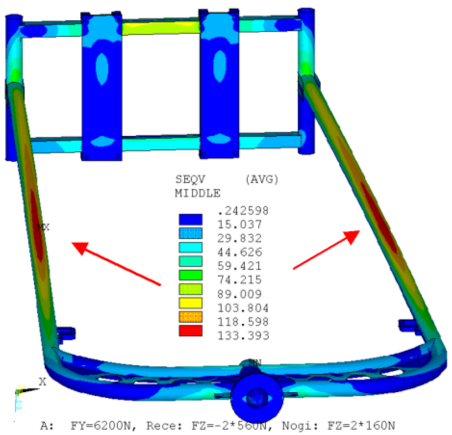
Total displacements in the looping device



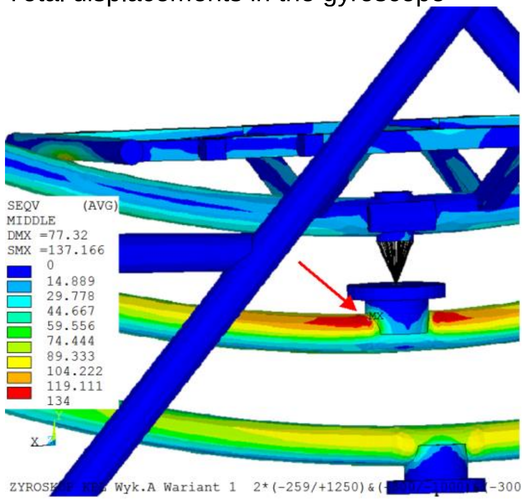
Total displacements in the gyroscope



Von Mises stress distribution



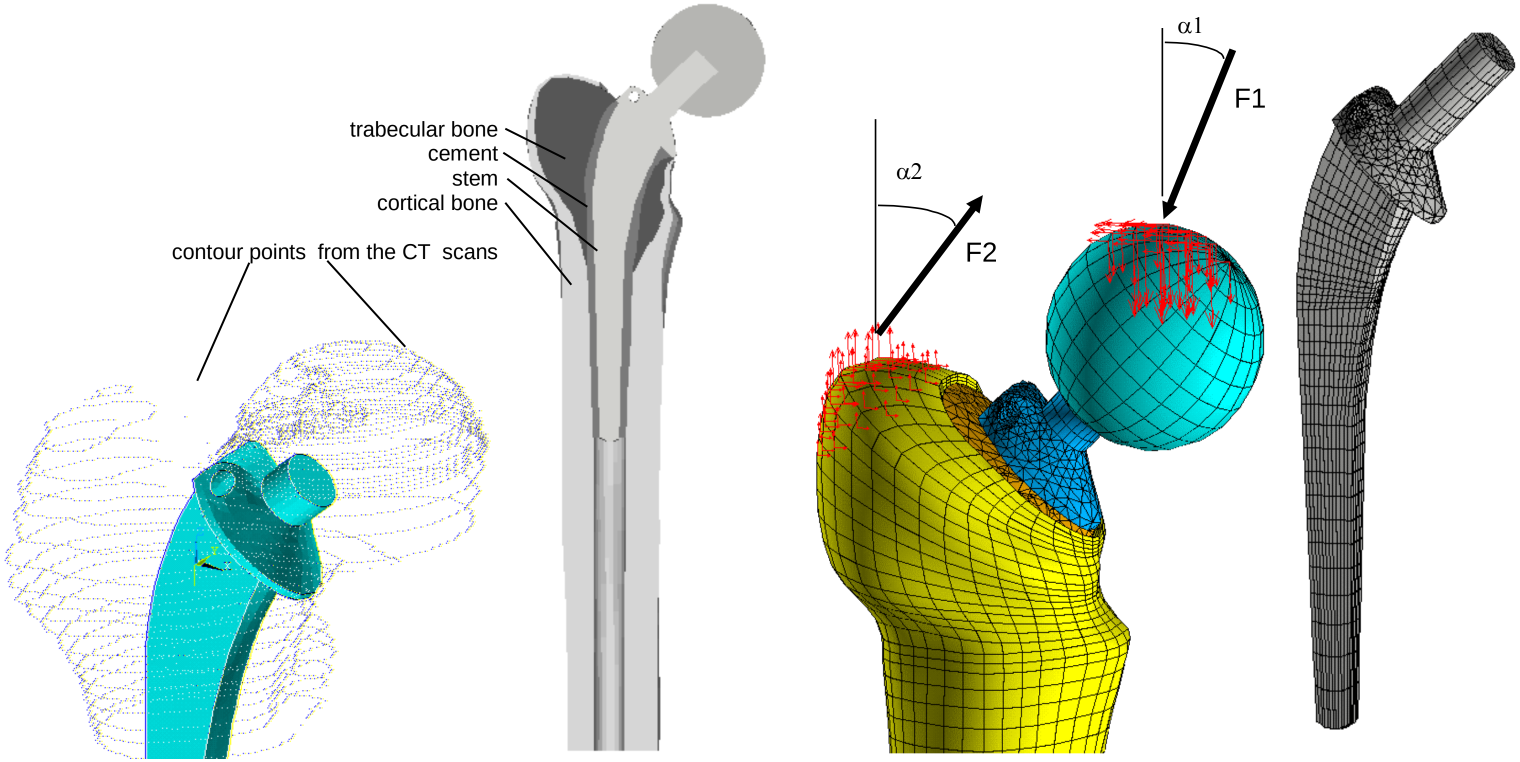
Von Mises stress distribution



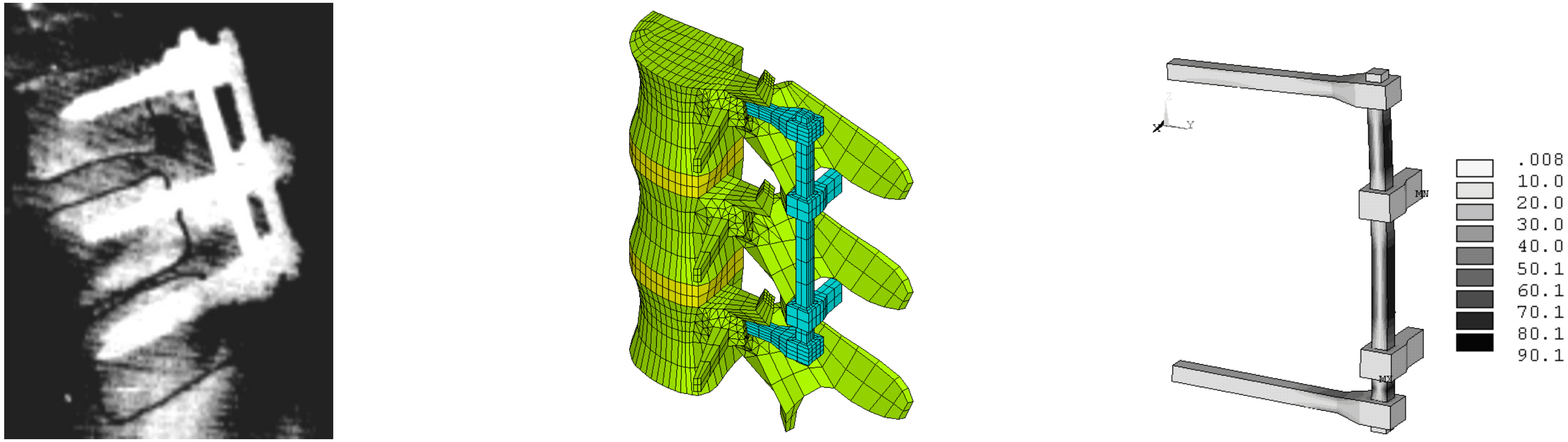
Von Mises stress distribution

Finite element method in bone-implant system strength analysis

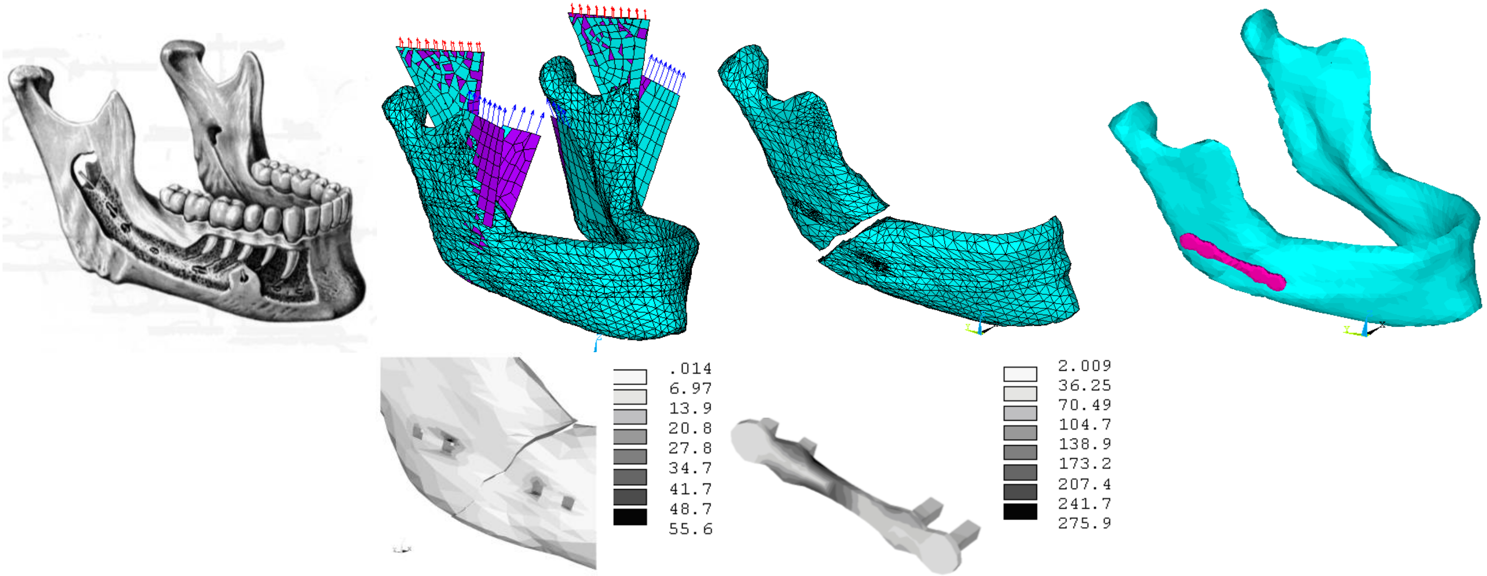
The three-dimensional FE models of the living tissues-implant systems can deliver the valuable information about mechanisms of stress transfer for the considered bone –implant system. In the presented example some different variants of the hip stem were considered to find the best solution, which should 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, $\alpha 1=28°$, $\alpha 2= 40°$*



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



Von Mises Equivalent stress (MPa)

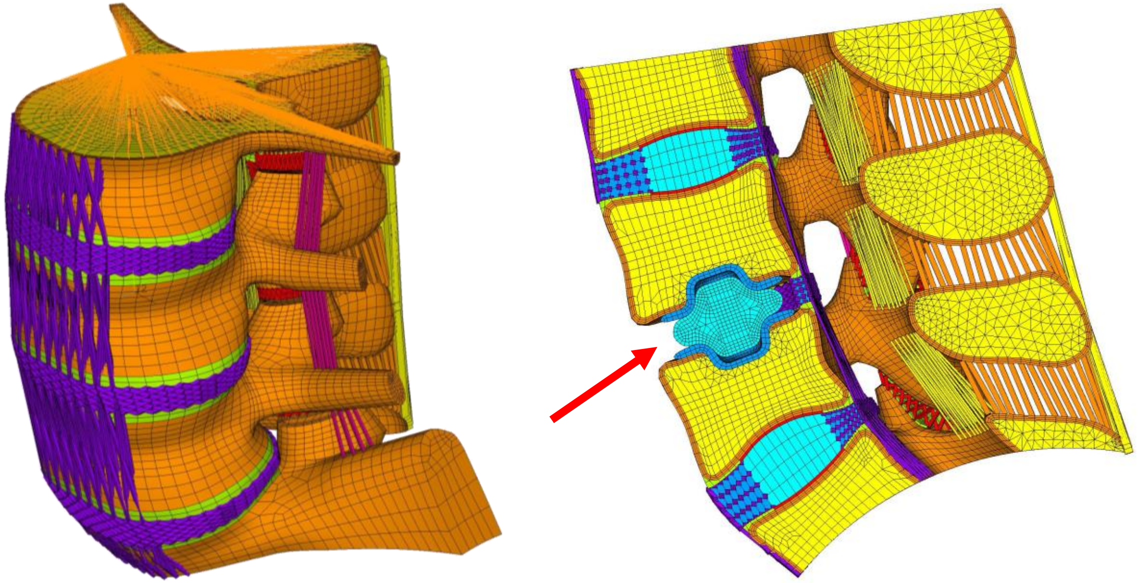
FE model and chosen results of numerical simulation of mandibular ostheosynthesis

Teoretyczne i doświadczalne badanie, projektowanie i wytworzenie prototypu sztucznego krążka międzykręgowego nowej generacji o podwyższonych właściwościach

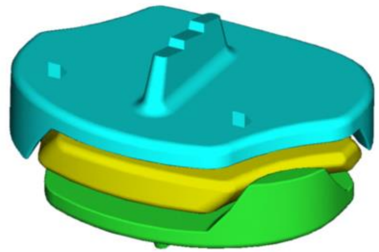
Od 2004 r. Pracownicy Zakładu brali udział w trzech projektach badawczych nad protezami krążków międzykręgowych, w których wykonano szereg symulacji MES w biomechanice kręgosłupa lędźwiowego.

Jedną z metod operacyjnego leczenia choroby zwyrodnieniowej krążka międzykręgowego jest wszczepienie protezy. Zadaniem sztucznego krążka jest zniesienie dolegliwości bólowych przez odtworzenie fizjologicznych zakresów ruchów w operowanym segmencie kręgosłupa. Proteza powinna posiadać długotrwałą wytrzymałość zmęczeniową oraz spełniać wymagania dotyczące biozgodności materiałów, wytrzymałości statycznej i dynamicznej, sztywności, sposobu implantacji i zamocowania do kręgów oraz obróbki powierzchni implantu zapewniającej zrost z kością.

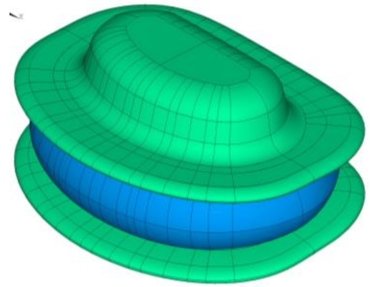
Na podstawie uzyskanych wyników analiz MES opatentowano przegubowy implant krążka międzykręgowego (patent P.382241 Implant krążka międzykręgowego kręgosłupa lędźwiowego, 16.07.2012) oraz zgłoszono wniosek o udzielenie patentu na implant podatny. ***(Projekt badawczy finansowany przez KBN (2001-2004), i NCBiR (2010 – 2013).***



Modelowanie MES kręgosłupa



Przegubowy implant krążka



Podatny implant krążka