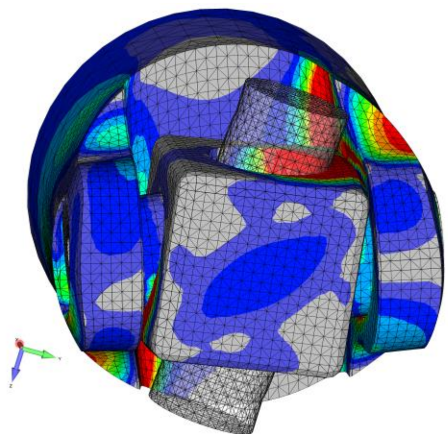
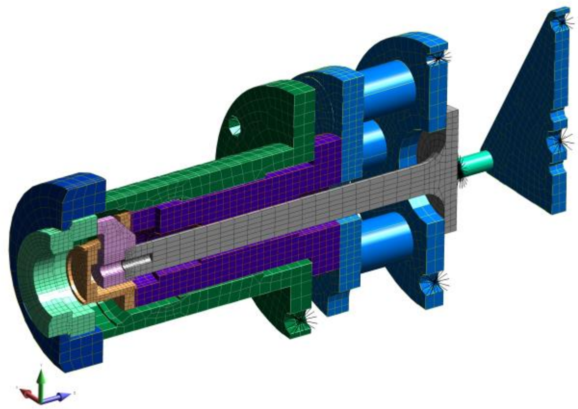
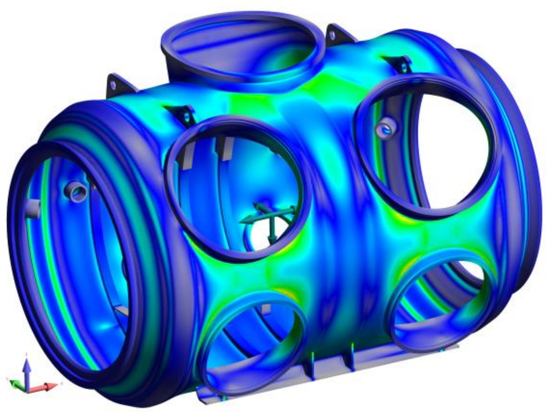
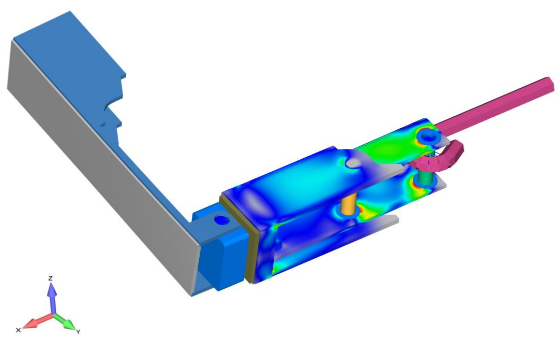
Insight Through Numerics: The World of FEA Simulation at Predictive Engineering









Title:

Insight Through Numerics: The World of FEA Simulation at Predictive Engineering

Keywords:

FEA simulation, nonlinear stress analysis of U-Joint, stress and vibration analysis of metal forming machinery, Buckling analysis of two-person submarine, stress and deflection analysis of hand tools, stress and fatigue analysis of 50 kW wind turbine blade hub nacelle and tower, vibration optimization of pulp mill machinery, vibration analysis space frame structures, stress analysis of vacuum chamber for scanning electron microscopy, deflection analysis of vacuum chamber, stress and fatigue analysis of

helicopter tail rotor blade, MMPDS fatigue analysis, nonlinear stress stiffening analysis of rotating blade aircraft, Femap, NX Nastran, ADINA,

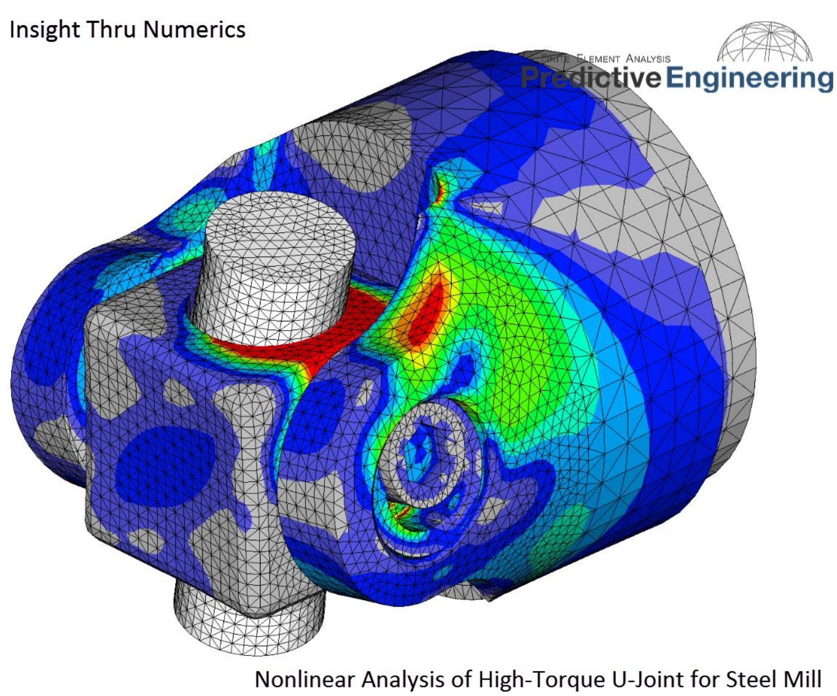
Project Overview

Predictive Engineering has done over 600+ FEA simulation projects in its 18 years in business. Our experience allows us to build FEA models quickly and correctly. This case study presents just a few of the classic stress analyses that we have done over the last several years. A more detailed listing can be found in this text listing Direct Text Listing of FEA LS-DYNA and CFD Projects.pdf

This short overview presents project work on:

- High-Torque U-Joint for a Steel Mill Leveler
- Vibration Analysis of Metal Forming Tool (Beer Can Domer)
- Stress and Vibration Optimization of Space Frame Structure (Car Wash Frame)
- Nonlinear Buckling Analysis of Two-Person Submarine
- Vibration Optimization of Paper Mill Machine Component
- Stress and Fatigue Analysis of 50 kW Wind Turbine Blade Hub, Generator Shaft and Nacelle
- Stress and Fatigue Analysis of Heavy-Lift Helicopter Tail Rotor Blade

YouTube Video:



Title: Insight Thru Numerics: The World of FEA Simulation at Predictive Engineering

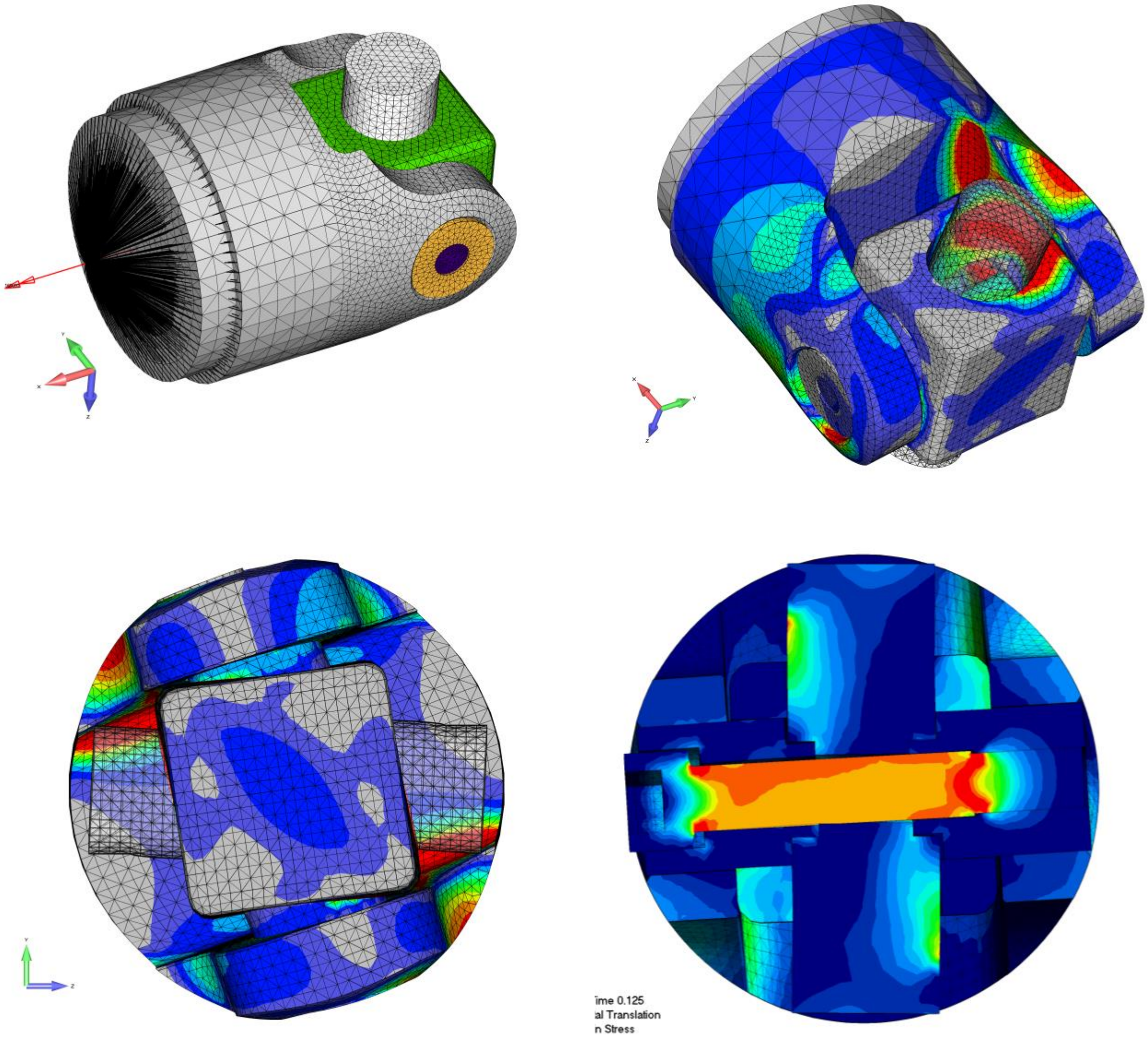


Figure 1: Stress and fatigue analysis of a high-strength U-joint for steel mill leveling table. The U-joint was fabricated using tool steel and high-strength steel. The Femap model was analyzed with all surfaces contacting and allowing for material plasticity using NX Nastran SOL 601 (ADINA) and LS-DYNA.

A high-strength U-Joint design was analyzed to determine its ultimate torque capability while maintaining acceptable fatigue life. The simulation work focused on improving the fatigue life of the main U-joint bolt and bolt-preload was one of the pivotal parameters investigated. The bolt was pre-loaded to 80% of its yield strength and the U-joint was torqued. Bolt stresses were determined to be very sensitive to machining tolerances between the block and the yoke.

At the end of the study, it was recommended that the prior Grade A8 bolt be replaced with an aircraft grade and polished. With these changes, the design has exceeded design specifications.

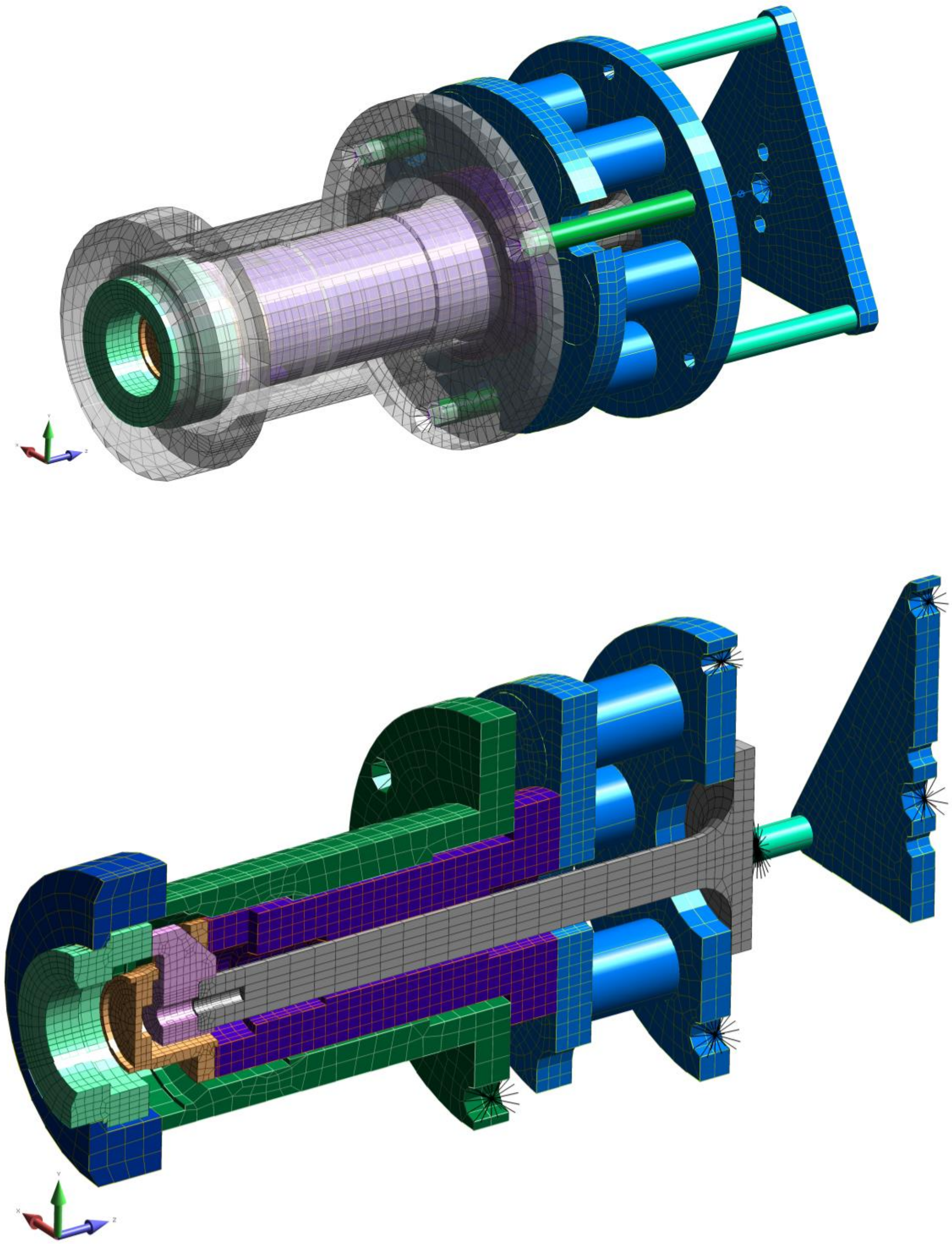


Figure 2: FEA simulation of stress and normal modes vibration of a beer can domer machine. The all hex model was built to allow quick and simple optimization of the components.

A major US beer brewer wanted to extend the service life of their beer can domer machines which were suffering from vibration problems after a few years of service. After an on-site visit in St. Louis to meet the engineering team and to see how the machines operate in the beer can manufacturing process, a finite element analysis model was built of the domer. This machine forms the bottom of the beer can by creating a uniform dish in the bottom that reinforces the can. Simulation work (stress and normal modes) determined that machine tolerances due to normal wear and tear were responsible for the increased vibration problem and pinpointed the component to maintain within specifications. This design solution was accepted and implemented.

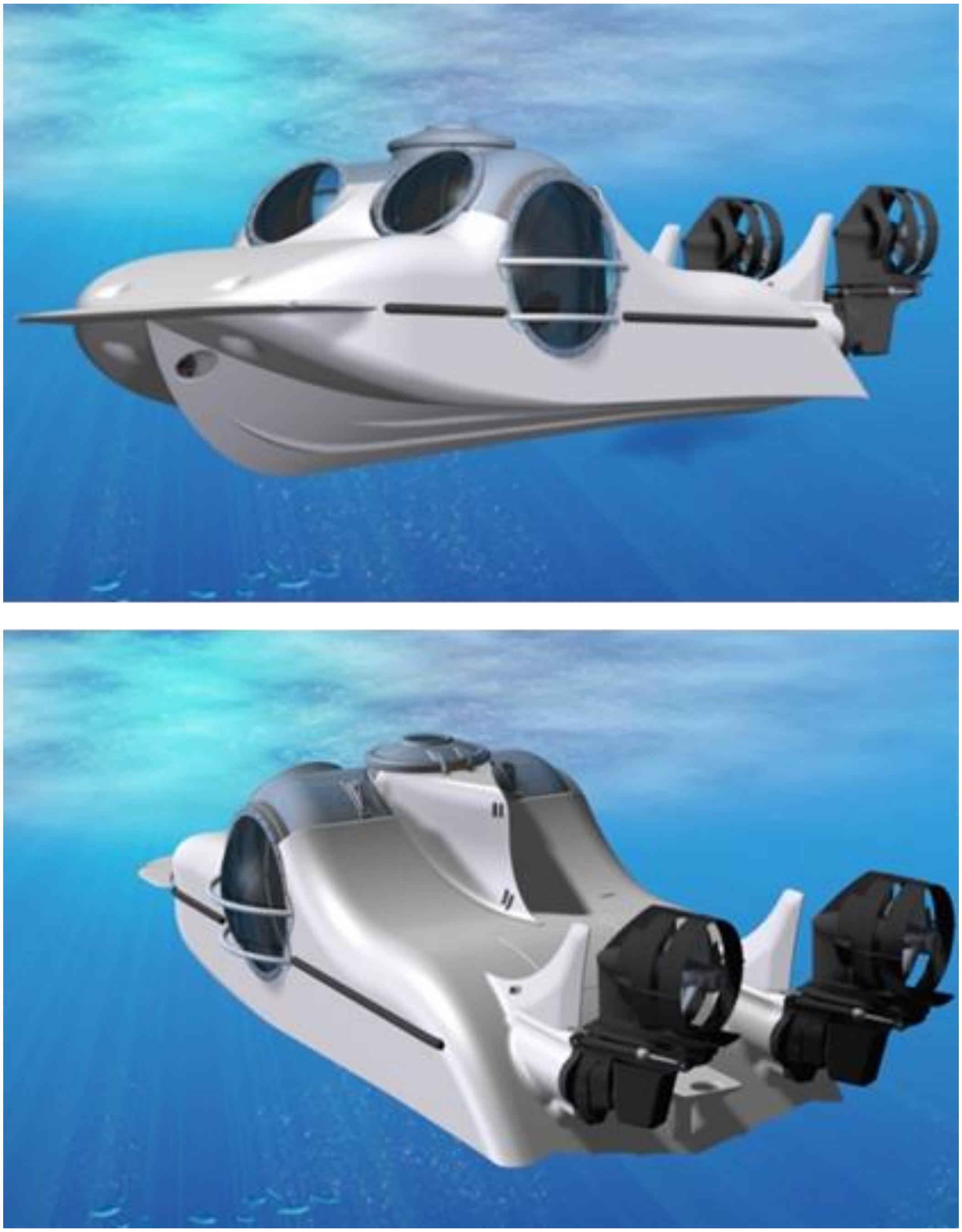


Figure 3: Another type of pressure vessel is that of a submarine hull. This hull was analyzed under the American Bureau of Shipping (ABS) Rules for Building and Classing Underwater Vehicles, Systems and Hyperbaric Facilities. Hand calculations were done on the basic hull design followed by a finite element model of the hull. Lifting loads and hatch cover calculations were also performed.

Figure 4: FEA simulation of stress and buckling of two-person submarine (shown previously) built using Femap from Siemens PLM Software.

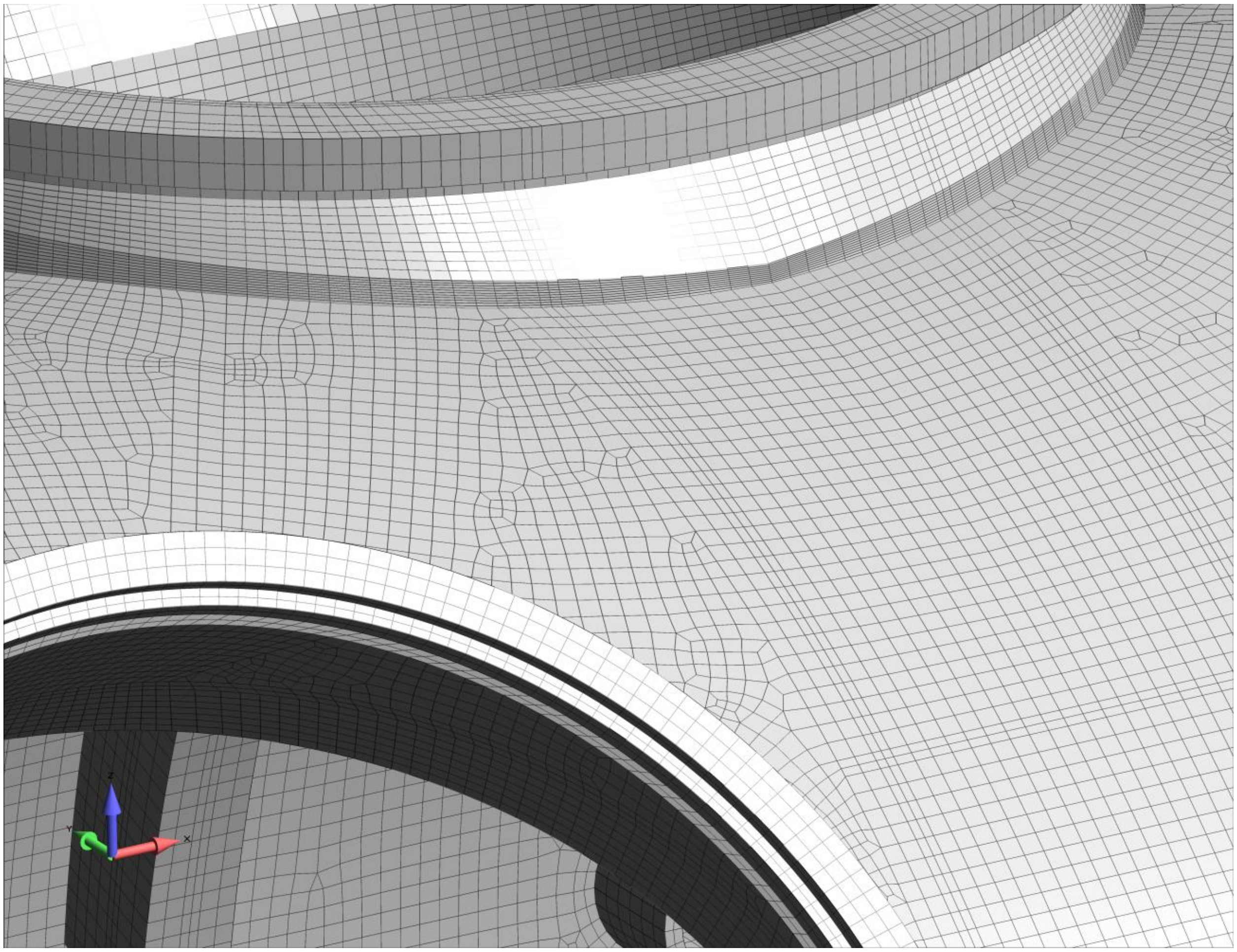


Figure 5: All hex (brick) mesh of submarine hull that provided high accuracy at low numerical cost. The hex mesh approach was chosen due to nonlinear analysis requirements to determine maximum dive pressure prior to hull collapse (buckling).

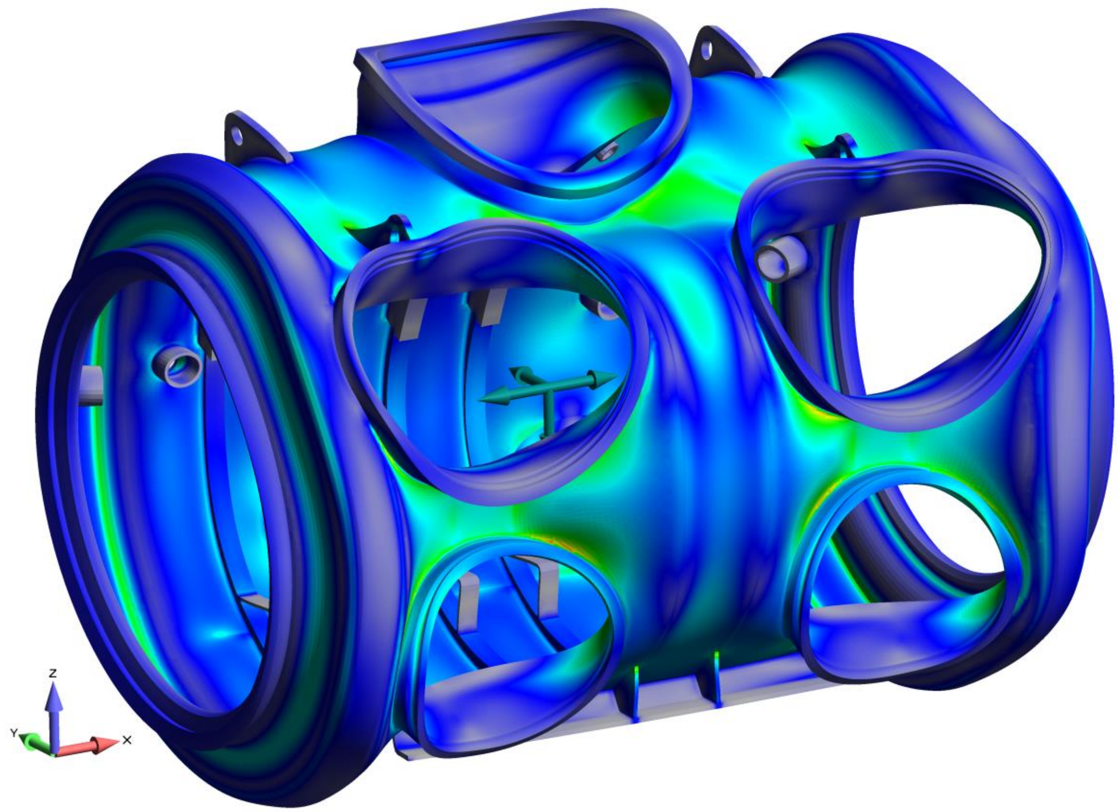


Figure 6: Stress pattern due to nonlinear buckling of pressure hull.

Stress and buckling analysis of submarine hulls is not common analysis due to the financial and human costs if the calculation is done wrong. Over the years, Predictive Engineering has worked on three submarine projects from a deep-diving luxury submarine to the world's deepest diving bathysphere (please see Fracture Mechanics of Glass.pdf) to this two-person compact model. In all three analyses, the analyses work tied to hand calculations for the American Bureau of Shipping (ABS) and for ASME calculations for hydrostatic pressure. The critical load analysis was for buckling since the hand calculations do not provide the level of detail that can provide the assurances that only FEA can provide. The buckling analysis is of particular interest since in all three models it was performed via linear methods (eulerian) and by a complete nonlinear approach with material plasticity. Of interest is that the numerical work showed that the dominate factor in hull buckling was the geometric aspects and that once the hull wants to buckle, material plasticity is not of a concern.

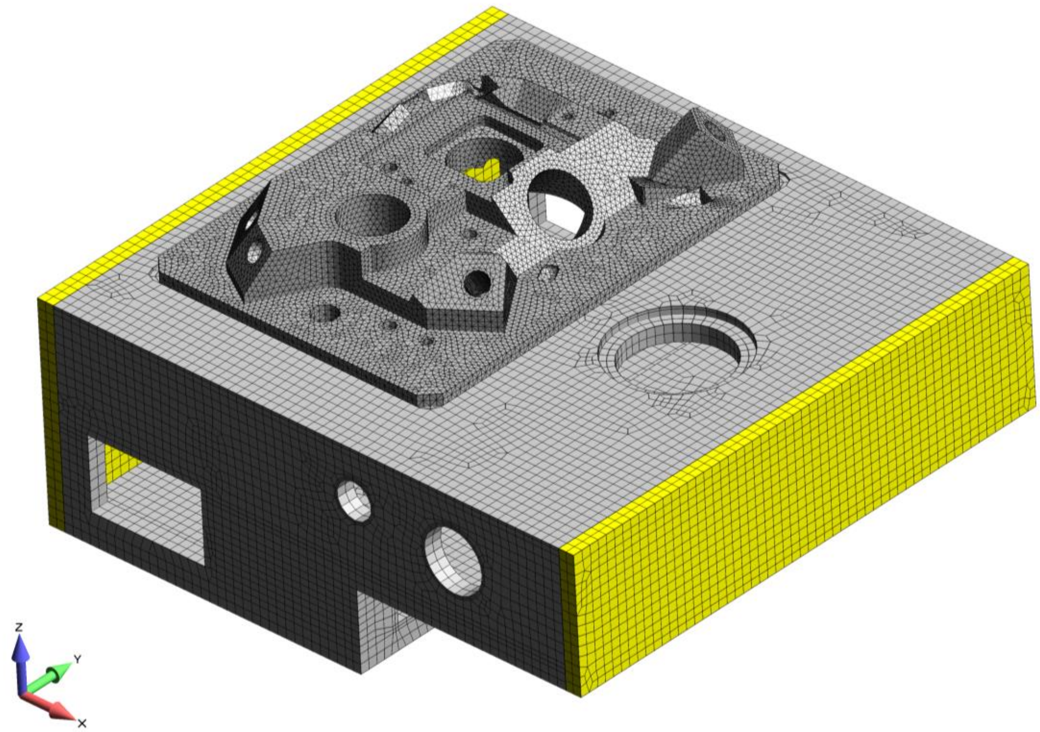


Figure 7: Finite element analysis model of electron microscope vacuum stage chamber built with Femap and analyzed with NX Nastran. High-precision deflection analysis for column alignment during operation under full vacuum.

The production of integrated circuits and their internal silicon "chip on a die" components relay upon detailed microscopy for the inspection of defects during the manufacturing process. Scanning and transmission electron microscopes are regularly used but with this one manufacturer, the process has been automated to fit within the production lines of major manufactures of integrated circuits. The FEA work was performed to determine the deflection of the column mounts and other mounts for apparatus used for ion milling and spectroscopy. The finite element mesh was built as a mix of hex and tet meshes. The hex was used in regions of the main chamber where the walls were thin and wide whereas the tet mesh was for the detailed region around the instrument mounts. The model was validated against experimental measurements and was found to be in excellent agreement. With the baseline validated, trade studies were performed to increase the stiffness of the structure without adding excessive weight.

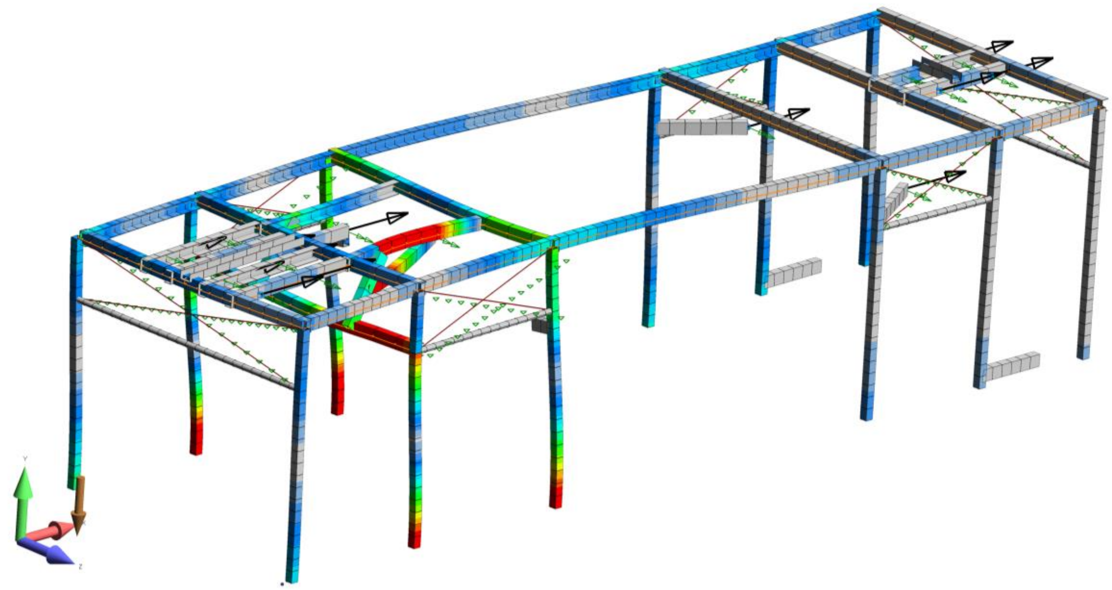


Figure 8: Predictive Engineering specializes in the idealization of structures and systems into numerical models. The space frame structure above is an idealization of the steel frame of a automotive car wash system. Loads were a combination of masses and forces to represent hanging weights and moving brushes.

This FEA simulation of car wash frame or space frame structure was extensively optimized to meet stress, vibration and fatigue requirements for its service life. The model was built with beam elements with static and moving weights idealized using mass elements. Structural steel shapes were used as standards for the beam properties such that the modeling results could be taken directly into manufacturing.

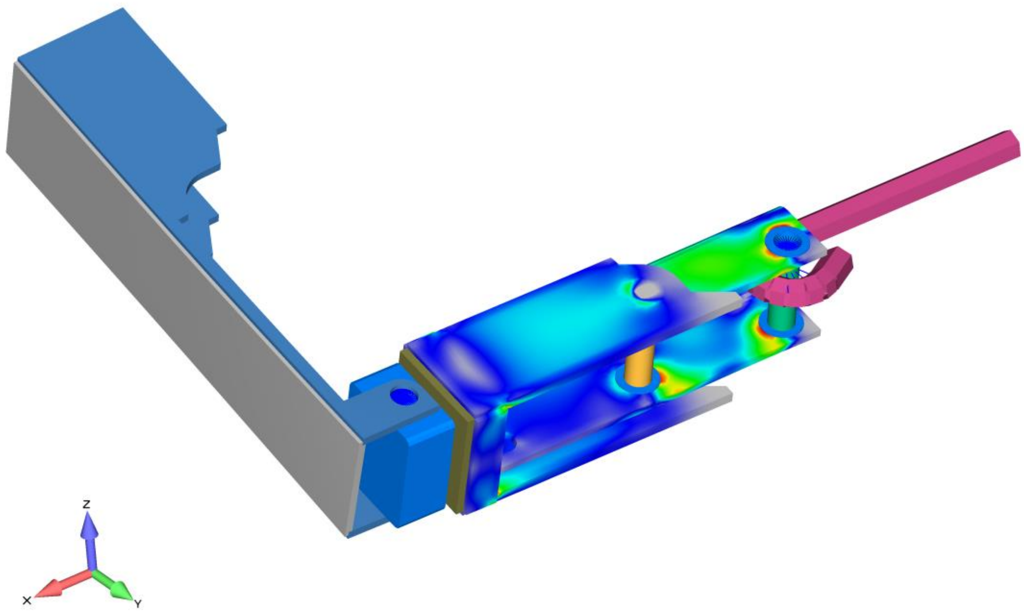
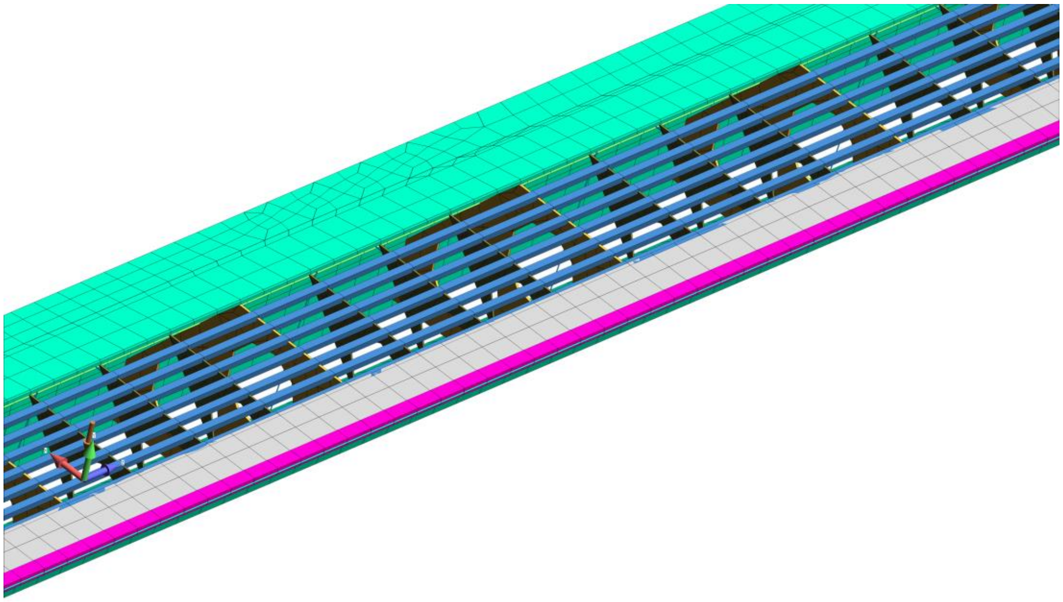


Figure 9: FEA Simulation of hand tool for ultimate mechanical strength during hex nut torque operations. Stress and deflection analysis to determine optimized ergometric operation.

This simple hand tool was analyzed for stress and deflection to determine its response to a maximum possible human torque load. The load definition was one of the most challenging parts of this analysis. The normal human hand power was estimated from the literature to be around 130 lbf. Stress and deflection results were used to optimize the tool dimensions and lower manufacturing costs.



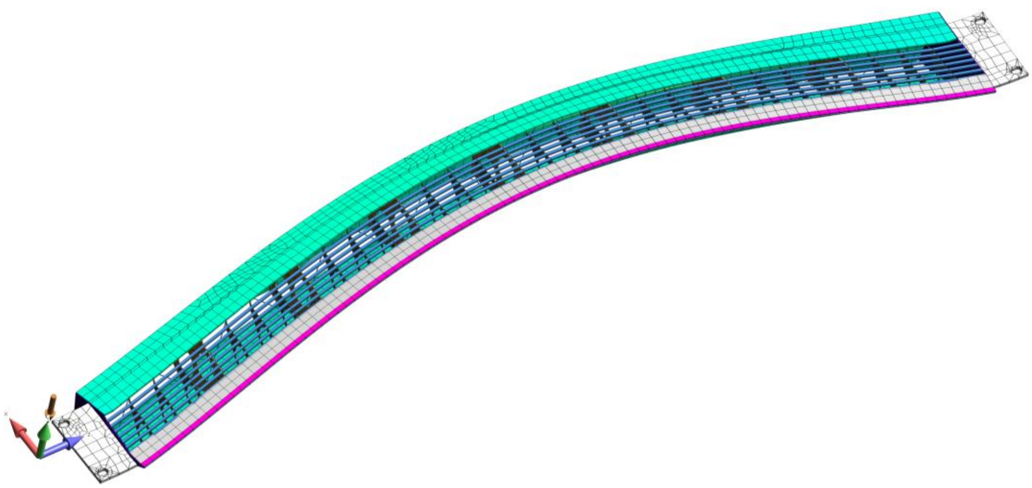


Figure 10: Normal modes analysis using NX Nastran of a paper mill forming board. Optimization of the structure developed a design that was stiffer and lighter than any other structure on the marked.

This FEA simulation optimized the frequency response of a five meter long forming board that is used in a paper mill. Through clever engineering, the forming board was stiffened using diagonal rods that moved its first mode response by more than 50% and added only 10% more weight. Given that the structure is 100% stainless steel, the new design cut material costs in half and more than paid for the analysis work. It is one of our favorite optimization examples where engineering sense beats any highly marketed optimization program that promises much but rarely delivers.

Figure 11: A stress and fatigue investigation was done on the tail rotor blade of a Sikorsky 54B helicopter. The objective was to extend the life certificate of the blade via a side-by-side fatigue life comparison to meet FAA requirements.

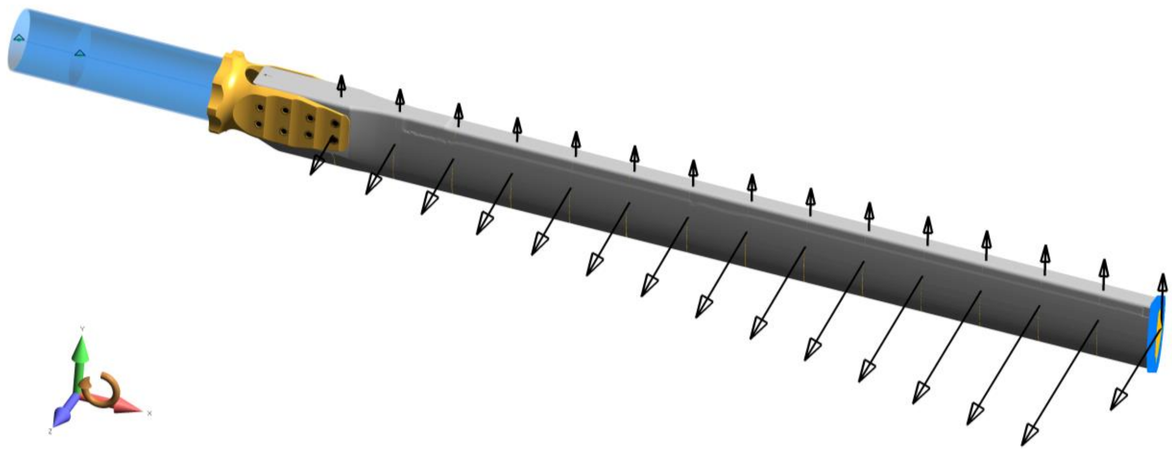


Figure 12: The load case was based on lift and drag calculations performed by a noted expert. The dominate load was the centripetal force of the blade spinning at 820 RPM.

The objective of this work was to demonstrate that the tail rotor blade would have comparable fatigue life as the original tail rotor blade. Fatigue calculations were done and tied to MMPDS data and prior work by a noted DER consultant based on the Walker Equivalent Stress Equation.

The analysis approach was based on nonlinear stress stiffening since the blade cuff was allowed to align itself to the centripetal force loading. Stress analysis demonstrated that the new blade exceeded prior requirements.

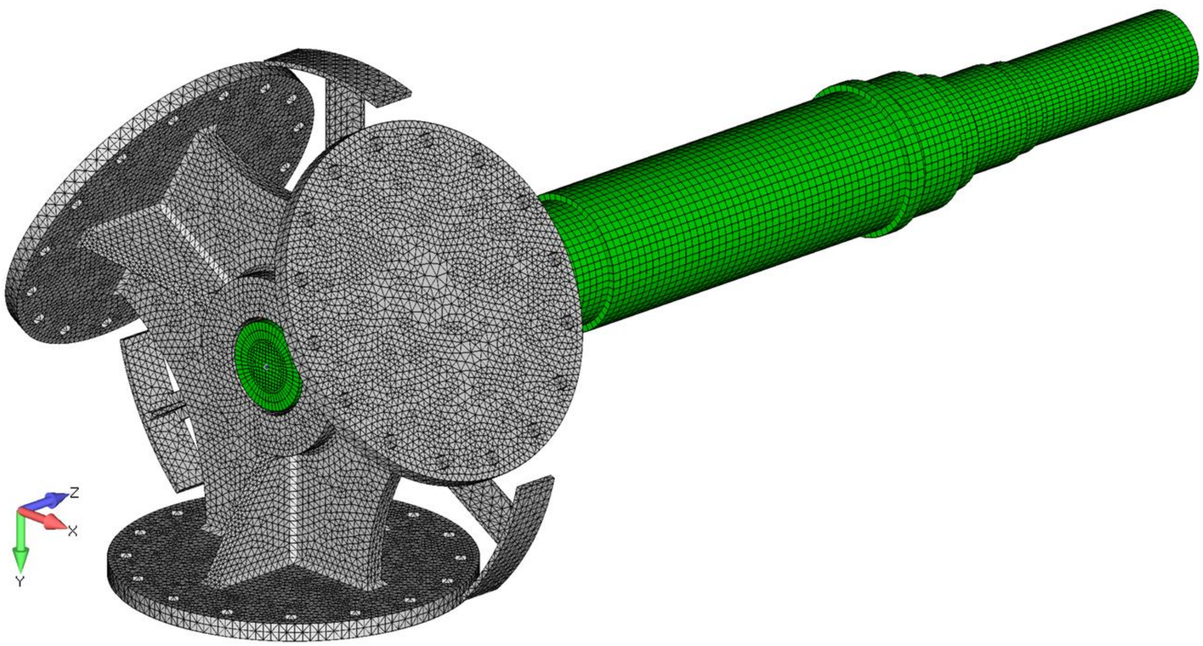


Figure 13: FEA simulation work by Predictive Engineering on 50 kW wind turbine blade hub, drive shaft and nacelle.

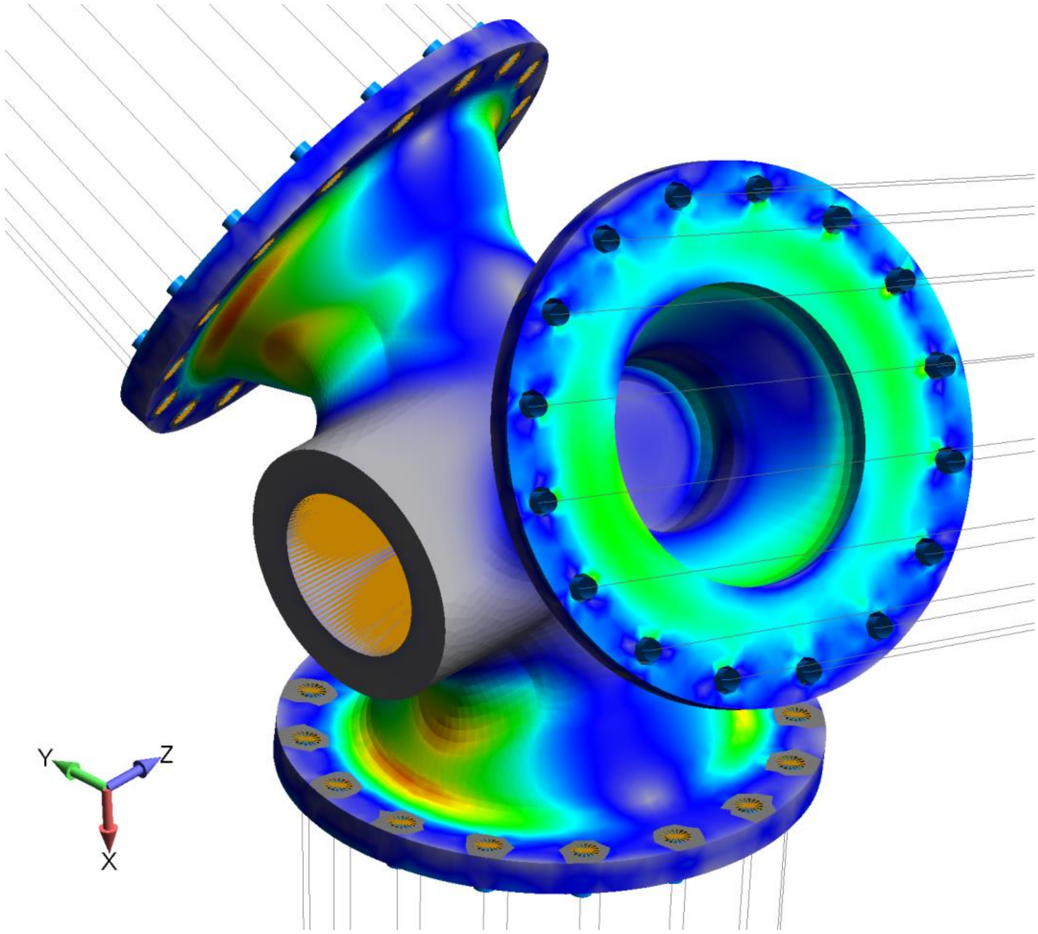


Figure 14: FEA simulation of 50 kW wind turbine blade hub. The lines are rigid links (RBE2) used to apply the blade loads onto the hub.

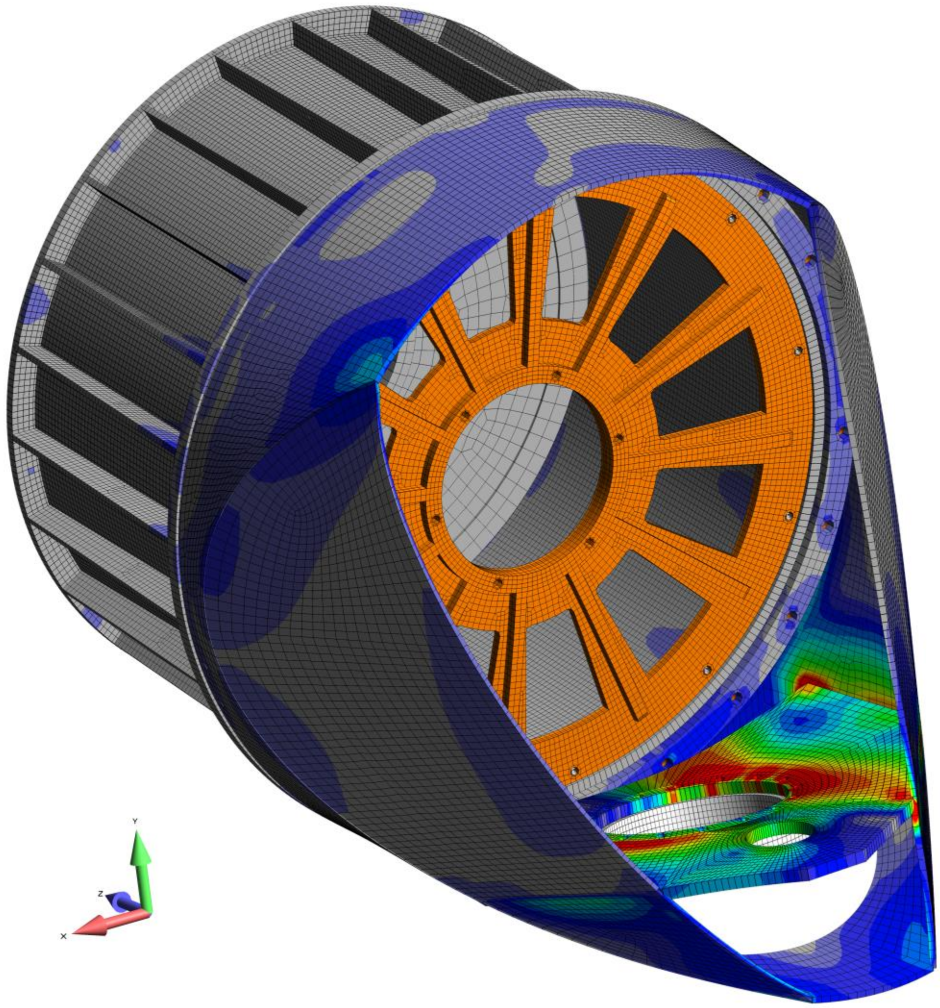


Figure 15: Stress analysis of 50 kW wind turbine nacelle given tower and blade hub loading.

This project was one of three wind turbine systems analyzed by Predictive Engineering. What makes this project unique is that it was the complete system from tower erection, stability, vibration to the stress and fatigue analysis of the blade hub, generator shaft and then the composite nacelle.

Contact

Predictive Engineering

2505 SE 11th, Suite 310

Portland Oregon 97202-1063

503-206-5571 (phone)

866-215-1220 (fax)

www.PredictiveEngineering.com