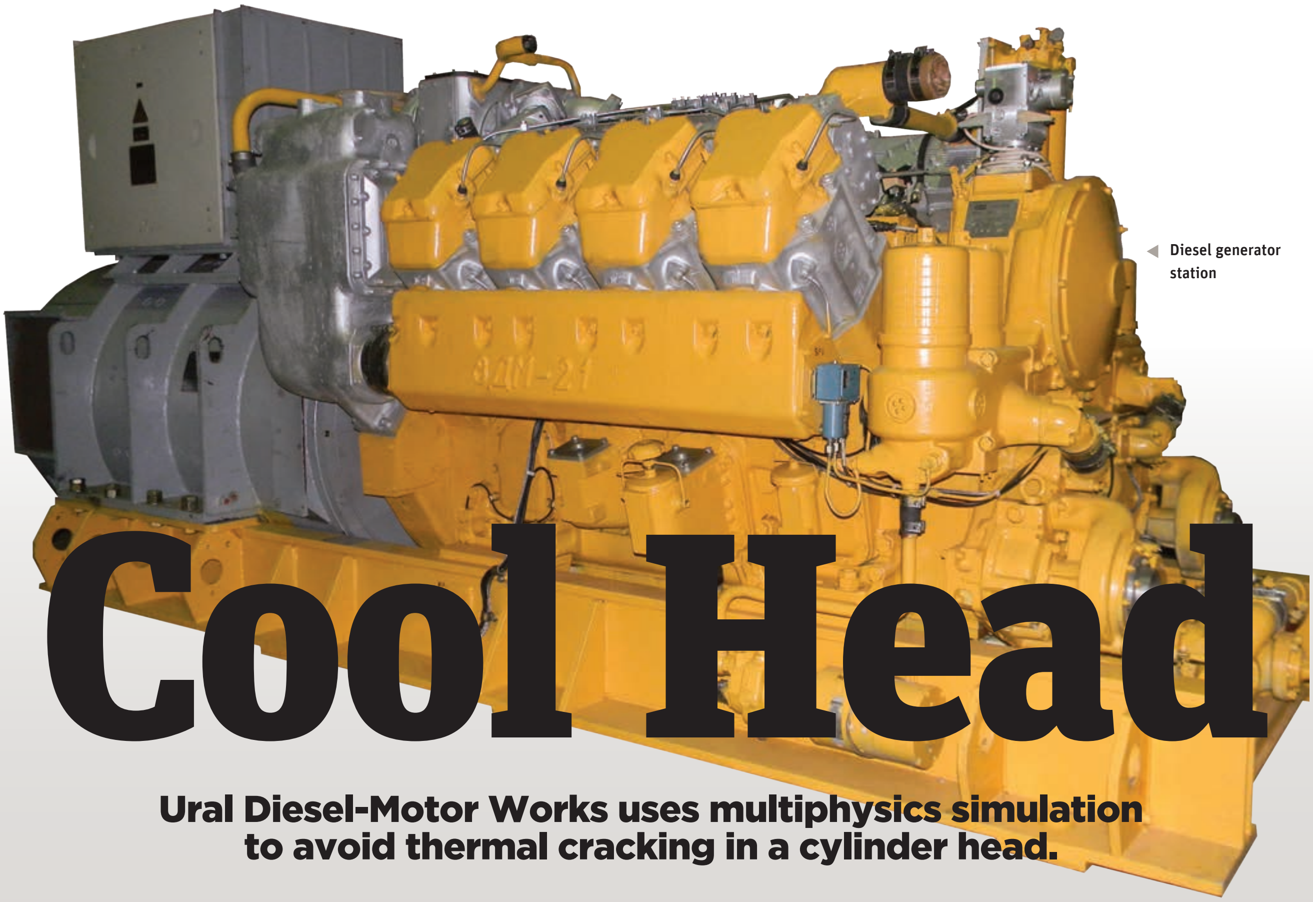


Diesel generator station

Cool Head

Ural Diesel-Motor Works uses multiphysics simulation to avoid thermal cracking in a cylinder head.

By **Dmitry Frolov**, Lead Engineer, and **Yuriy Abramov**, Lead Engineer, Ural Diesel-Motor Works LLC, Yekaterinburg, Russia

Robust product design requires engineers to consider the complete range of physical conditions a product will experience throughout its lifetime. Multiphysics simulations that involve, for example, fluid, thermal and structural influences to predict fatigue are an efficient way to thoroughly examine the range of conditions and ensure reliability. An engine made by Ural Diesel-Motor Works LLC that was designed several decades ago without the benefit of simulation experienced periodic problems with the aluminum cylinder heads. Cracks appeared in the cylinder head near the injector, and water leaked from the cooling jacket into the space between the head and injector. In extreme cases, the water mixed with fuel, causing the engine to stop running.

Ural Diesel-Motor Works opened for business in 2003 in what used to be the JSC Turbomotor diesel engine plant. The company builds diesel engines used in marine and railroad applications ranging from 1,050 HP to 2,600 HP. It also builds diesel generator stations with power up to 1,600 kW. The company recently established a simulation team, and the cylinder head problem was one of the first issues it addressed.

CFD SIMULATION

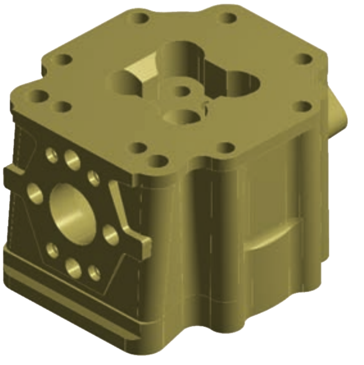
The specific Ural Diesel-Motor Works engine is used in locomotives and diesel generator stations. To improve its durability, the company's engineers started by importing the CAD model of the original cylinder head into ANSYS DesignModeler to replicate the

A multiphysics simulation that involves, for example, fluid, thermal and structural influences to predict fatigue is an efficient way to ensure reliability.

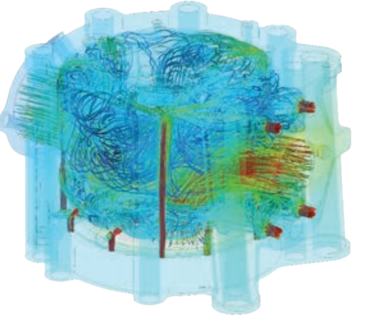
The new design delivers the long life and high quality that the company's customers have come to expect.

head's internal geometry and simulate the flow through the cooling jacket. Unnecessary details were suppressed, and engineers meshed the fluid region using tetrahedral elements with an inflation layer near the wall for accurate resolution of the boundary layer.

Engineers opened the mesh in ANSYS CFX computational fluid dynamics (CFD) software and assigned properties for the fluid and boundary conditions. They simulated the flow through the cooling jacket, looking for areas where fluid velocity is low. This low velocity restricts heat transfer from the head to the cooling water and causes excessive temperatures that can, in time, reduce reliability. Next, engineers applied temperatures from experiments to the CFD model and reran the simulation to generate temperature fields throughout the inside walls of the cylinder head.



▲ CAD model of original aluminum block



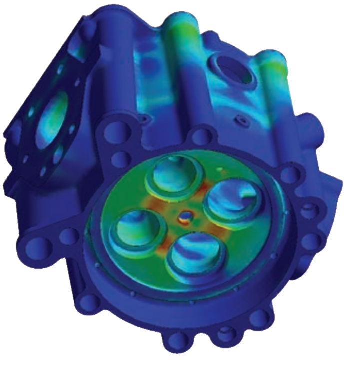
▲ CFD simulation of cooling jacket

THERMAL–STRESS SIMULATION

The team returned to the original CAD model in ANSYS Workbench and used it to create a structural model in ANSYS Mechanical finite element analysis (FEA) software. Automatic contact detection capabilities were used to identify and configure contacts in the model. Mechanical loading was applied to the model, first by initiating contact of components that mate with the cylinder head and pretension of bolts used to hold the head to the block and other components. Engineers then used ANSYS Workbench coupling to apply temperature fields (determined by the CFD simulation) for use in ANSYS Mechanical to calculate the thermal stresses associated with these temperatures. Engineers selected the model's inside faces and applied the temperature fields generated in the CFD simulation as temperature loads. The last step was applying the pressures generated during the engine's working cycle. In the early stages, engineers generated several load cases based on different phases of the engine cycle, but later they determined that they could save time by using only the highest load case.

ANSYS Mechanical calculated the mean stresses and amplitude of stresses from the complete loading applied to the head. The calculated stresses were higher than the yield strength of the aluminum material. Engineers had clearly identified the root cause of the cracking problem.

Ural Diesel-Motor Works engineers then worked to address the problems uncovered by simulation. They decided to change the head material to a grade of cast iron, which has a considerably higher yield strength than aluminum. Engineers modified the cooling system to address the low-velocity flow areas revealed in the earlier CFD simulation. They reran the CFD simulation and generated temperature fields for the modified head, then imported these temperature fields into ANSYS Mechanical.



▲ Von Mises stresses caused by temperature and pressure on block

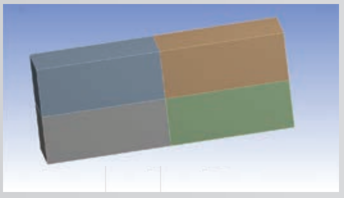
SHAPE OPTIMIZATION

Because cast iron is heavier than aluminum, the team needed to change the head's geometry to reduce its weight. Engineers used the ANSYS shape optimization module to redistribute the material in the cylinder to reduce mass to a minimum while maintaining stiffness above a defined minimum value. The output of shape optimization is a contour plot that shows where material can be removed with the least impact on overall stiffness.

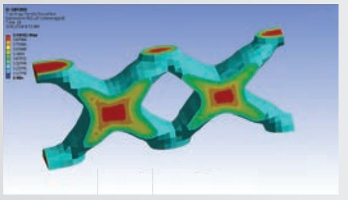
Engineers were limited in the changes they could make because they needed to maintain interfaces with mating parts. However, they were able to substantially reduce the head's weight using the shape optimization tool. The weight of the resulting design is somewhat higher than the previous aluminum head but still

▲ Optimized geometry of cast-iron head

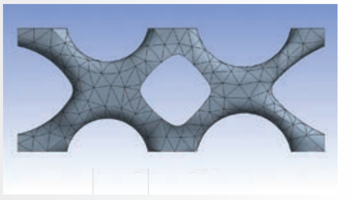
Topology Optimization for ANSYS Mechanical



▲ Initial Design Space



▲ GTAM Solution



▲ Final Part

Engineers realized 70 percent mass reduction while minimizing strain energy (maximizing stiffness) using GTAM.

Due to stringent governmental regulations for improved fuel economy and emissions, vehicle manufacturers and suppliers around the world are pursing lightweighting to make their products lighter while ensuring that design requirements are satisfied. Weight reduction can be realized through advanced material design and the introduction of new material types, such as carbon nanotubes or carbon fibers. Manufacturers recognize that tremendous potential exists to reduce vehicle weight even with traditional materials, like carbon steel, plastic and aluminum.

Topology optimization is not new to the modeling and simulation space, but its adoption has been restrained over the years for a number of reasons, including fabrication limitations. The realization of additive manufacturing has circumvented many of these constraints, leading to a resurgence of topology optimization.

GENESIS® Topology for ANSYS Mechanical (GTAM) is a new partner solution that adds topology optimization to the ANSYS Mechanical environment. ANSYS customers can benefit from automatically generating innovative designs in a reliable, robust and easy-to-use interface. The tool is offered by Vanderplaats Research & Development, Inc., whose developers collectively have nearly 100 years of experience in optimization, research and software development.

— **Shane Moeykens,** Strategic Partnerships Manager, ANSYS

ANSYS nCode DesignLife captured the data, data flow and parameters in the ANSYS Workbench integrated environment and performed a comprehensive fatigue analysis using the stress-life approach.

PREDICTING FATIGUE LIFE WITH ANSYS nCODE DESIGNLIFE
ansys.com/82cool

at an acceptable level. (See sidebar for information on recent advancements in ANSYS topology optimization.)

FATIGUE ANALYSIS

Finally, Ural Diesel-Motor Works engineers used ANSYS nCode fatigue analysis software to calculate high-cycle fatigue safety factors. They reran the thermal-stress analysis with the new design, exported the stresses to nCode, and combined the results with a material model and description of the repetitive loading that the product is expected to undergo during operation. ANSYS nCode DesignLife captured the data, data flow and parameters in the ANSYS Workbench integrated environment and performed a comprehensive fatigue analysis using the stress-life approach.

The results showed that the new design delivers the long life and high quality that the company's customers have come to expect. Engineers are confident that cracks will not reappear based on simulation results. The weight of the new cylinder head is only slightly more than the old design. This is just one in a series of applications in which the company's new simulation capability is helping to improve the quality and reliability of its products.