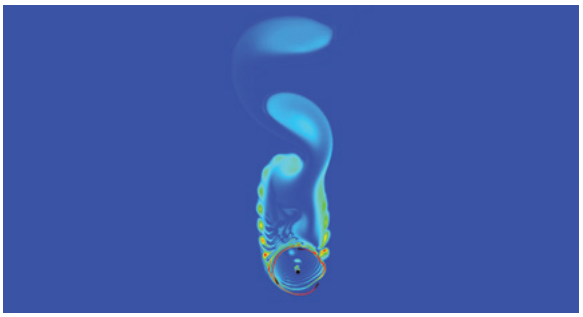
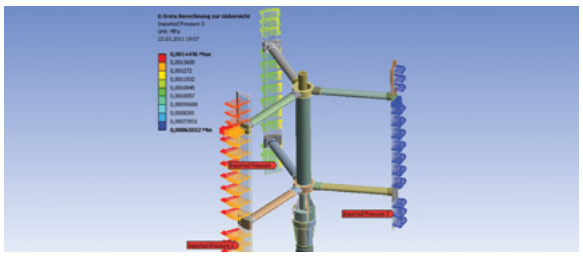
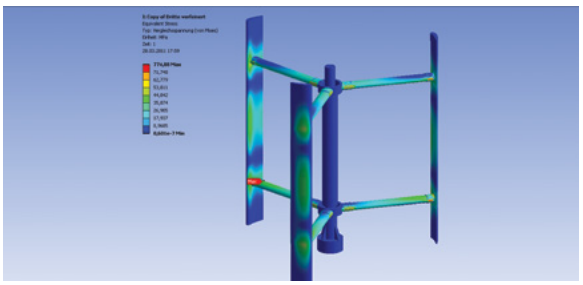
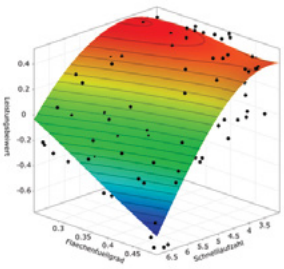
Robust Fluid-Mechanical Design





▲ CFD simulation and results





▲ Structural simulation and optimization

At Dortmund University of Applied Sciences, master's students in Professor Marius Geller's Computer Simulation in Mechanical Engineering course use ANSYS to conduct parametric studies and apply robust design principles. Recently, Geller's students focused on optimizing the structural and aerodynamic characteristics of wind turbines — using ANSYS in an HPC environment to consider hundreds of design variations.

At Dortmund University of Applied Sciences in Germany, Professor Marius Geller challenges his students to conduct extremely sophisticated engineering simulation exercises as part of the master's in Mechanical Engineering program. Each year, only 15 to 20 students are admitted to this highly competitive program, which is designed to provide young engineers with the advanced skills — such as simulation — that they need to succeed in the workplace.

Geller teaches an 18-week course called Computer Simulation in Mechanical Engineering, which begins by introducing students to the mathematical and physical problems underlying performance of a variety of products — ranging from Formula 1 cars and robotic hands to industrial compressors and ship propellers. Next, students begin to optimize product performance by modeling these problems using ANSYS software.

The course culminates in a real-world engineering project, in which each student applies engineering simulation and optiSLang to robustly design a product optimized for characteristics such as strength, energy efficiency, light weight or aerodynamics. While they work individually, all students in the class focus on designing the same product. In recent years, Geller's design projects have included a boat, a wind turbine and a bicycle.

"Whether they come into the course with any knowledge of ANSYS or engineering simulation, by the end, my students are working at a highly sophisticated level," notes Geller. "They create robust products by using the same parametric design principles that the world's leading engineering teams apply. For example, if they are optimizing a ship's hull, they set up ANSYS software to test more than 200 design variations in an automated fashion. They apply multiple physics, simply because that is the way engineering teams work in the real world today."

Why is ANSYS the right tool on which to build this challenging class? "I have used ANSYS for almost 20 years, and I find it to be the best software for multiphysics studies," says Geller. "Its ability to produce native results for both finite element analysis and fluid dynamics is unmatched. The user-friendly ANSYS Workbench platform — combined with the software's parametric capabilities and compatibility with high-performance computing environments — place even the most advanced simulations within the reach of my students."

Yet another reason is that, in Geller's opinion, ANSYS is the industry-standard simulation software for German engineering teams. "Recently, one of my students received 10 job offers when he showed his ANSYS simulation project to potential employers," adds Geller. "In fact, it has become a challenge to keep students in the program after they learn ANSYS, simply because their expertise in engineering simulation makes them so attractive to recruiters."