By Murali Kadiramangalam, Director, Academic Program, ANSYS, Inc.

Engineers need advanced skills to tackle today’s complex, multidisciplinary problems. By partnering closely with engineering schools, ANSYS helps ensure that the next generation of engineers is competent to tackle real-world problems in the highest-impact manner. In addition, at labs around the world, ANSYS fuels groundbreaking academic research that has the potential to shape the future.
Since ANSYS was founded in 1970, a company hallmark has been its close connection to the academic world. Just as multiphysics engineering simulation software has revolutionized the way professional engineers design and test products, it has changed the work of academic researchers — allowing them to produce groundbreaking technical research reliably, faster and more cost-effectively than ever. In classrooms around the world, ANSYS solutions have helped generations of students prepare to tackle real-world engineering challenges.

Each day and in every corner of the globe, thousands of engineering faculty, researchers and students leverage the power of ANSYS. Annually, at least 8,000 academic papers and 12,000 dissertations rely on simulations using ANSYS solutions. More than 2,400 teachers have embedded the tools into their engineering curricula, with over 86,000 students enrolled in these classes. Another 10,000 faculty employ the software in academic research.

In the world’s leading engineering schools, ANSYS solutions have proven their value not only in delivering the highest-quality, most-relevant education to students — but in supporting the kind of outstanding research that attracts funding, while helping to increase national and global rankings.

REAL TOOLS FOR REAL ENGINEERING PROBLEMS
In this special academic issue of ANSYS Advantage, university professors from around the world describe why they use ANSYS software in their classrooms. They all give basically the same answer: because simulation is a requisite part of the skill set that today’s entry-level engineers are expected to have.

In the professional workplace, the majority of engineering teams have reduced cost- and time-intensive physical experimentation and testing, replacing them with virtual product design and verification. The sooner that entry-level engineers can play active roles in simulation-driven product development, the greater their value to an engineering organization. According to Professor Marius Geller of Dortmund University of Applied Sciences, graduate students with proven simulation skills are often recruited before they even finish their master’s degrees.

ANSYS software allows academic researchers to produce groundbreaking technical research reliably, faster and more cost-effectively than ever.
At the University of Iowa, Professor Frederick Stern has developed a customized educational approach using the ANSYS Workbench interface and tools that help students quickly learn to use ANSYS Fluent for advanced computational fluid dynamics (CFD) simulations, while reinforcing introductory and intermediate fluid mechanics concepts. This work was done in partnership with research scientist Dr. Maysam Mousaviraad, graduate student Timur Dogan and undergraduate student Michael Conger. This approach, and a series of teaching modules created by Stern and his team, form the foundation of CFD laboratories for two undergraduate courses at the university: Mechanics of Fluids and Transport Processes, and Intermediate Mechanics of Fluids.

“I wanted my students to be able to perform complex CFD simulations without a steep learning curve,” says Stern, who is the George D. Ashton Professor of Hydroscience and Engineering at Iowa. “So I worked with academic experts at ANSYS to create a unique approach that teaches students systematic CFD modeling, numerical methods and procedures in a hands-on, user-friendly, interactive manner.”

Stern’s educational approach for ANSYS Fluent automates the CFD simulation process, leading students step by step through setup and solution of a range of realistic engineering problems. This approach has proven popular with students at the University of Iowa and has also been adopted by other universities.

A number of recent ANSYS initiatives are making it easier than ever for students to access and apply simulation software. One great benefit is the flexibility built into the student version of ANSYS software, which can be accessed in the dorm room, classroom or lab. Students can take advantage of a range of live and recorded training sessions as well as web-based learning portals to improve simulation skills in a self-guided manner. The company sponsors a student-focused Facebook page, supported by a growing user community, where students can exchange best practices, ask questions and offer advice.

Around the world, student competition teams rely on ANSYS software to design cars, aircraft, robots and other products for competition. By leveraging the power of engineering simulation to make design candidates sturdier, lighter-weight, faster and more energy-efficient, they attack the same complex problems they will face as working engineers. ANSYS technology has helped dozens of teams win awards, scholarships and other accolades.
Robust Fluid-Mechanical Design

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.”
As the editor-in-chief of the journal *Building and Environment*, Professor Qingyan Chen of Purdue University is an internationally recognized expert in designing comfortable, healthy and environmentally responsible environments.

In both his teaching and research at Purdue — where he is the Vincent P. Reilly Professor of Mechanical Engineering — Chen relies on ANSYS Fluent to visualize and address air quality issues in interior and exterior spaces. His work is aimed at designing innovations such as new building facades or heating, ventilation and air conditioning (HVAC) systems that optimize occupants’ health and comfort, while also meeting environmental goals.

In Chen’s popular class Indoor Environment Analysis and Design, seniors and graduate students apply Fluent to model interior spaces and map air flow patterns. Students test the effects of various architectural changes — such as operable windows or new HVAC vent locations — on air quality, temperature, humidity and energy efficiency. Chen’s students also work to ensure healthier spaces by creating airflow patterns that minimize the spread of viruses and other airborne contaminants.

“ANSYS Fluent is an ideal tool for students, because it has an intuitive interface that is easy to learn,” notes Chen. “In addition, Fluent is very stable, even when managing the numerically large simulations that are typical in environmental design. Students can become easily frustrated by slow processing times or system crashes, but these are not an issue with ANSYS software.”
In addition to using ANSYS Fluent to introduce graduate and undergrad students to concepts in fluid dynamics, Professor John Cimbala has relied on that software in his own research for over two decades. Cimbala leads the Hydropower Research Program at Pennsylvania State University, which applies funding from the U.S. Department of Energy to solve real-world industry problems via advanced simulation and analysis.

Cimbala’s current research efforts focus on understanding turbulent flow patterns as water flows through the hydroturbine at a hydropower plant. Certain characteristics of the flow – such as cavitation – can have a negative impact on equipment, shortening its life and increasing maintenance requirements.

“When water exits a hydroturbine, it expands and makes a 90-degree turn, which causes an unsteady vortex rope to form inside the flow. This is a real problem, causing vibration and material erosion inside the exit shaft — while also limiting the operating range and efficiency of the turbine,” notes Cimbala.

The professor and his research team use the detached eddy simulation (DES) capabilities of Fluent to simulate flow conditions that combine high turbine performance with minimal vibration and unsteadiness in the exit shaft, called the draft tube. They also are exploring the effects of new equipment and process innovations, such as injecting the vortex rope with a jet of water to minimize its effects.

“Obviously, turbulence is a complex problem, and our models routinely include tens of millions of cells,” says Cimbala. “We are running massively parallel simulations across hundreds of high-performance computing processors. We have been very pleased with the fidelity, accuracy and speed of ANSYS Fluent as it runs these simulations. The software is easy to work with and well suited for this kind of unsteady-state work.”
Professor Ever Barbero has written the book on using ANSYS software to perform finite element modeling (FEM) for advanced composites materials. This faculty member at West Virginia University recently published the second edition of his popular book, *Finite Element Analysis of Composite Materials Using ANSYS*.

"In many industries, composites are now the material of choice for combining light weight with high strength," says Barbero. “But it can be difficult to mimic the behaviors of the various materials, including fibers and layers, via engineering simulation. I have found ANSYS capabilities to be ideal for accomplishing this task — providing fast, accurate answers about how to best configure composites in designing new products.”

Researchers at University of Rome Tor Vergata used ANSYS Fluent and RBF Morph to predict airfoil performance in the presence of ice formations. These tools helped to accurately account for the presence of complex shapes that originate during ice accretion — and automatically updated the airfoil mesh.

Campus-wide licensing assists different departments, schools and research groups in coming together for interdisciplinary studies. The multidisciplinary nature of research is probably the one trend that has most impacted engineering schools in recent years — and ANSYS has responded by creating seamless, unrestricted access and tool sharing via the license-bundling program. Academic teams can venture into new areas and study systems-level problems without incurring additional costs or navigating a cumbersome procurement process.

As academic customers worldwide see the advantages, campus-wide licensing is growing dramatically. For example, in Australia, ANSYS partner LEAP has helped virtually every leading university on that continent access the software via campus-wide licenses. More and more customers are expected to embrace license bundling as a solution to academic budget cuts, as well as in recognition of the software's ability to support collaborative, interdisciplinary study.
Researchers at National Taiwan University of Science and Technology used ANSYS Workbench to investigate different methods for lumbar fusion. Simulation helped to optimize implant procedures for outcomes such as range of motion, spinal stress and implant stress. Nonlinear contact conditions and tension-only springs were applied to simulate bone-implant interfaces and spinal ligaments.

Based on the dozens of schools behind the articles in this special edition of *ANSYS Advantage* — along with hundreds of other college highlights that could not fit into this magazine — it’s clear that engineering simulation has become an essential feature of the modern academic landscape. The overarching ANSYS goal is to ensure that the software can be found at every top engineering school, where it can help accelerate results, encourage collaboration, control costs and create a solid foundation for a new generation of engineering innovation.