HANDS-ON LEARNING AT THE UNIVERSITY OF IOWA

By *ANSYS Advantage* Staff

Undergraduate students, with a graduate teaching assistant, conduct CFD labs for Professor Stern's course, Mechanics of Fluids and Transport Processes.

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 [1]. This approach, and a series of teaching modules created by Stern and his team, form the foundation

Students gain a practical introduction to Fluent that ensures they will become expert users of simulation software — which is a requirement in today's technology-intensive engineering environment.

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. My students can quickly implement practical engineering applications without learning computer programming."
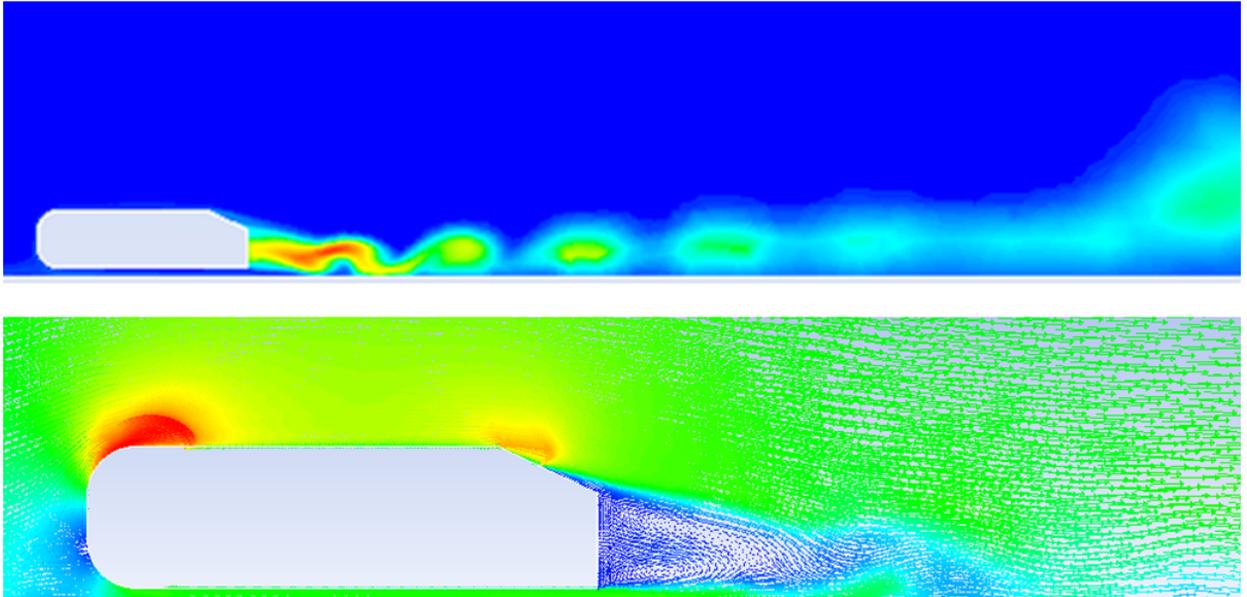
Using a customized ANSYS Fluent teaching approach created by Professor Frederick Stern, undergraduates at the University of Iowa create complex CFD models, such as this unsteady simulation of turbulent flow over a generic slant-back car. The top figure shows contours of turbulent viscosity ratio visualizing the wake flow, and the bottom figure shows velocity vectors colored by axial velocity.

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.

"My approach is a practical one, because it replicates real-world engineering practice: geometry, boundary definition, physics, mesh specification, solution procedure, reporting and post-processing," explains Stern. "The process automation I have created eases students from an introductory level of knowledge to the kind of expert knowledge of ANSYS Fluent they will need as practicing engineers after graduation."

Stern's customized educational approach has proven popular with students at the University of Iowa and has also been adopted by other universities.

"Engineering students like to begin working in a hands-on manner and seeing solution results as quickly as possible. By automating a series of complex process steps, the ANSYS Fluent educational approach turns them into working CFD engineers very quickly," notes Stern. "Simultaneously, they are gaining a practical introduction to ANSYS Fluent that ensures they will become expert users of simulation software — which is a requirement in today's technology-intensive engineering environment." ∧

References
[1] The Naval Engineering Education Center at the University of Michigan, under the administration of Professor Steven Ceccio, provided partial support for Timur Dogan and Michael Conger.

Stern, F.; Yoon, H.; Yarbrough, D.; Okay, M.; Oztekin, U.; Roszelle, B. Hands-On Integrated CFD Educational Interface for Introductory Fluids Mechanics: Invited Paper. *International Journal Aerodynamics*, **2012**, Vol. 2 Nos. 2/3/4, pp. 339–371.

Stern, F.; Xing, T.; Muste, M.; Yarbrough, D.; Rothmayer, A.; Rajagopalan, G.; Caughey, D.; Bhaskaran, R.; Smith. S.; Hutching, B.; Moeykens, S. Integration of Simulation Technology into Undergraduate Engineering Courses and Laboratories. *International Journal Learning Technology*, **2006**, Vol. 2, No. 1, pp. 28–48.

Stern, F.; Xing, T.; Yarbrough, D.; Rothmayer, A.; Rajagopalan, G.; Otta, S.P.; Caughey, D.; Bhaskaran, R.; Smith. S.; Hutching, B.; Moeykens, S., Hands-On CFD Educational Interface for Engineering Courses and Laboratories. *Journal Engineering Education*, **2006**, Vol. 95, No. 1, pp. 63–83.

Making the Grade
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 ground-breaking academic research that has the potential to shape the future.