# Teaching Simulation to Future Engineers

ANSYS Workbench facilitates integration of simulation into the engineering curriculum.

By Rajesh Bhaskaran, Swanson Director of Engineering Simulation, Cornell University, Ithaca, U.S.A.

Cornell University is highly regarded in the engineering community: Its College of Engineering has been ranked as one of the top seven engineering programs in the United States by *U.S. News & World Report*. Cornell uses software from ANSYS to deploy modern simulation technology in four undergraduate courses in the mechanical and aerospace engineering (M&AE) curricula. The courses cover diverse subject areas such as fluid dynamics, heat transfer, solid mechanics and numerical methods. These offerings include required, lab-based classes as well as elective, lecture-based classes.

Cornell's M&AE department is transitioning to the ANSYS Workbench platform so students can employ the same user interface and workflow regardless of subject area. This transition will create a uniform learning experience across the M&AE curricula. Other advantages related to ANSYS Workbench include a more user-friendly interface, easier CAD imports and automation of repetitive tasks. The transition is occurring on a course-by-course basis and is expected to be completed by spring 2011.
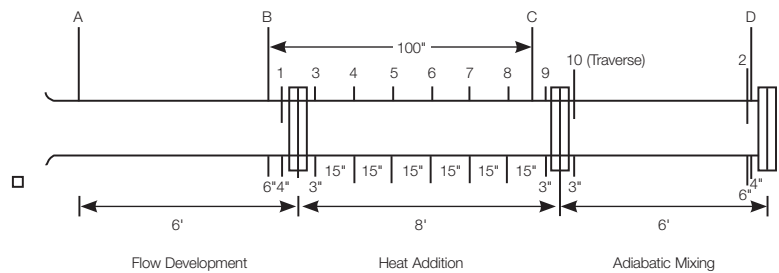
Schematic representation of the heated flow apparatus used in *M&AE 4272 Fluids/Heat Transfer Lab* at Cornell. A, B, C and D are pressure taps; labels 1 to 10 are locations of thermocouples.

In *M&AE 4272 Fluids/Heat Transfer Lab*, Cornell students use ANSYS Academic software to simulate a forced convection experiment consisting of three sections of a smooth-walled circular pipe. The first pipe section is unheated and generates a fully developed turbulent velocity profile; the middle section is heated to provide energy input to the air; and the final section is insulated, providing an adiabatic mixing length that enables a single reading of the final air temperature. Instruments provide measurements of mass flow rate and heat input, as well as pressure and temperature at various locations. A fluid dynamics simulation corresponding to this experiment was introduced in 2003 so students could see the experimental and simulated approaches side by side. FLUENT/FLOWLAB software was used for this until 2009, when it was replaced with the ANSYS Academic CFD product, which now fully integrates ANSYS FLUENT software.

The incorporation of ANSYS FLUENT and ANSYS Workbench into the ANSYS Academic CFD product has allowed Cornell to replace FLUENT/FLOWLAB technology while retaining important features that contribute to a rapid learning curve and ease of use. Students learn the simulation process via web-based tutorials that use nominal data. Students then repeat the calculations using their own experimental conditions. The geometry creation and mesh steps are optional, and the solve step is simplified using default settings. This process provides students with an adequate solution to the simulations while allowing quick turnaround time.

Course Title	Type	Approximate Enrollment	Software
Mechanical Property & Performance Lab	Required	120	ANSYS Mechanical 11.0
Fluids/Heat Transfer Lab	Required	120	ANSYS FLUENT 12.0
Finite Element Analysis	Elective	40	ANSYS Mechanical 11.0
Intermediate Fluid Dynamics	Elective	50	FLUENT 6.3

2010 courses at Cornell involving software from ANSYS

Students review background information on the simulation process through a presentation while waiting for the experiment apparatus to reach steady state. They learn that in the fluid flow simulation, ANSYS FLUENT software solves a boundary value problem (BVP) corresponding to the experimental setup, building on BVP concepts considered in earlier courses. The presentation asks students to consider what the governing equations, boundary conditions and domain should be for this BVP. Students learn how to specify the BVP in the software via the web-based tutorial, which provides a high-level conceptual description of discretization, iterative convergence and Reynolds averaging of the governing equations.

The fluid dynamics simulation provides rich visualization capabilities that are extremely useful in developing a physical understanding of fluid flows. The result plots show the axisymmetric temperature field in the pipe, demonstrating how the heat added at the wall diffuses into the
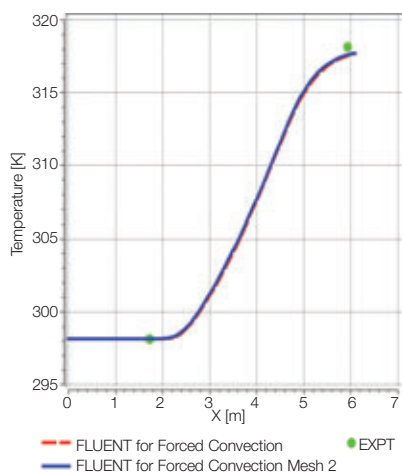
Temperature variation along the pipe axis obtained from CFD and experiment

Temperature distribution in the pipe obtained using ANSYS FLUENT software
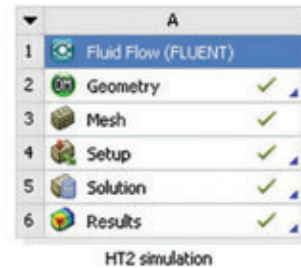
Steps in the tutorial (left) are a superset of the project schematic in ANSYS Workbench (right).

flow. Students can see that the flow is well mixed at the end of the adiabatic mixing section, which confirms an assumption in the experiment. The comparison between experiment and simulation for the temperature variation along the pipe axis can be plotted along with results from the original and refined meshes. The final bulk temperature rise obtained from both experiment and simulation agrees well.

The steps in the tutorial are a superset of the project schematic in the ANSYS Workbench environment. A verification and validation (V&V) step follows the results step to reinforce the importance of checking computational results through a formal process. The verification step considers the question: "Did we solve the BVP correctly?" In this introduction to fluid dynamics, students are required only to refine the mesh for the simulation. The validation step considers the question: "Did we solve the right BVP?" Students check this by comparing their simulation results with experiment. Similar web-based tutorials are used in the other M&AE undergraduate courses; V&V is the final step in each tutorial. Cornell plans to incorporate the latest V&V thinking into the tutorials and to introduce students to related standards from ASME and NAFEMS. In this way, V&V can be given a more prominent and consistent role in the curriculum.

There is a concern in academia that the overhead of learning to use commercial simulation products is high, and this prevents integration of such tools into the curriculum. Cornell has been successfully addressing this concern with the help of software interface improvements offered by the ANSYS Workbench environment. These include the integration of structural mechanics and fluid flow simulation tools in a common intuitive interface and the automation of repetitive tasks; the tools are augmented by carefully designed online tutorials that teach software use and problem-solving skills.

In transitioning other courses to the ANSYS Workbench platform, Cornell plans to use the same tutorial structure previously described, expanding on the uniform structure offered by ANSYS Workbench across application areas. Students will be able to see that the same high-level principles apply whether solving solid mechanics or thermal–fluids problems. Instructors at Cornell expect that if students see the same solution process being followed repeatedly in different subject areas, they are more likely to internalize this approach and to apply it in new situations. ANSYS Workbench is helping Cornell to provide students with a good formative experience in simulation across the M&AE curriculum, one that undergraduates can carry into their careers as practicing engineers. ■