Working at the Frontiers of CFD Simulation

As they push the boundaries of engineering simulation, academic research teams must meet exacting technology standards.

By Gilles Eggenspieler, Senior Product Manager, ANSYS, Inc.

While leading-edge CFD simulations occur in many industries and applications, universities are at the forefront of some of today's most exciting engineering innovations. From automotive and aeronautic engineering to biomedical applications, the work of academic researchers is not only inspiring other engineers, but ensuring that CFD software keeps pace with their needs. University research teams insist on trusted performance and continuous innovation, and they choose their CFD simulation software based on an exacting set of standards.

Ensuring Technology Breadth, Depth and Reliability

Often, university teams require exceptional depth in physical modeling (such as turbulence, chemistry, particulates and real fluid properties), as well as physics breadth (including structural mechanics, fluid dynamics and thermal radiation) and coupling (for example, fluid–structure interaction or thermal–electric models). Many applications also require transient analysis of moving parts. An integrated environment that supports multiphysics simulations is critical.

CFD solutions must produce accurate results. Greater fidelity allows researchers to explore increasingly realistic designs, which contribute greatly to making important decisions. Essential to improved accuracy are advances in numerical algorithms and mainstream use of higher-order discretization methods.

Because pioneering work is often funded by competitive grants and third-party sources, university teams must ensure that their CFD software is ISO certified and subject to the most stringent quality assurance standards.

Maximizing HPC and Other Investments

Simulation software is just one component of a university's investment in pioneering CFD research, but it has the power to maximize performance of the entire laboratory. A sub-par software solution will negatively affect overall deliverables. In contrast, a high-quality solution will energize the team, leading to faster, higher-fidelity, more accurate results that minimize the need for physical tests that consume precious time and resources.

To fully leverage academic high-performance computing (HPC) resources, CFD software needs the scalability to run on thousands of cores, the ability to simulate up to a billion cells, physics-based load-handling capabilities, and parallel file reading and writing capabilities that deliver near-linear scaleup in processing speed for numerically large simulations.

The Value of Expert Support

To keep pace with universities, technology solutions must be backed not only by an expert development team that continually shapes future innovations, but also by an expert support team that helps extend solution capabilities to meet groundbreaking new applications.

Training is a critical aspect of support for university research teams. These groups often undergo high annual staff turnover, as experienced researchers graduate and new team members enter the program. Instead of relying on limited internal resources, university teams should be able to rely on their trusted solution provider to ensure that their newest engineers are leveraging the full capabilities of CFD solutions in their daily work.

The Power of Customization

Finally, CFD solutions must be readily customizable to meet researchers' evolving simulation and work flow requirements both now and in the future. Users should be able to easily add new physical models, create additional solution variables, define property relationships, set boundary conditions and profiles, and perform other critical customization tasks. Solution providers should create extensions of software to meet unique modeling requirements as well as develop automated tools that streamline setup, solution and visualization of frequently repeated analyses.

By offering expert customization services, solution providers allow both academic teams and other customers to focus on core research, rather than on routine programming and maintenance tasks.

Taking a Partnership Approach

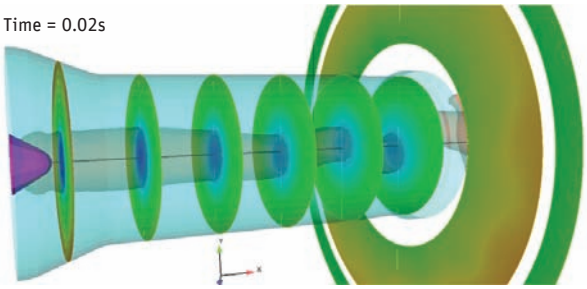
Perhaps the most important selection criteria for university teams is identifying a solution provider who will act as a true partner in their exacting CFD research. A solution provider who will be available to answer questions, customize technology, provide custom-tailored training and otherwise assist in pushing the boundaries of CFD simulation is an invaluable catalyst to any leading-edge engineering endeavor — academic or commercial.

Every engineering team that engages in CFD simulation can look to academic researchers as role models for taking a true partnership approach with their solutions provider. By forming close, collaborative relationships, all engineering teams can accelerate their simulation efforts and maximize investments in CFD solutions.

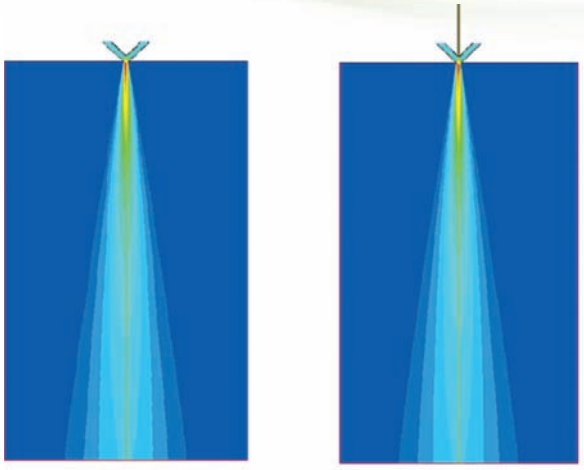
Cases in Point: ANSYS at Work in Leading Universities around the World

University of Southampton, U.K.

Neil Bressloff has used ANSYS software for more than 15 years to solve a range of CFD problems — from the aerodynamic design of Formula 1™ race cars to biological flows. His Computational Engineering and Design research group has selected ANSYS technology as its tool of choice for Ph.D. students because of its versatility in addressing all the complex CFD challenges these students encounter in their diverse analyses. "ANSYS provides us with an integrated environment so we no longer have to rely on separate software packages or legacy codes," said Bressloff. He and his student researchers also benefit from dynamic meshing capabilities. "For a recent biomedical project, we modeled the flow of fluid in the gap that opens and closes adjacent to a hip implant. As the domain changes, ANSYS CFD automatically adapts, changing the movement and shape of the model. This capability provides the means to efficiently and accurately simulate this challenging problem." A longtime ANSYS user, Bressloff is confident in his observation that the solution continues to improve over time. "The software has grown to accommodate the kinds of large problems we solve on a daily basis, including turbulent combustion, which is one of the most complex CFD modeling challenges. ANSYS technology has the power and capability to create comprehensive models, as well as the parallelization and speedup to solve relatively quickly, even for numerically large simulations," Bressloff said.



Bressloff's research group is using ANSYS Fluent software to study the design of a modern lean-burn combustor for propulsion applications, using a multi-swirler fuel injector system under partially premixed combustion conditions. Shown here is the central vortex core structure inside the injector's air-blast atomizer.



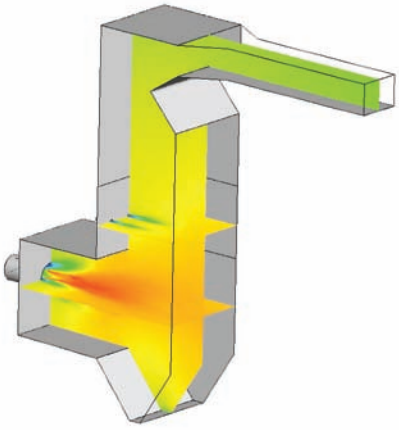
Pourdeyhimi's research group uses ANSYS Fluent software to model air flow fields during melt-blowing production processes for nonwoven fibers. During melt blowing, molten polymer streams are injected into high-velocity gas jets, and polymer fibers are attenuated by the resulting drag force.

North Carolina State University, U.S.A.

Behnam Pourdeyhimi has been using ANSYS software for more than 12 years in his work leading the Nonwovens Cooperative Research Center. Pourdeyhimi relies on this technology to deliver large-scale, high-fidelity modeling capabilities to study fiber entangling, absorbency and fluid flow, material quenching, particle and liquid filtration, and other complex CFD issues related to manufacturing nonwoven fabrics. "From water jet modeling to multiphase flows, the software offers everything we need in a single tool that is very convenient to use," said Pourdeyhimi. He also values the high level of support his team receives. "When you are engaged in pushing the performance envelope in 3-D modeling, there are frequent questions that arise — and ANSYS is very responsive to our needs," he added. "We have also turned to ANSYS to train our student researchers in emerging topics such as HPC-enabled simulations, and we have been very pleased with the results." Pourdeyhimi has another, more practical reason for choosing ANSYS: "We work with a consortium of more than 70 companies that have a vested interest in the nonwovens industry, and the vast majority of consortium members are using ANSYS solutions. ANSYS has emerged as the industry standard in CFD simulation."

University of Leeds, U.K.

Mohamed Pourkashanian's team uses ANSYS fluid dynamics solutions daily in the Centre for Computational Fluid Dynamics. There, researchers study CFD problems ranging from aero engines and fuels to advanced power generation. "We choose ANSYS solutions because their quality assurance and control processes enable us to meet rigorous quality standards in our own work," said Pourkashanian. "I would estimate that, by enabling us to confidently replace physical testing with engineering simulation, our partnership has saved millions of dollars in time and resources." Complex models of jet engines, fuel chemistry and combustion, power generation, and other challenging problems require Pourkashanian and his team to customize the software to a certain extent, with excellent results. "We have been very impressed with the way ANSYS has supported our needs for originality and customization. It is very easy to develop new physical models and then move them into ANSYS solutions. This approach has truly represented 'the best of both worlds' by combining the ease and reliability of a commercial product with a customized in-house solution," said Pourkashanian. ■



Pourkashanian and his team used engineering simulation software to predict oxy-coal combustion in an industrial combustion test facility with recycled flue gas to address the major technical challenges of using this system for power generation with carbon capture and sequestration.