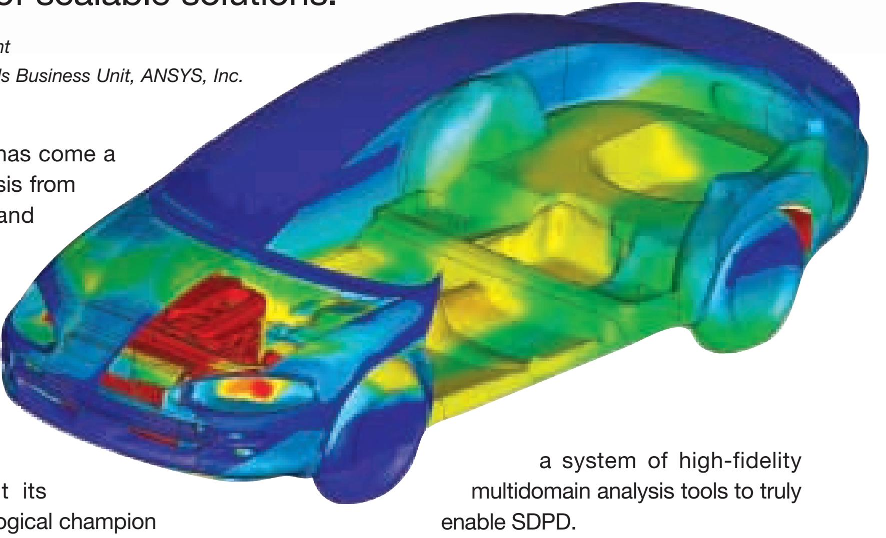
The New Wave of **Fluids Technology**

Fluid flow simulation software from ANSYS provides a broad range of scalable solutions.

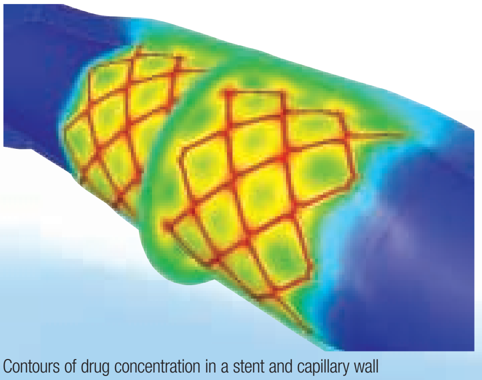
By Paul Galpin, Director of Product Management
and André Bakker, Lead Product Manager, Fluids Business Unit, ANSYS, Inc.



Contours of temperature on a car body calculated in FLUENT software

The world of product engineering has come a long way in its quest to advance analysis from laborious hand-drawn sketches and simplistic models to virtual computer-created models initiated at the touch of a button. There has been a long evolutionary path from the inception of computational fluid dynamics (CFD) to today's integration of this technology into Simulation Driven Product Development (SDPD) processes. Throughout its history, ANSYS, Inc. has been a technological champion for such commercial engineering simulation. The company has viewed simulation as the key to predicting how products will perform; it has enabled the rapid comparison of many different alternatives prior to making a design decision — well before customers might identify problems. ANSYS now has a fluids product line that is both broad and deep, along with a large commercial and academic user base that is reaping the benefits.

This CFD evolution has required, and continues to demand, that ANSYS go beyond merely providing advanced mathematical flow solvers. ANSYS espouses a multiple physics approach to simulation in which fluid flow models integrate with other types of physics simulation technologies. The ANSYS vision is clear: to provide a system of high-fidelity multidomain analysis tools to truly enable SDPD.



Contours of drug concentration in a stent and capillary wall

SDPD is centered on the highly adaptive ANSYS Workbench architecture. The next major ANSYS Workbench release will provide another big step toward this vision. It will be the first release in which a number of the original Fluent CFD products will be data-integrated into the ANSYS Workbench platform, and thus the tools will work together with various other applications from ANSYS.

The ANSYS Workbench approach allows ANSYS to provide a large variety of software choices tailored to meet individual needs while ensuring interoperability and a clear future upgrade path. This includes a very broad fluids product line with all tools falling into one of three categories: general-purpose fluid flow analysis, rapid flow modeling and industry-specific products.

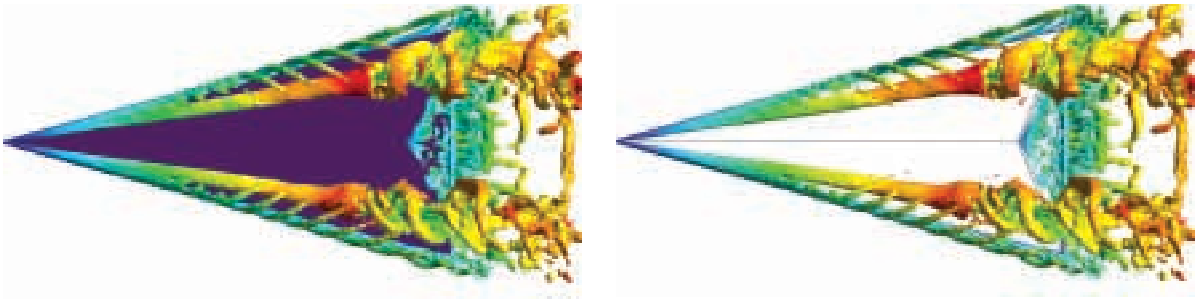
General-Purpose Fluids Solvers

The well-known FLUENT and ANSYS CFX products are the main general-purpose CFD tools from ANSYS. These two solvers, developed independently over decades, have a lot of things in common but also some significant differences. Both are control volume–based for high accuracy and rely heavily on a pressure-based solution technique for broad applicability. They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies.

The ANSYS CFX solver uses finite elements (cell vertex numerics) to discretize the domain, similar to those used in the mechanical analysis side of the business. In contrast, the FLUENT product uses finite volumes (cell-centered numerics). Ultimately, though, both approaches form "control volume" equations that ensure exact conservation of flow quantities, a vital property for accurate CFD simulations. ANSYS CFX software focuses on one approach to solve the governing equations of motion (coupled algebraic multigrid), while the FLUENT solver offers several solution approaches (density-, segregated- and coupled pressure-based methods). Both solvers contain a wealth of physical modeling capabilities to ensure that any fluids simulation has all of the modeling fidelity required.

The ANSYS CFX-Flo tool is a version of ANSYS CFX software that limits the physics accessible by the user to the models most commonly used by design engineers. It is compatible with other applicable ANSYS Workbench add-ins. The reduced complexity and cost of ANSYS CFX-Flo make it a good choice for design departments in organizations that already use ANSYS CFX software or other products compatible with the ANSYS Workbench environment.

The FLUENT for CATIA V5 product offers many of the same benefits of FloWizard and ANSYS CFX-Flo software. Completely embedded into the CATIA V5 system, it is fully compatible with the standard, full FLUENT solver. It is most useful for companies that use CATIA V5 in their design departments and FLUENT software in their analysis groups.



Turbulence on a delta wing calculated by ANSYS CFX software

These two core CFD solvers represent more than 1,000 person-years of research and development. This effort translates into the key benefits of fluid flow analysis software from ANSYS: experience, trust, depth and breadth. The fluids core solvers from ANSYS are trusted, used and relied upon by companies worldwide.

Rapid Flow Modeling

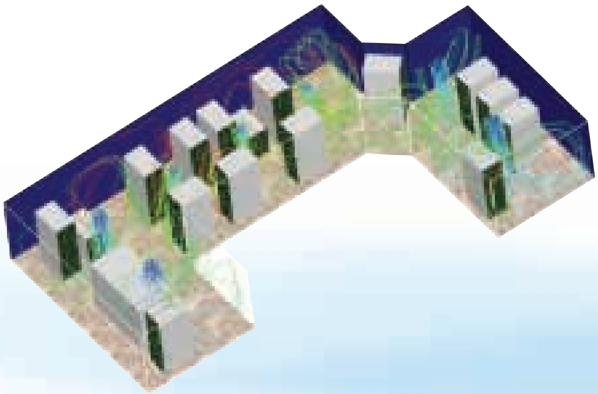
ANSYS addresses the fluid flow analysis needs of designers, who work on the front lines of their company's product development process and often need to make important design decisions quickly with no time to set up and solve complex mathematical models. For these time-limited engineers, ANSYS offers a choice of rapid flow modeling (RFM) products. RFM technology from ANSYS compresses the overall time it takes to do a fluid flow analysis by providing a high level of automation and focusing on only the most robust physical models. Three RFM tools are available: FloWizard, ANSYS CFX-Flo and FLUENT for CATIA® V5 software.

FloWizard software integrates all steps in the fluids process into one smooth interface. Computer-aided design (CAD) files can be sent to the FloWizard product, flow volumes extracted, models set up, calculations completed and HTML reports generated. FloWizard software is fully compatible with the FLUENT product, making it a good choice for designers in companies that use FLUENT technology in the analysis department.

Industry-Specific Fluids Simulation Tools

Flexibility and generality are important, but sometimes not required for specific applications. In addition to providing general-purpose CFD and rapid flow modeling products, ANSYS makes fluids simulation even more accessible and focused with its industry-specific analysis tools. These products are often called vertical applications because of the way they integrate all the steps for the analysis of a specific type of system into one package. The technologies offer industry-specific functions as well as employ the language of the industry in which they are used.

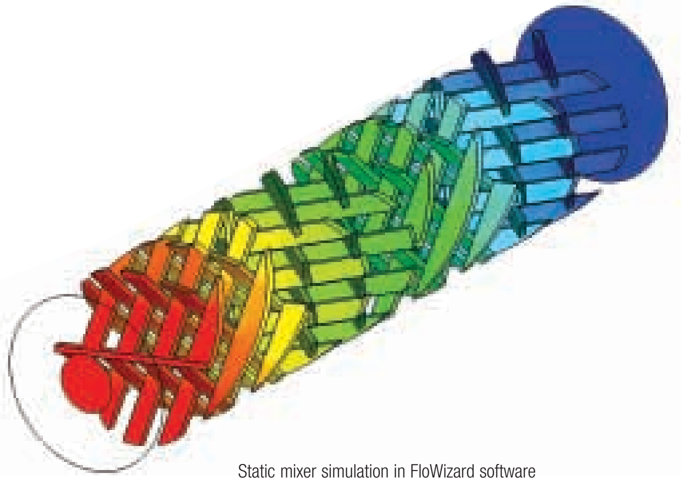
Turbomachinery is one of the world's single most successful CFD vertical applications, due to the similarity of the geometry and physics across a broad range of rotating



Airflow in a datacenter as simulated using ANSYS Airpak software machinery sectors. The turbosystem technology from ANSYS includes custom geometry and meshing tools as well as special modes within the general-purpose fluids simulation tools.

The ANSYS Icepak product is a family of applications focused on electronics design and packaging. In order to improve the performance and durability of electronic boards and other components for optimized cooling systems, the product calculates the flow field and temperatures in electronics and computer systems.

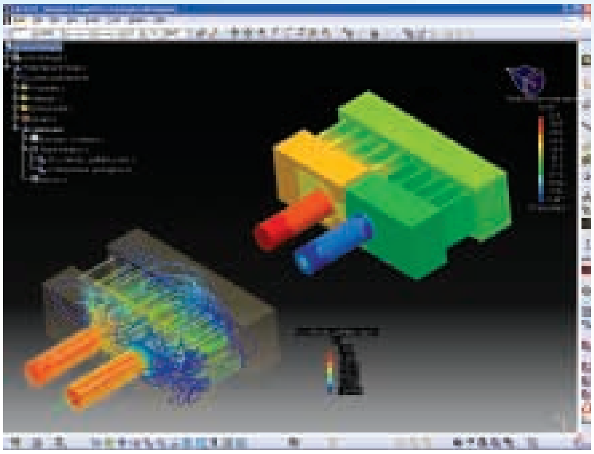
ANSYS POLYFLOW software is focused on the needs of the materials industry, such as polymer processing, extrusion, filmcasting and glass production. It can model the flow of fluids with very complex behavior, such as viscoelastic fluids. The ANSYS POLYFLOW product offers unique features such as the ability to perform reverse calculations to determine the required die shapes in extrusion. It also can calculate the final wall thickness in blow-molding and thermoforming processes.



Static mixer simulation in FloWizard software

The ANSYS Airpak product is aimed at the design of heating, ventilation and cooling systems in buildings, such as offices, factories, stadiums and other large public spaces. It accurately and easily models airflow, heat transfer, contaminant transport and thermal comfort in a ventilation system.

Finally, end-users can create their own vertical applications within the general-purpose fluids simulation products: ANSYS CFX software offers user-configurable setup wizards and expression language; FLUENT technology provides user-defined functions; and the FloWizard tool offers Python scripts. All of these can be used to create custom vertical applications. It is not uncommon for an analysis department to create such vertical applications for deployment within a design department. The main benefit of this approach is to ensure repeatable simulation process control, and hence quality control, for any CFD process.



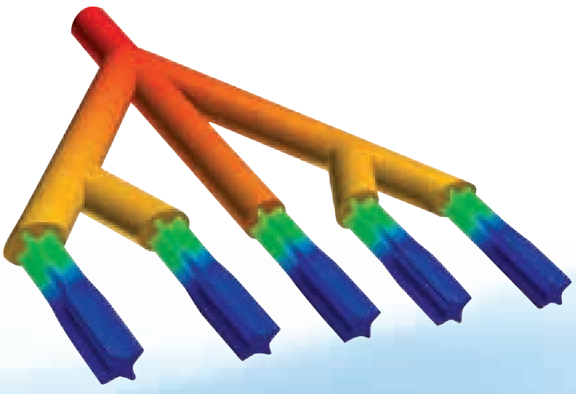
FLUENT for CATIA V5 software works within the CATIA V5 PLM environment, as shown in this simulation of a heat exchanger.

The Future of Fluids Simulation from ANSYS

To help customers replace more and more of their traditional capital-intensive design processes with a Simulation Driven Product Development method, ANSYS will continue to innovate and integrate.

In the very near future, users will see tremendous progress toward the ANSYS integration vision, including common geometry, meshing and post-processing tools for all users of CFD products from ANSYS. Many steps in the fluids simulation process will be automatically recorded, enabling parametric simulations. Improvements in fluid–solid connectivity will be evident, enabling a number of new multiphysics possibilities.

The upcoming ANSYS 12.0 release will lay a firm foundation for the future while carefully preserving and extending current software value. Over time, ANSYS plans to achieve the tightest possible integration of all its fluids technologies as well as an intimate integration with ANSYS mechanical technologies. The goal is to combine the best of the best into a simulation system with unprecedented power and flexibility. ■



The extrusion of a viscoelastic food material is simulated with ANSYS POLYFLOW software. The pressure drop between the inlet and the five outlets is shown. The outlet shape is computed as part of the analysis.