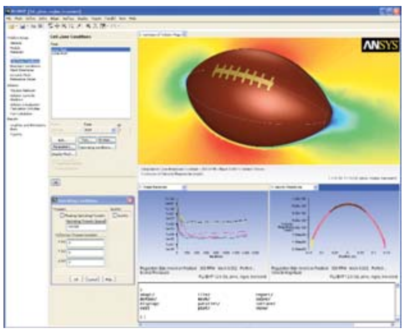
ANSYS®

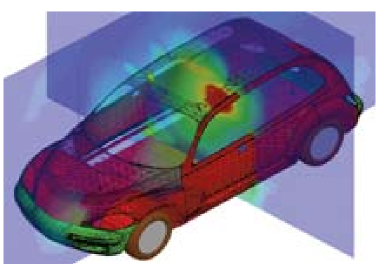
12.0

12.0 RELEASE

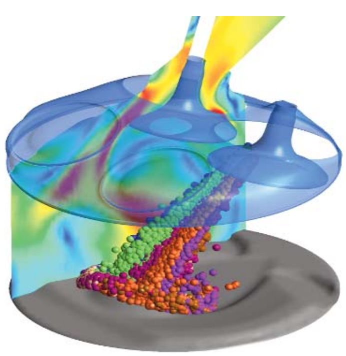
Fluid Analysis Solutions



ANSYS FLUENT software is integrated into ANSYS Workbench for efficient design optimization. This model shows the flow around a football.



Contours of temperature on a car body calculated in ANSYS FLUENT technology



Automotive in-cylinder flow modeled with ANSYS CFX software

www.ansys.com

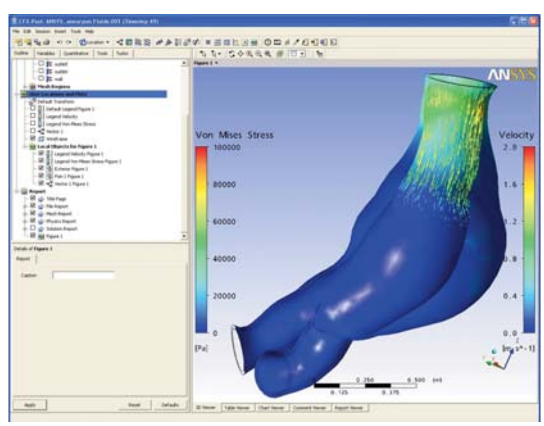
Computational fluid dynamics (CFD) is an engineering method in which flow fields and other physics are calculated in detail for an application of interest. The CFD, or fluid simulation, results can be used as part of a Simulation Driven Product Development™ (SDPD) process to illustrate how a product or process operates, to troubleshoot problems, to optimize performance and to design new products.

With a large commercial and academic user base, along with a fluid simulation product line that is both broad and deep, ANSYS, Inc. is an industry leader and technological champion for commercial engineering simulation.

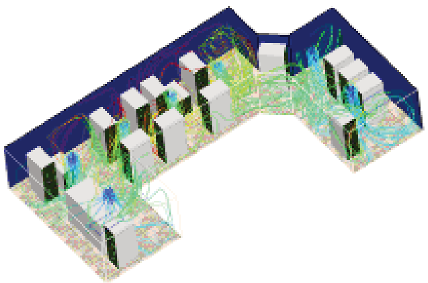
As with most advanced technologies, there has been a long evolutionary path from the inception of CFD to today's integration of this technology into SDPD processes. This progression occurred as ANSYS moved beyond providing advanced mathematical flow solvers. ANSYS now uses a multiphysics approach to simulation in which fluid flow models integrate seamlessly with other types of physics simulation technologies. The ANSYS vision is clear: to provide a system of high-fidelity multiphysics analysis tools to truly enable SDPD using the adaptive and unified ANSYS® Workbench™ architecture. The ANSYS Workbench platform integrates a large variety of technology choices tailored to meet individual needs, while ensuring interoperability and a clear future upgrade path. This includes a very broad fluid simulation product line from ANSYS, with the products falling into two categories: general-purpose fluid flow analysis and specialty products. The breadth and depth of the ANSYS fluid dynamics portfolio is unparalleled.

General-Purpose CFD Solvers

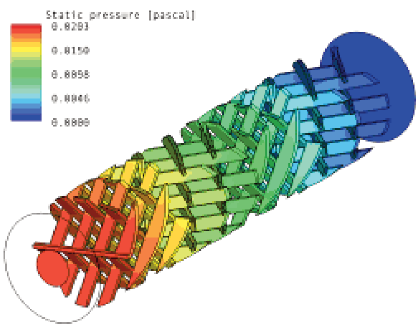
ANSYS CFD technology provides access to both the well-known ANSYS® FLUENT® and ANSYS® CFX® products. Also available separately, these are the main general-purpose fluid simulation products offered by ANSYS. These two solvers were developed independently over decades and have a number of things in common, but they also have 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.



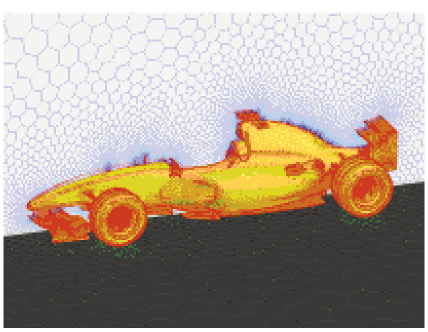
ANSYS CFX software is fully integrated in ANSYS Workbench. This biomedical aneurysm model shows the stresses in an artery wall.



Airflow in a data center calculated by ANSYS® Airpak® software



Static mixer simulation in ANSYS fluid dynamics software



Polyhedral mesh of a race car as used with ANSYS FLUENT software. The use of polyhedral meshes speeds up many simulations.

The ANSYS CFX solver uses finite elements (cell vertex numerics), similar to those used in structural analysis, to discretize the domain. In contrast, the ANSYS FLUENT solver 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 ANSYS FLUENT product offers several solution approaches (density-based, and both segregated and coupled pressure-based methods). Both solvers contain a wealth of physical modeling capabilities to help ensure that any fluid dynamics simulation has all of the modeling fidelity required.

Furthermore, ANSYS® CFD-Flo™ software addresses the fluid flow analysis needs of designers who work on the front lines of their company's product-development process. The ANSYS CFD-Flo tool 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 CFD-Flo software make it a good choice for design departments.

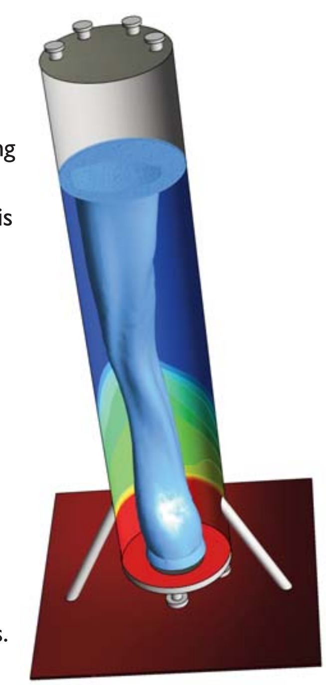
ANSYS CFD software is highly scalable. Product development times can be shortened with ANSYS CFD HPC, the high-performance computing option. This allows performing large model calculations on parallel computing clusters. Linear scalability has been shown on systems with more than 1,000 processors.

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

Specialty Fluid Analysis Tools

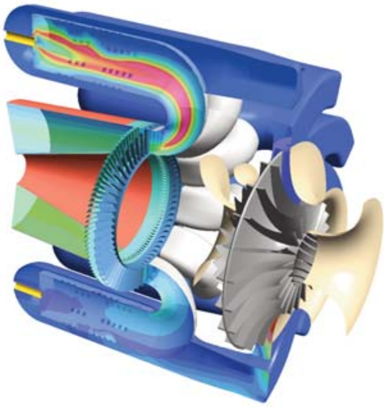
Flexibility and generality are important, but sometimes not required for certain specific applications. In addition to providing general-purpose fluid simulation, ANSYS makes fluid simulation even more accessible and focused with its specialty fluid analysis tools. These products are often called vertical 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 fluid dynamics vertical applications — due to the similarity of the geometry and physics across a broad range of rotating machinery sectors. Turbosystem technology from ANSYS includes custom geometry and meshing tools as well as special modes within the general-purpose fluid simulation tools.



Multiphase mass fractions in a three-phase bubble column as analyzed with ANSYS FLUENT software

ANSYS® Icepak® software is a family of products focused on electronics design and packaging. In order to improve the performance and durability of electronic boards and other



ANSYS CFX is commonly used to model complex turbomachinery.

components during the design of 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, mold-filling, thermoforming and glass production. It can model the flow of fluids with very complex behavior, such as viscoelastic fluids. ANSYS POLYFLOW technology 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 processes.

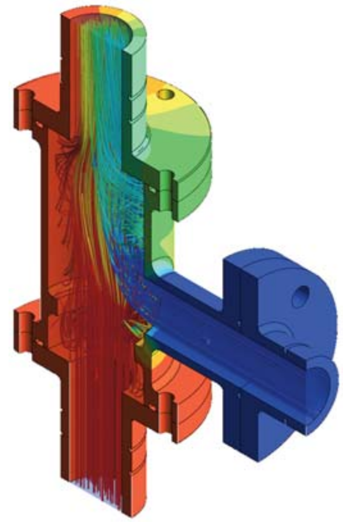
The ANSYS Airpak product is aimed at the design of heating, ventilation and cooling systems in buildings such as offices, factories, stadiums and other large public spaces. It accurately and easily models airflow, heat transfer, contaminant transport and thermal comfort in a ventilation system.

FLUENT® for CATIA® V5 software brings fluid flow and heat transfer analysis into the CATIA V5 product lifecycle management environment. The fluid flow physics available are most commonly used by design engineers. It is fully compatible with the ANSYS FLUENT product.

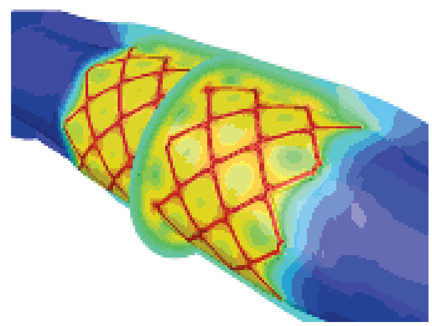
Finally, end users can create their own vertical applications within the general-purpose fluid simulation products. ANSYS CFX software offers user-configurable setup wizards and expression language. ANSYS FLUENT technology provides user-defined functions. Combined with the scripting tools available in the other ANSYS Workbench based applications, 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 fluid dynamics process.

The Benefits of Fluid Simulation from ANSYS

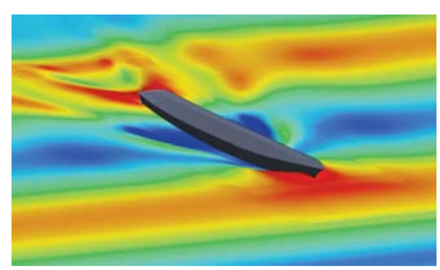
ANSYS fluid flow analysis technology allows for an in-depth analysis of the fluid mechanics in many types of products and processes. Not only does it reduce the need for expensive prototypes, it provides comprehensive data that is not easily obtainable from experimental tests. Fluid simulation can be used to complement physical testing. Some designers use it to analyze new systems before deciding which validation tests, and how many, need to be performed. When troubleshooting, problems are solved faster and more reliably because fluid dynamics analysis highlights the root cause, not just the effect. When optimizing new equipment designs, many what-if scenarios can be analyzed in a short time. This can result in improved performance, reliability and product consistency. ANSYS will continue to innovate and integrate so that customers can replace more of their traditional capital-intensive design processes with a Simulation Driven Product Development method.



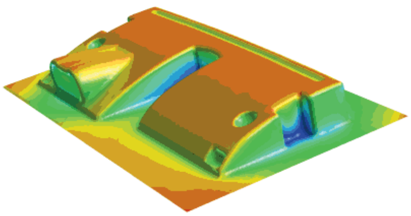
The fluid flow in a pipe-junction modeled with ANSYS CFX technology
Geometry courtesy of CADFEM



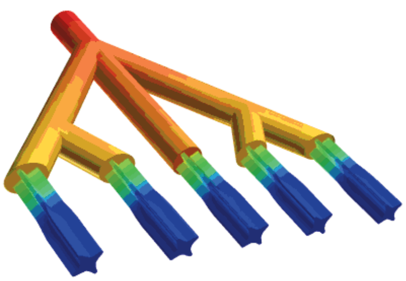
Contours of drug concentration in a stent and capillary wall as predicted by ANSYS FLUENT technology



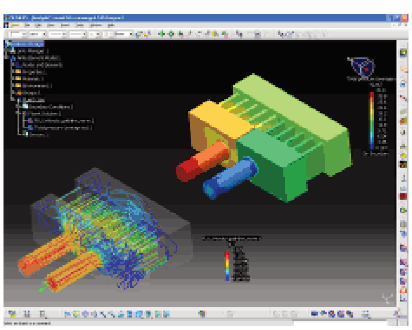
Wave formation around a ship hull modeled with ANSYS CFX software



ANSYS POLYFLOW is commonly used in thermoforming and blow-molding applications. This example shows the final thickness of a thermoformed dashboard component.



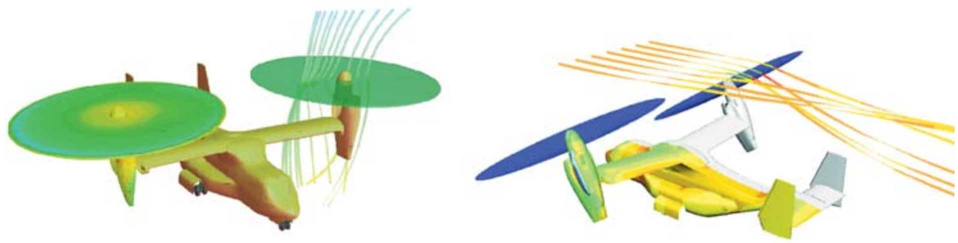
The extrusion of a viscoelastic food material is simulated with ANSYS POLYFLOW software. The pressure drop between the inlet and the five outlets is shown.



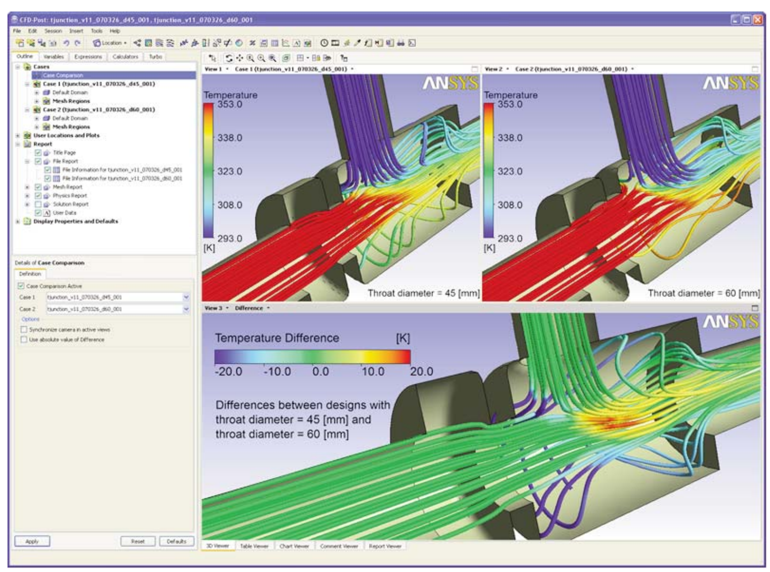
FLUENT for CATIA V5 software works within the CATIA V5 PLM environment, as shown in this simulation of a heat exchanger.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products from ANSYS and processes that work. Today, 97 of the top 100 industrial companies on the "*FORTUNE* Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of our solutions delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of almost 40 years as the industry leader — for the best in engineering simulation.



The moving mesh technology in ANSYS FLUENT software is used to model the changing tilt of the rotorcraft's wings while changing flight modes.



ANSYS CFD-Post provides powerful quantitative and graphical post-processing and reporting capabilities. The performance of different designs can be compared with ease, enabling efficient design optimization.

www.ansys.com

GSA Contract Holder

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com