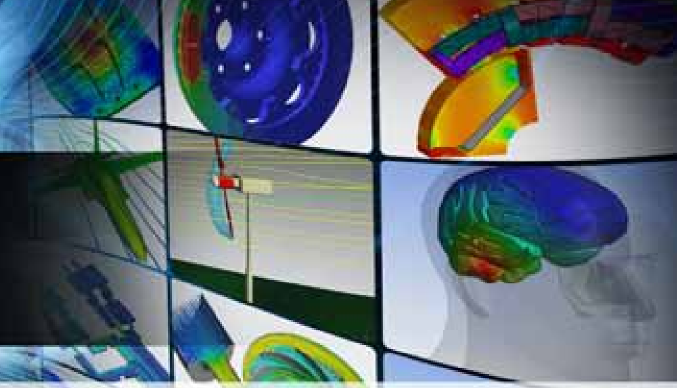
ANSYS® RELEASE 13.0



ANSYS CFD

Computational fluid dynamics (CFD) is an engineering method for simulating the behavior of systems, processes and equipment involving flow of gases and liquids, heat and mass transfer, chemical reactions and related physical phenomena. More specifically CFD, or fluid simulation, can be used to reduce pressure drops, to predict aerodynamic lift or drag, to predict the rotor thrust, to calculate the airflow in air conditioned rooms, to ensure adequate cooling, to optimize mixing rates, and so on.

ANSYS combines the most respected names in fluid simulation — ANSYS® FLUENT® and ANSYS® CFX® — to expertly address your evolving CFD needs at a time when product reliability, safety and market performance are paramount. ANSYS offers the most complete suite of advanced CFD software tools available, coupled with unrivaled modeling capabilities, to help you achieve a faster total time to solution. More product development leaders worldwide trust ANSYS software as their fluid dynamics simulation platform for its accuracy, reliability and speed.

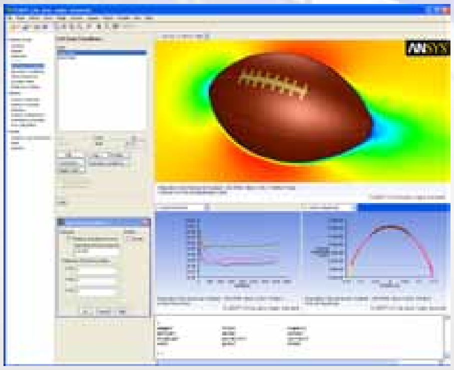
ANSYS is committed to providing world-class high-fidelity fluid dynamics technology for Simulation Driven Product Development. In addition to providing the most well validated and used CFD products on the market its ANSYS® Workbench™ platform is built on the ability to co-simulate. Users of ANSYS' CFD solutions can include structural mechanics or electrical aspects to a model through our other industry-leading solver solutions. By that means customers reduce engineering assumptions and increase the fidelity of their models.

CFD solutions from ANSYS deliver unprecedented productivity to help you analyze multiple, automated parametric design variations — all without complex programming. ANSYS provides integrated tools that not only help you to conveniently understand which parameters your design is most sensitive to, but also determine which design parameters require the tightest control. Integrated tools for design (shape) optimization and Six Sigma analysis are also available. ANSYS simulation software gives you the power to design for robust performance and deliver better products, faster.

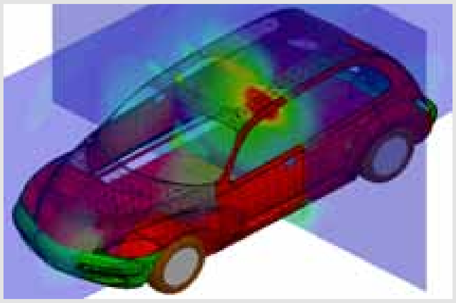
CFD solutions from ANSYS are readily customizable and extensible to meet your ongoing simulation and workflow process requirements - whether it's extending solver capabilities to predict unique environmental factors or customizing our products to automate your design work flow.

With ANSYS tools, your designers have the power to create better products more profitably. Because they can yield significant benefits (incl. more innovation, cost savings, reduced development time,

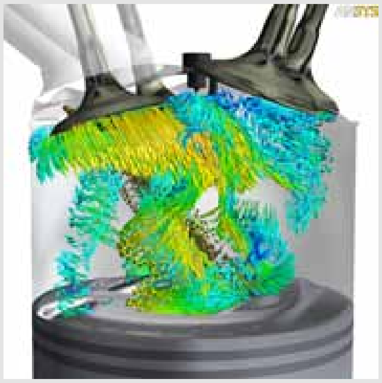
Industry Solutions



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



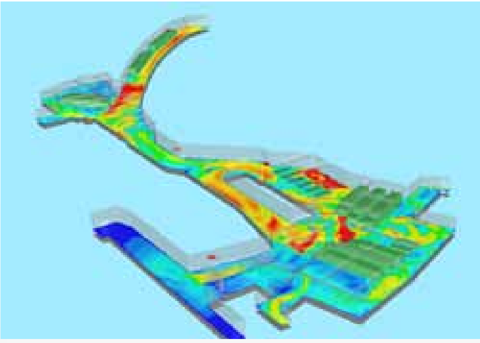
Contours of temperature on a car body calculated in ANSYS FLUENT software



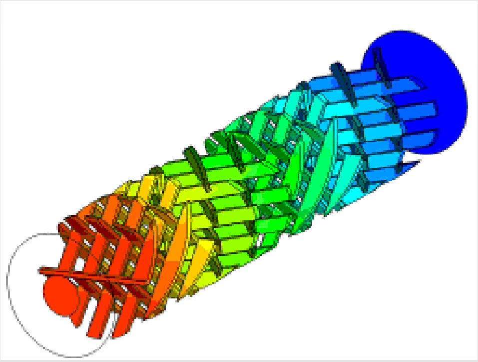
Automotive in-cylinder flow modeled with ANSYS CFX software

Courtesy of BMW AG.

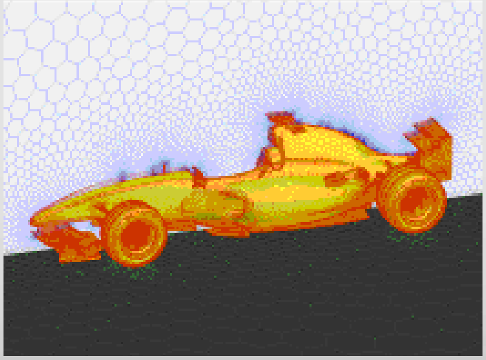
increased quality), they have become an integral part of the engineering design and analysis environment of companies in the widest range of industries.



Mechanical ventilation of an underground service area
Courtesy of Land Securities Group PLC.



Static mixer simulation



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

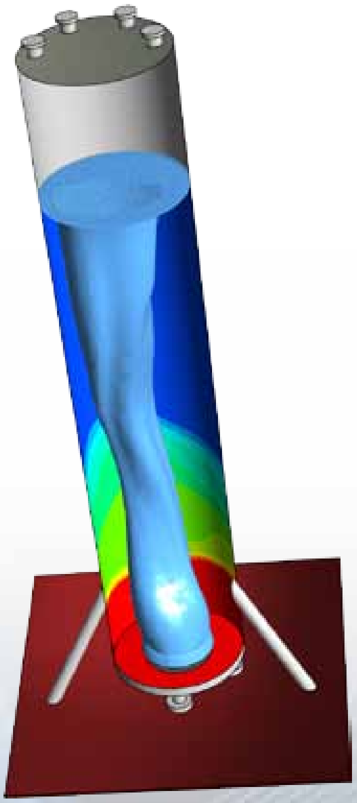
General-Purpose CFD Solvers

ANSYS fluid dynamics 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. The products differ mainly in the way they integrate fluid flow equations and in their equation solution strategies.

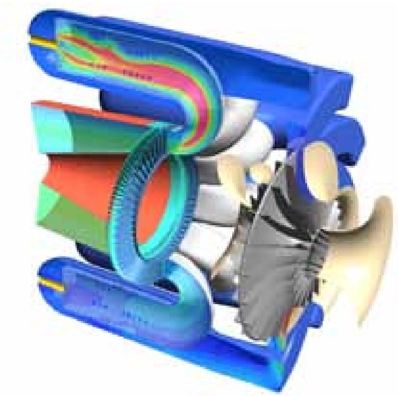
The ANSYS CFX solver uses finite elements (cell vertex numerics), similar to those used in structural analysis, to discretize the domain. In contrast, the ANSYS FLUENT solver uses finite volumes (cell-centered numerics). Ultimately, though, both approaches form control volume equations that ensure exact conservation of flow quantities, a vital property for accurate CFD simulations. ANSYS CFX focuses on one approach to solve the governing equations of motion (coupled algebraic multigrid), while ANSYS FLUENT offers several solution approaches (density-based as well as 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 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. Time to solution, and therefore product development times, can be shortened with ANSYS® 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.



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



ANSYS CFX is commonly used to model complex turbomachinery

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.

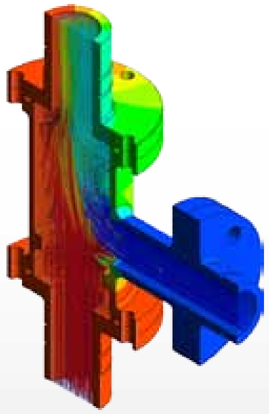
Turbomachinery is one of the world's single-most successful fluid dynamics vertical applications — due to the similarity of the geometry and physics across a broad range of rotating machinery sectors. Turbosystem technology from ANSYS includes custom geometry and meshing tools as well as special modes within the general-purpose fluid simulation tools. ANSYS® Icepak® software is a family of products focused on electronics design and packaging. To improve the performance and durability of electronic boards and other components during the design of optimized cooling systems, the product calculates the flow field and temperatures in electronics and computer systems.

ANSYS® POLYFLOW® software focuses 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 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.

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.

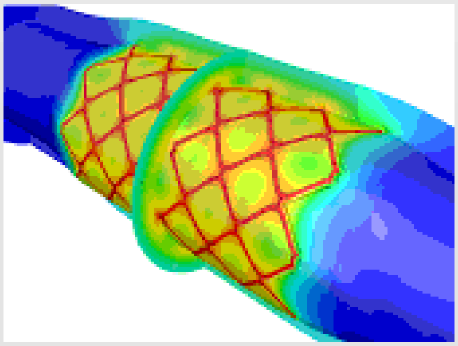
Finally, end users can create their own vertical applications within the general-purpose fluid simulation products. ANSYS CFX offers user-configurable setup wizards and expression language. ANSYS FLUENT provides user-defined functions. Combined with the scripting tools available in the other ANSYS Workbench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

Industry Solutions

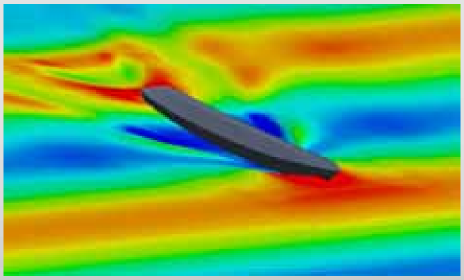


The fluid flow in a pipe-junction modeled with ANSYS CFX technology

Geometry courtesy of CADFEM GmbH.

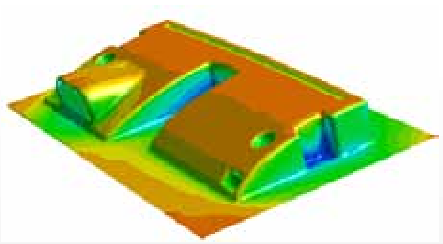


Contours of drug concentration in a stent and capillary wall as predicted by ANSYS FLUENT technology

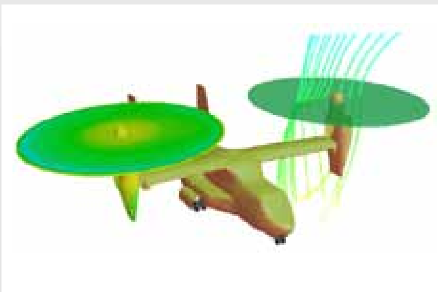


Wave formation around a ship hull modeled with ANSYS CFX software

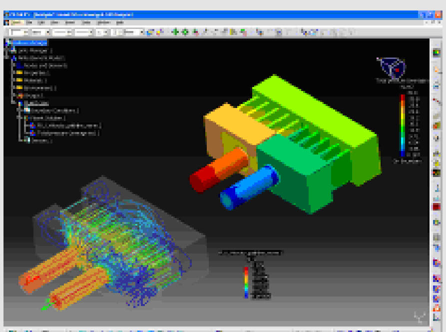
Industry Solutions



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



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

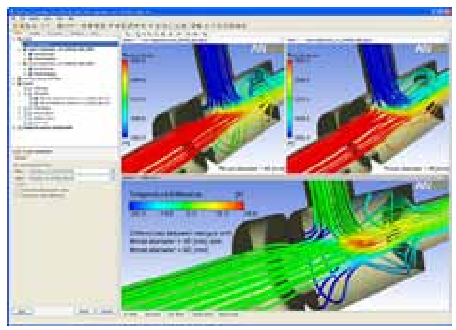


FLUENT for CATIA V5 software works within the CATIA V5 PLM environment, as shown in this simulation of a heat exchanger

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 continues to innovate and integrate so customers can replace more of their traditional capital-intensive design processes with a Simulation Driven Product Development method.



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.

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, almost all 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 solutions from ANSYS 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 40 years as the industry leader — for the best in engineering simulation.

ANSYS®

ansys.com

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

GSA Contract Holder

MKT0000506 10 -10