Realize Your Product Promise™

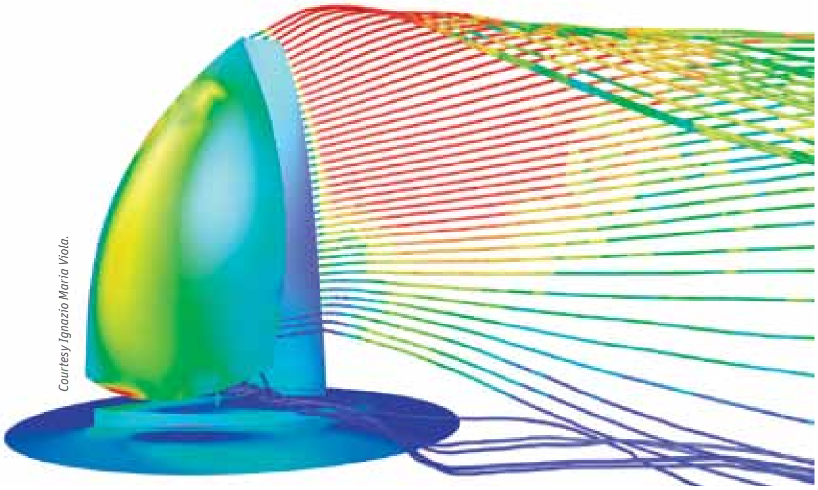
ANSYS®

Fluid Dynamics

Fluid Dynamics
Realize Your Product Promise

With the industry's deepest and broadest fluid dynamics capabilities, ANSYS helps you ensure a steady stream of product and process innovation.

Fluid dynamics plays a critical role in many of the products that we encounter every day — from obvious applications such as water treatment systems and auto and aircraft aerodynamics to boundary-pushing usage in developing Olympic swimsuits, America's Cup racing yachts, eco-friendly skyscraper HVAC systems, new plastic and glass materials, high-speed roller coasters, and leading-edge medical therapies. Behind the scenes, fluid dynamics is involved in the design and manufacture of hundreds of consumer, industrial and defense products. In any application that involves gas flow, liquid flow or heat transfer, fluid dynamics analysis can help deliver innovation and greater efficiency.


Courtesy Ignazio Maria Viola.

America's Cup sailing teams have become top-shelf CFD users, pushing the technology envelope. Just a decade ago, experiments using wind tunnels and towing tanks were industry standard. Computer analysis has the potential to include some of the most complex physics effects possible, requiring analysis of hundreds of millions of cells. When ANSYS broke the 1 billion cell barrier, we set the stage for addressing simulation of full systems and multifaceted subsystems.

As products and processes have become more complex, so too have your fluid dynamics problems: complex moving parts that need transient analysis (such as pistons and valves), phase changes caused by heating or cooling liquids (for example, boilers and warmers) and multiphysics phenomena such as fluid–structure interaction (oil rig and airfoil design).

Engineering simulation software — which enables product design and analysis in a virtual environment — has revolutionized fluid dynamics by automating the solution, even for problems that are numerically large. By identifying physical forces and flow characteristics that are sometimes impossible to measure or gain insight into, CFD solutions from ANSYS can help your company dramatically improve time to market, slash development costs and fulfill your critical product promises — minimizing warranty expenses and driving higher customer satisfaction.

ANSYS: Your Trusted Partner for Industry-Leading CFD Solutions

ANSYS brings together two of the most respected names in fluid dynamics simulation — Fluent® and CFX® — to expertly address your CFD needs with industry-leading technology depth and breadth. Backed by reliable technology that has been verified by academic and independent researchers, technology partners, and a multitude of customers both large and small, our solutions are the tools of choice for innovative product development organizations around the world, in virtually every industry.

The simultaneous pursuit of CFD speed and accuracy can seem next to impossible, and maybe you think you have to choose one over the other. But our software incorporates both

"We use ANSYS software because we need to speed up our development process for new products by speeding up all phases of design. With simulation we can investigate inside our products virtually, not physically, and look at detail that would be impossible to evaluate otherwise. We can improve the efficiency of our products by investigating small changes in parameters and spend less time than we would for creating a real prototype and testing."

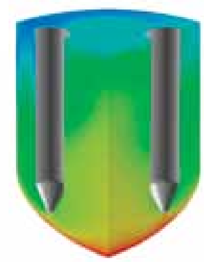
Matteo Cipelli
Advanced Engineering COE Manager
Lowara Srl

capabilities — and without compromise. By reducing the overall time to a reliable solution, ANSYS fluid dynamics software empowers you to do more in less time with fewer resources than ever before. The reliable solution gives you the confidence to predict that your products will thrive, as designed, in the real world.

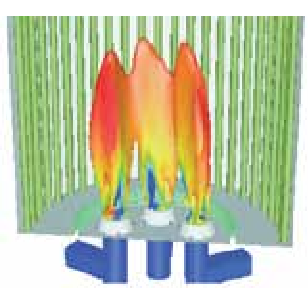
With ANSYS engineering simulation, you can take product failure off your agenda, along with its consequences: profit loss, repair costs, fixed and variable operating costs from downtime, and myriad other expenditures that reverberate throughout the business. The damage can be measured in lost profits — millions per year — as well as in reputation, enough to send a company to bankruptcy.

At the same time, you can build in product innovation and success, since our accurate, efficient tools leave more time for exploring design alternatives and optimizing features. You can leverage our best-in-class CFD technologies to confidently predict the real-world performance of your products and processes — and fulfill your most important customer promises.

The engineering challenges that Terrafugia had to overcome in designing its innovative Transition® "flying car" were formidable. ANSYS CFD helped the team to conduct virtual airflow and wing stall speed tests, critical in advancing the project to production prototype with a high degree of confidence in the design.



Robust CFD tools are especially critical when testing is limited. In designing mixers for use in converting nuclear radioactive waste into storable materials, Bechtel National engineers needed sufficient confidence that their fluid flow model would provide pass–fail judgments. Our software enabled them to accurately predict just that.



Courtesy John Zink Company.

Engineers apply ANSYS CFD to optimize low-NOx burners for power generation, resulting in good flame quality and excellent run length with minimal downtime — not to mention reduced greenhouse gases.

In motorsports, accuracy can be the difference between winning and losing. Red Bull Racing Team won Drivers' and Constructors' championships by optimizing aerodynamics with the help of our CFD.

Maximizing Flow and Velocity

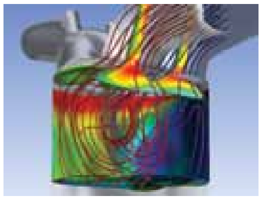
ANSYS solutions have emerged as the industry standard for CFD analysis, thanks to a unique combination of speed and accuracy.

New product designs start to incur costs from the very beginning of the development process and, if all goes as planned, generate revenue only when they make it to market. This fundamental aspect of product development economics is painfully obvious to any company whose products languish on the drawing board for too long — racking up development costs instead of recouping expenditures in the marketplace.

Fluid dynamics problems are becoming more complex as product designs and processes grow in sophistication. Your company already faces shorter product development schedules, which force your engineers to perform numerically large simulations quickly.

But product and process integrity is as important as speed. The high fidelity and accuracy of our CFD solutions provide you with the best of both worlds. Therefore, you can do more in less time, with fewer resources than ever before — while still upholding your performance promise.

Courtesy Daimler Chrysler.



ANSYS CFD can solve a wide range of automotive applications, including wind buffeting and engine optimization.

For complex tasks such as geometric meshing, ANSYS offers a high level of automation and user definition that speeds analysis. Customized technologies for specific industries — including turbomachinery and external aerodynamics — streamline common simulation tasks.

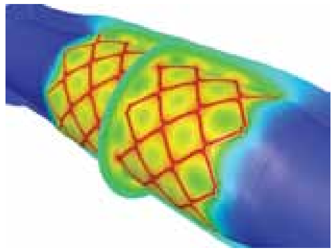
ANSYS: An Abundance of Customer Applications

Engineers in a diverse range of industries are using CFD solutions from ANSYS to manage even the most numerically large and complex simulations, driving high-velocity innovation within their companies.

For example, automotive engineers — including those on elite Formula 1™ race teams — take vehicle aerodynamic performance to new levels, gaining dramatic speed and fuel efficiency improvements. Our software helps automotive engineers with a range of other challenges, including internal combustion engine improvements, emissions control device development, thermal studies of underhood systems, and cooling and ventilation systemdesign for car interiors.

In industrial environments, engineering teams choose ANSYS software to analyze fluid dynamics in and around components such as pumps, valves and turbines. Not only do these engineers focus on developing new products and processes to improve output and performance, they also examine equipment retrofits, erosion patterns, and other causes of equipment failure, stresses and fatigue — with an eye toward reducing long-term maintenance costs.

In the aerospace industry, our CFD products are supporting initiatives to build lighter, more efficient, safer and more survivable aircraft.

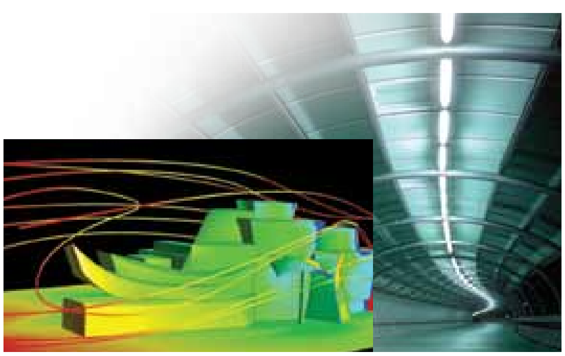


There are no shortcuts in developing life-saving, precise medical devices — such as stents to keep blood vessels open — but there is a way to shorten process. Biomedical engineers turn to our software to explore issues such as cardiovascular flows and stresses, drug delivery systems and their travel paths through the body, and the velocities inside inhalers and other respiratory devices.

As they explore novel composite materials and new designs for components such as landing gear, aerospace engineers must ensure that they are also maximizing aerodynamics, which leads to optimal range, efficiency, speed and maneuverability for all types of aircraft.

Researchers at universities and other organizations around the world turn to trusted ANSYS CFD solutions to support breakthrough applications in exciting new areas. Their fluid dynamics analysis of human blood vessels, jet fuel chemistries, lean-burn combustors, next-generation fabric manufacturing and other areas is pushing the current boundaries of engineering simulation — and driving new levels of CFD analysis. ANSYS is there to meet their specialized needs — as well as the customized fluid dynamic analysis demands of your business.

In the energy industry, ANSYS CFD tools can be applied to a wide range of combustion, emission and power generation problems. Software from ANSYS has been leveraged to design innovative coal mine ventilation systems, next-generation burners and furnaces, pollution control devices, oil drilling machinery, piping systems, and spent nuclear fuel processing technologies. From new product design through fatigue testing and maintenance reduction initiatives, CFD solutions from ANSYS play a critical role in defining our energy future.



The structural integrity of any building is only as good as its individual parts — which all contribute to how the building will perform under normal, or extreme, conditions. ANSYS CFD has helped explore innovation in iconic landmarks related to wind patterns and erosion, HVAC systems, roof drainage, and a plethora of other factors.

In designing the yoomi baby bottle warmer with our tools, Intelligent Fluid Solutions cut two years out of its development schedule (compared to build-and-test methods) and consequently saved tens of thousands in costs. The final step quickly crunched through thousands of possible geometric alternatives to find the best design. The result: yoomi engineers managed to double the performance of the initial prototype while building just four physical models.

Brimming with Best-in-Class Technologies

Robust CFD tools from ANSYS represent the state of the art in fluid dynamics simulation.

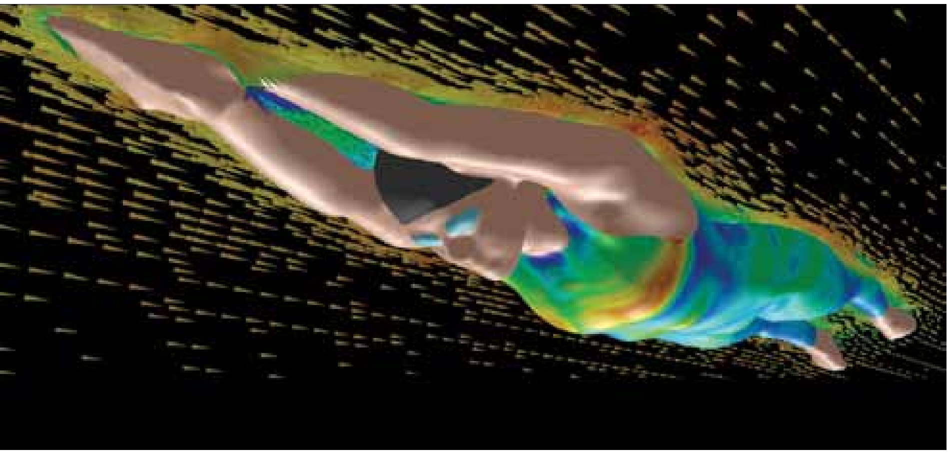
Combining the well-known expertise of Fluent and CFX, ANSYS solutions cover the entire spectrum of fluid dynamics analysis. From leading-edge turbulence and reacting flows to multiphase problems such as solidification and gasification, free surface, boiling, cavitation, wet steam, slurry, real gas, noise, solidification, and melting, we deliver industry-leading capabilities to meet your most challenging CFD simulation needs.

More sophisticated and accurate physical models drive more-authentic CFD simulations that can reveal new levels of insight into product designs. The ANSYS suite includes a variety of models for reacting and combusting flow (including surface and porous media reactions and discrete ordinates radiation models supporting entirely specular walls). So you can achieve reliable and efficient calculations, our CFD software uses industry-leading numerics to virtually represent all the physics happening in a process or device. Solutions include finite-volume solvers using both coupled and segregated methods for general fluid-flow modeling as well as a finite-element solver for viscous flows of complex fluids.

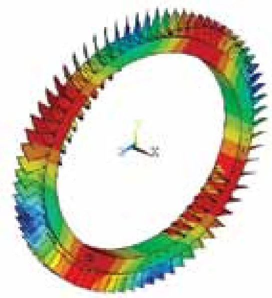
ANSYS CFD solvers are designed to handle all types of meshes, with moving and deforming mesh capabilities, advanced multi-grid methods, and solution-based, adaptive remeshing functionality. Our tools deliver highly accurate results across all flow regimes — from hypersonic to creeping flows, Newtonian to non-Newtonian.

The adjoint solver in ANSYS software provides specific information that is otherwise difficult to gather. Because adjoint solutions estimate the effect of a change prior to actually making the change, this exclusive capability adds to the speed of simulation.

Though such added complexity can lead to longer turnaround time, engineers don't have to make trade-offs: The world-leading CFD suite offers both speed and accuracy. HPC and parallel computing provide powerful and scalable options — so you get more geometric detail, larger systems (such as a full 360-degree blade passage, not a single-blade passage), and more complex physics (for example, an unsteady turbulence rather than a steady turbulence model). The result is enhanced insight into product performance, insight that can't be gained any other way. This detailed understanding can yield enormous business benefits by revealing design issues that might lead to product failure or troubleshooting delays.



Speedo® realized its product promise at the 2008 Olympics, where the majority of medals won and world records broken were achieved by competitors wearing swimsuits designed in part with our CFD software.



Courtesy PCA Engineers Ltd.

Blade flutter with compressors and turbines is a serious cause of machine failure — but until recently, design engineers were unable to satisfactorily investigate and avoid this phenomenon. Our bidirectionally coupled CFD and structural tools now predict vibration modes that occur over an entire wheel from a single-blade component model. Fluid–structure interaction accurately predicts how a design will function in a real-world environment.

ANSYS: Leadership across Every Physics Discipline

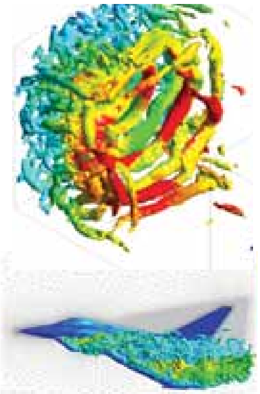
Our software is widely acknowledged among CFD engineers as the industry-leading fluid dynamics simulation technology. And you can combine the tools with our best-in-class technologies in related disciplines such as structural, electromagnetics and thermal analysis, allowing you to solve complex problems involving fluid–structure interactions, flow-induced vibration, and thermal stresses created by fluid flows. You can rely on ANSYS to deliver robust capabilities — individually or in tandem — to produce fast, accurate results.

These powerful cross-discipline tools come together via the ANSYS Workbench™ platform, which provides a powerful multi-domain simulation environment for CFD analysis. For example, in FSI studies, Workbench enables easy, intuitive problem setup; automated load mapping between physics; support for dissimilar mesh interfaces; and one-way, two-way and rigid-body FSI analysis.

Whatever your industry, application or specific CFD problem, ANSYS offers a tightly integrated software portfolio — and a flexible, intuitive platform — to help you wisely allocate engineering resources in creating optimal products, quickly and cost effectively.

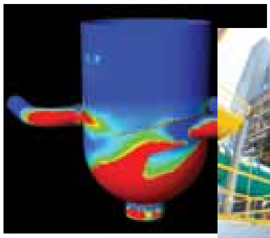
"Without ANSYS, we would go in blind designing new reactors or making modifications to existing equipment. The confidence level that we get from using simulation is very high: We generally understand the preferred direction to pursue, rather than going in all possible directions."

Matteo Fumagalli
Innovation Engineer
MEMC



Courtesy EADS Germany GmbH Military Air Systems and the DESIDER Project.

ANSYS industry experts continue to develop advanced turbulence models. Especially applicable to the aerospace industry, these ANSYS tools eliminate the trade-offs between speed and accuracy.



State-of-the-art research at Petrobras examines heat and mass transfer that occurs in chemical processes. The energy giant chose us because our advanced technology includes physical models important to the petrochemical processes as as well as excellent parallel performance for solving very complex industrial multiphase flows.

ANSYS

ANSYS, Inc.
www.ansys.com
ansysinfo@ansys.com
866.267.9724

ANSYS is dedicated exclusively to developing engineering simulation software that fosters rapid and innovative product design. Our technology enables you to predict with confidence that your product will thrive in the real world. For more than 40 years, customers in the most demanding markets have trusted our solutions to help ensure the integrity of their products and drive business success through innovation.

MKT 79