Realize Your Product Promise™

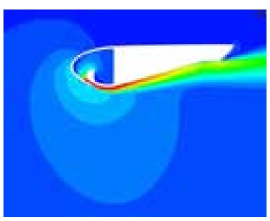
Fluent

"ANSYS simulation software is incredibly reliable and accurate. Simulation enables us to drastically reduce lead times and get solutions to the circuit much, much quicker so that we are more competitive race-to-race."

Steve Nevey
Business Development Manager
Red Bull Technology

The ANSYS Fluent package combines deep physics and years of simulation development expertise to solve CFD challenges — right out of the box.

Trust ANSYS fluid dynamics to deliver accurate insight into your products, fast.



"In developing a revolutionary fan concept, the Dyson team investigated 200 different design iterations using simulation, which was 10 times the number that would have been possible had physical prototyping been the primary design tool."

Richard Mason
Research, Design and Development Manager
Dyson

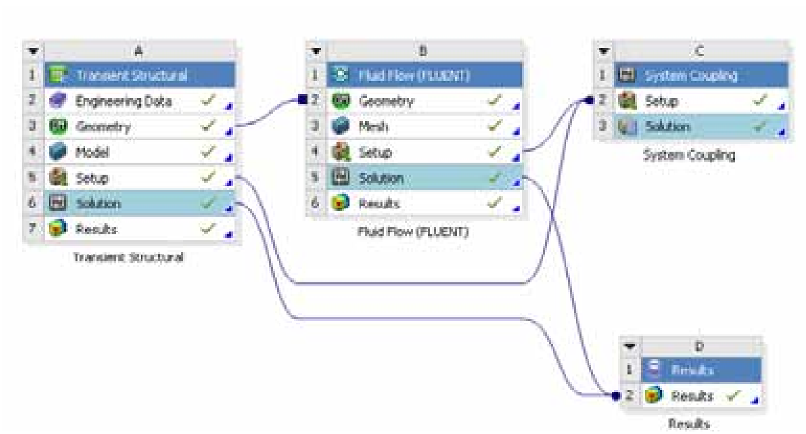
There are huge numbers of engineering applications that can benefit from computational fluid dynamics simulation. Whether you analyze commonplace fluid flow and heat transfer or work with complex transient reacting flows, ANSYS Fluent® software should be an integral part of your product design and optimization process.

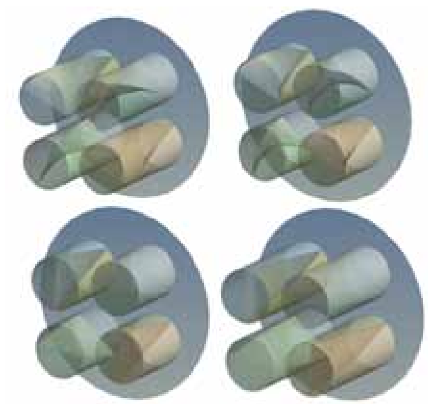
A fully featured fluid dynamics solution for modeling flow and other related physical phenomena, Fluent offers unparalleled analysis capabilities. It provides all the tools needed to design and optimize new equipment and to troubleshoot existing installations. The versatile technology offers insight into how a product design will behave in the real world, all before a single prototype is built. Fluent's capabilities are developed by world-renowned experts and supported by extremely experienced engineers — so you can have confidence in the solution as you develop higher quality products faster, decrease time to market, reduce risk and increase innovation.

Fast, Accurate Solutions Glean Product Behavior Insight

Even amid global competitive pressures and complex requirements, you cannot afford to sacrifice accuracy. We understand this tenet, delivering software that provides accurate insight into your product's behavior — quickly and efficiently.

To provide high productivity, the ANSYS Workbench™ platform directly couples with your CAD software and automatically extracts and meshes fluid volumes. You control this easy-to-perform operation via a small set of parameters; the yield is the high-quality meshes critical for both accurate and fast CFD simulation.





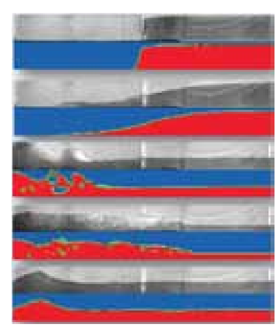
ANSYS Fluent is both customizable and fully integrated within ANSYS Workbench, allowing you to adapt capabilities to quickly solve specific challenges with great ease.

Parametric simulation helps to evaluate fluid dynamics performance of a large number of designs, such as this selective catalytic reduction mixer.

Our CFD package includes solvers that accurately simulate behavior of the broad range of flows that engineers encounter daily — from Newtonian to non-Newtonian, from single-phase to multi-phase, and from subsonic to hypersonic. Each solver is highly robust, well tested, validated and optimized for fast simulation time. Time tested and part of a single environment, the highly efficient solvers deliver both accuracy and speed.

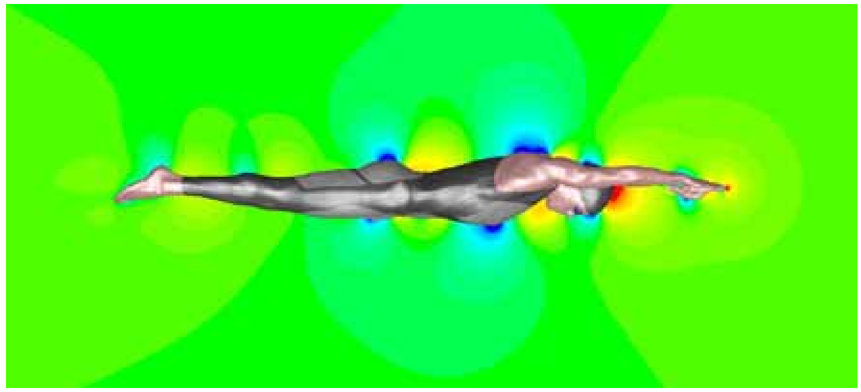
For deeper insight — such as making informed decisions about small adjustments that yield large performance improvements — you can increase the granularity of the analysis. Such improved resolution requires more computational resources and parallel computing. Fluent has a record of outstanding parallel scalability, ranging from two processors to thousands, giving you high-fidelity results in the shortest possible time.

Optimizing your product requires evaluating a large number of designs. Your toolkit should include capabilities for design exploration and optimization studies in a fast, easy-to-use and robust manner — which provides better insight into product performance. Capabilities within ANSYS Workbench enable efficient, fully automated optimization (or design of experiments) for tens or hundreds of design points; the technology can evaluate many design points concurrently. Workbench makes the process easy by controlling the execution, results data and file management for each set of design points.

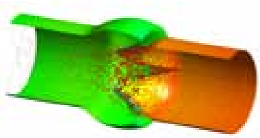


Courtesy Technical University of Berlin

Independent validation attests to Fluent's accuracy. This dam break simulation compares experiment (grey) and multiphase simulation (color) at different times. White regions correspond to breaking waves in the experiment.



To decrease drag on its LZR RACER® suit, Speedo® used Fluent simulations involving a precise boundary layer meshing technique to resolve fine fluid flow details.



In this two-way FSI solution of transient blood flow through a three-leaf mitral valve, blood flow is represented with non-Newtonian fluid, and biological tissue is modeled with anisotropic hyperelastic material properties. The solution includes 3-D remeshing of the fluid domain, nonlinear materials and contact.

Addressing the Complex Physics of a Complete System

Over the years, products have become extremely complex. So too have your fluid dynamics problems: Systems with moving parts (such as pistons and valves) require transient analysis; systems with phase changes due to heating or cooling liquids (for example, heat exchangers) require accurate, multi-phase capabilities; and systems with challenging multiphysics phenomena (such as fluid–structure interaction) call for easy-to-use, accurate advanced capabilities.

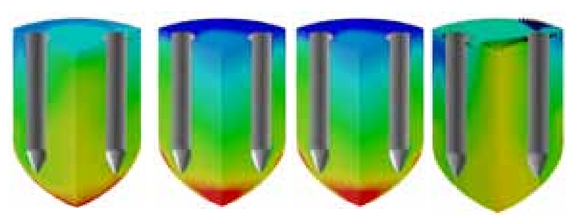
To gain more insight into product behavior, you need to consider the full range of physics present in the system you are designing. Fluent offers state-of-the-art advanced capabilities to model laminar and turbulent flows as well as more-complex physics including multiphase flows, chemical reactions, radiation and particulate dynamics. You can be confident that Fluent will accurately predict product behavior because all models are thoroughly tested and validated.

For best-in-class products, engineers can no longer rely on analyzing one type of physics (fluids, structural or electromagnetics). Instead, you must study all physics along with their interactions.

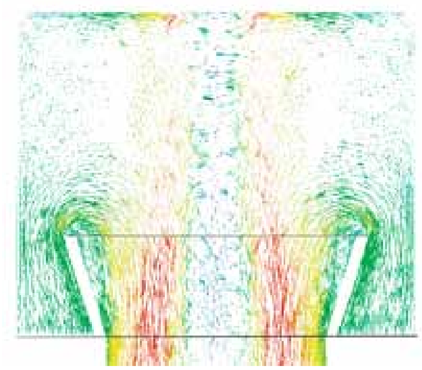
You can seamlessly couple Fluent with ANSYS structural mechanics or electromagnetics simulation tools to gain insight into the entire system. For example, you can study how the fluids system deforms the structure that contains it, or how the heat generated by an electronic component affects fluids temperature.

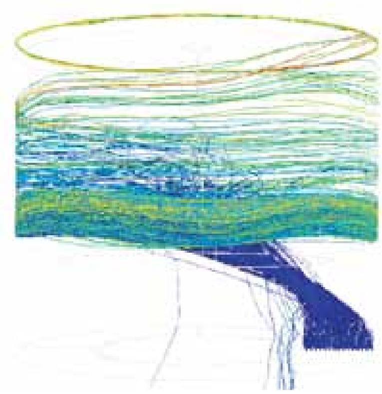
The Right Investment in the Right Technology

Simulation R&D outlays — in terms of time, effort and dollars — must provide a positive financial return, now and over the long term. Fluent is a safe CFD investment. The technology is the tool of choice for many of the world's largest and most innovative companies. Its reliability is well validated by academic researchers, independent organizations, technology partners and clients.



Because of our commitment to quality assurance, customers use our CFD tools with full confidence. Bechtel National applies Fluent to convert nuclear waste into a stable, glass-like material for storage. Multiphase tools accurately predict the mixing ability of vessels in which the waste is treated.

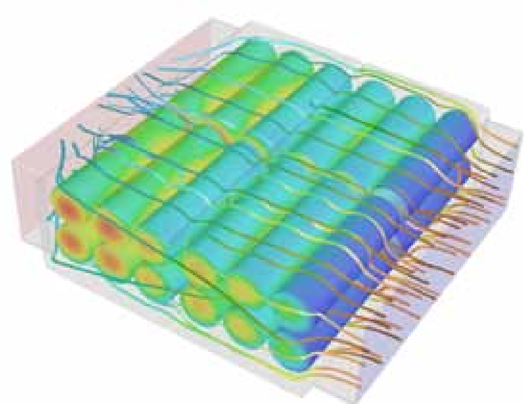




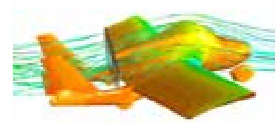
With a goal of extending off-highway vehicle filter, engine and component life, Donaldson Company leveraged Fluent to develop and optimize an air pre-cleaner. Physical tests showed that, as predicted by the simulation, the pre-cleaner separates 99 percent of contaminants 20 microns and larger — contributing greatly to air filter service life.

Choosing the right CFD software also means choosing the right software company, one in which developers and support teams have the expertise and knowledge to create tools that enable robust and accurate solutions. One of the world's largest pools of simulation experts, the ANSYS team serves a comprehensive range of industries and applications. We provide the assistance and training needed to ensure success in any simulation challenge you face.

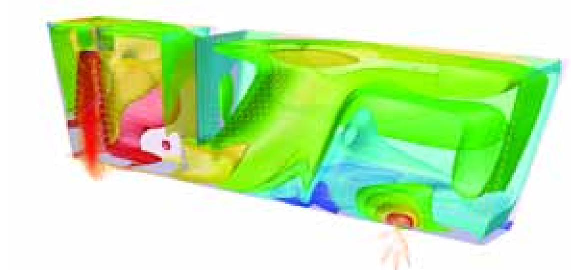
Some applications require the use of software products that have traceable and well-established quality assurance processes. ANSYS has a deep and longstanding commitment to quality. We were the first organization to receive ISO 9001 certification for design analysis software, and we maintain this certification year after year.



For HEV/EV development, such as cell temperature distribution and flow field in a battery pack, the ANSYS integrated platform enables researchers to simultaneously evaluate individual components and subsystems under various physical conditions — fluids, structural stress, thermal, electromechanics and more — as well as the interactions between them at the system level.



Airplane stall prediction can pose a difficult problem, even for sophisticated CFD tools. In designing a roadable aircraft, Terrafugia developed a virtual blade modeler that plugged in to Fluent and created additional capabilities to model the aircraft's propeller. This way, the aircraft could be reshaped as required.



Because turbulence has a strong effect on flow characteristics over a wide range of applications, such as this tundish, Fluent offers a number of advanced turbulence models in the form of algebraic, one-equation, two-equation and Reynolds stress models.

Advanced physics capabilities and ease of use help ensure that you meet your product development targets.

The oil and gas industry uses multiphase models extensively to troubleshoot separation equipment performance.

"ANSYS software improvements have led to reduced run times; multiphase and turbulence models have a greater ability to handle primary and secondary phases."

David Stanbridge
Managing Director
Swift Technology Group

CAD Import and Meshing

From CAD import to geometry meshing, our flexible tools allow you to automatically create meshes or hand-craft them. ANSYS meshing can extract fluid volume from a CAD assembly and automatically create tetrahedral or hexahedral meshes with inflation layers. We also offer advanced repair tools so you can import and prep geometry for partly or fully manual meshing. ANSYS pre-processing tools provide the high-quality meshes your project needs, so you obtain accurate results.

Advanced Physics Modeling Capabilities

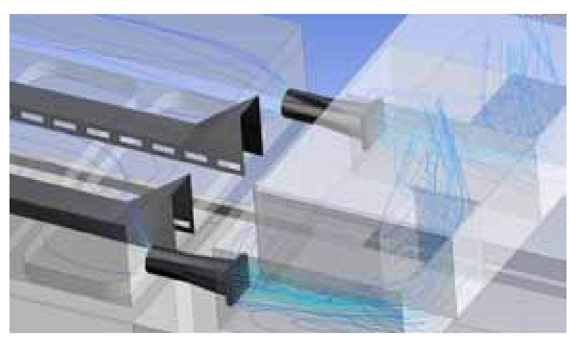
Our CFD technology incorporates all the advanced physics modeling capability required for a wide range of applications. Models are fully tested and validated to ensure that your simulation consistently yields results you can trust.

Robust Solvers

The software includes a pressure-based coupled solver, a fully segregated pressure-based solver, and two density-based solver formulations. No matter the problem you are studying, ANSYS has the solver to fit your needs

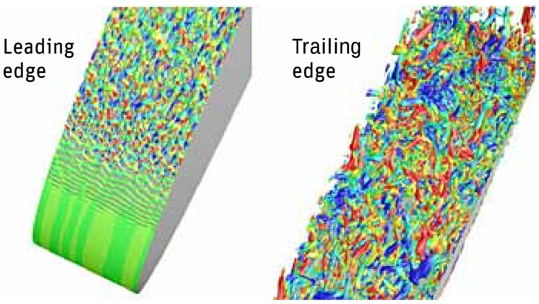
Turbulence Models

Fluent delivers cutting-edge turbulence modeling capabilities in an unparalleled breadth of models. These include several versions of the popular k-epsilon and k-omega models as well as the Reynolds stress model for highly anisotropic flows. Advanced scale-resolving turbulence models are also available: large eddy simulation (LES), detached eddy simulation (DES) and scale-adaptive simulation (SAS). In addition, innovative transition models accurately predict flow in which the boundary layer transitions from laminar to turbulent regime.



"Zitron uses Fluent because the tool provides the exceptionally wide range of physical models needed to accurately predict tunnel ventilation system performance."

Ana Belén Amado
Mining Engineer
Zitron



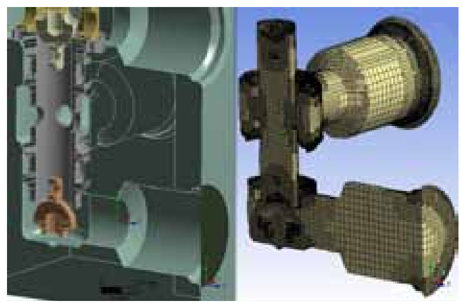
Aero-acoustic simulation of trailing-edge noise from NACA 0012 airfoil

Reacting Flows

Our software incorporates a comprehensive suite of reacting flow-modeling capabilities. You can simulate gaseous reactions using either reduced or complex chemistry. Pollutant models are built in to allow easy and accurate pollution emission predictions for NO, SO and soot. You also can simulate surface reactions. All reaction models are fully compatible with all turbulence models.

Multiphase Flows

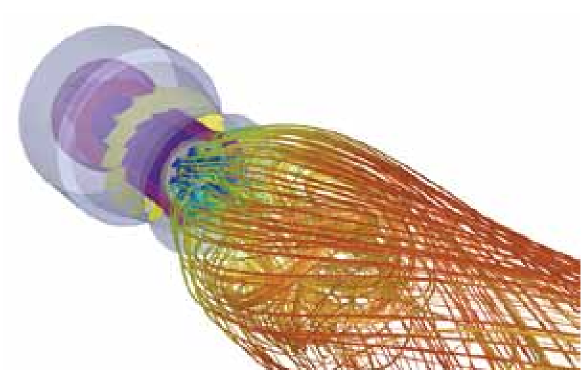
You can access a wide range of multiphase flow models to gain insight into your designs. You can model the behavior and interactions of a large number of phases — including any combination of liquid, solid and gas as well as granular flows — using either a mixture fraction or Eulerian model. You can track immiscible flows very economically using the volume-of-fluid (VOF) model. Particles, droplets breakup and evaporation are handled using the discrete phase model. Our tools also model phase changes.



Assembly meshing enables a dramatic reduction in meshing time for typical CAD models by quickly extracting and meshing the fluid volume of a CAD assembly.

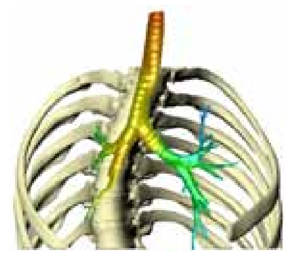
Other Advanced Capabilities

Fluent includes advanced acoustics-modeling capabilities (such as the Ffowcs Williams–Hawkings acoustic analogy and computational aero-acoustics capabilities) to model propagation of acoustic noise. Dynamic/moving mesh capabilities allow you to model flows in or around moving parts.



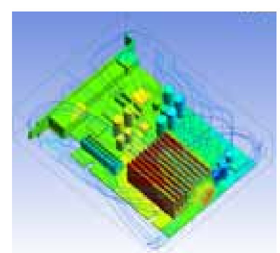
Courtesy GE Energy.

Fluent's advanced reacting flow models tackle a vast array of gaseous, coal and liquid fuel combustion simulations, such as this low NOx burner.

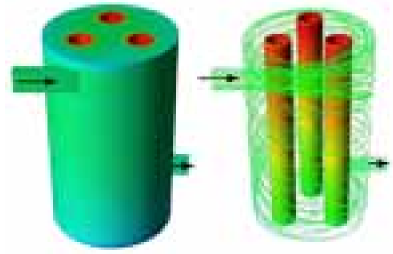


Courtesy FluiDA nv.

Obstruction site (and subsequent location) of an intrabronchial stent, which re-inflated blocked lower right lung lobe. Pressure contours are plotted in airway.



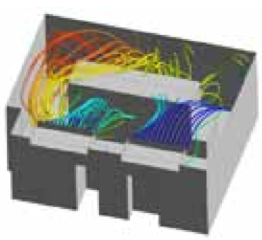
Temperature contours and streamlines from conjugate heat transfer simulation of PCB

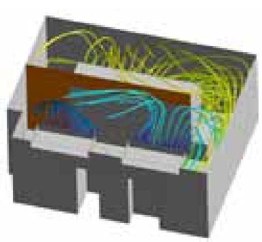


"Using the Fluent UDF capability, our research team automated the process of averaging three-dimensional temperature and oxygen mole fraction data from the shell-side simulation and mapping it onto the tube-side band mesh."

Robert J. Kee
George R. Brown Distinguished Professor
of Engineering
Colorado School of Mines

Integrated capabilities — from HPC enablers to design exploration tools — expedite breakthrough productivity.





Our CFD tools delivered insight into better temperature control in a data center. Based on simulation results, a simple alteration to the room helped to maintain the appropriate temperature and save energy.

Parallel Scalability

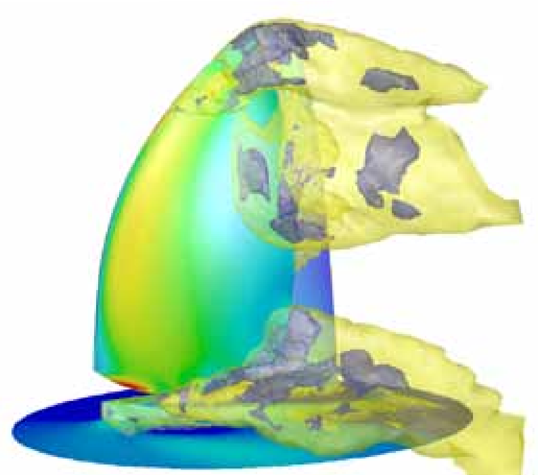
Fluent's parallel scaling capabilities ensure that your simulation efficiently utilizes networks of homogenous or heterogeneous processors. Dynamic and physics-based load balancing technologies automatically detect and analyze parallel performance and adjust the distribution of computational cells among the different processors to maximize computation speed.

Customization

ANSYS Fluent is readily customizable and extensible. For example, you can implement your own specialty physics or tailor and script the user environment to implement best practices or further automate your workflow.

Design Exploration and Optimization

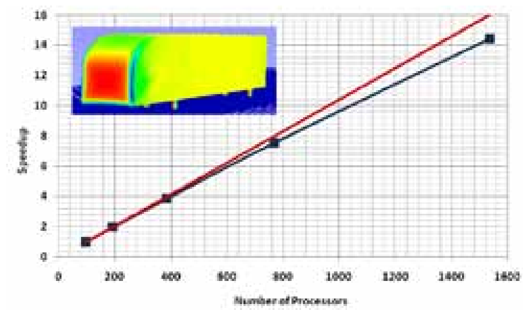
To increase simulation throughput, Fluent allows automatic investigation of multiple, parametric design variations — all without any programming. The ANSYS integrated solution for design of experiments provides for easy analysis of thousands of data points in a single user environment. Which parameters is your design most sensitive to? Which design parameters require the tightest control? Fluent leads you to the answer via integrated tools for design optimization and Six Sigma analysis.



Courtesy Ignazio Maria Viola.

Aerodynamic simulation of a yacht model with more than a million cells was achieved using HPC capabilities.

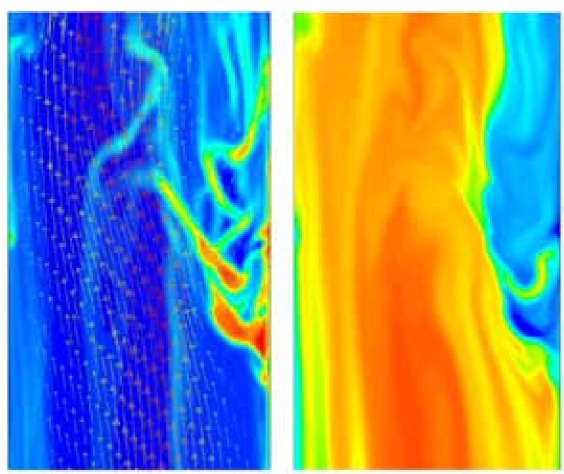
Courtesy KBC Advanced Technologies.



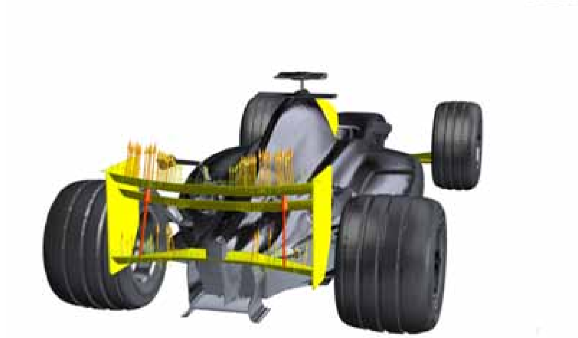
Using the Euler–granular model, simulating the effect of hardware on distribution of droplets and catalyst for a fluidized catalytic cracking unit can lead to a better understanding of riser reactor performance.

Parallel scalability results (black) of aerodynamics simulation, 111 million cells, compared with ideal (red)

ANSYS Fluent is the first commercial CFD code to provide innovative adjoint solver technology. This tool provides optimization information that is difficult or cumbersome to determine. Because it estimates the effect of a given parameter on system performance — prior to actually modifying the parameter — the adjoint solver further increases simulation speed and, more importantly, contributes to innovation.

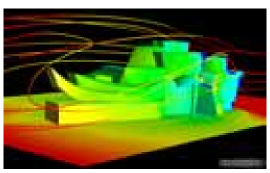


Fluent's well-established granular flow modeling is central to progress made by SINTEF Materials and Chemistry for chemical-looping combustion, an important technology in reducing emissions. Developing fundamental models was eased with user-defined functions (UDFs) compiled within the stable, parallelized solver.



The Fluent adjoint solver indicates what portion of geometry should be modified as well as how to alter it for optimized down-force on F1 car design.

ANSYS Fluent is part of a streamlined platform that manages your product design process from geometry import to optimization.



"ANSYS software was absolutely critical to the engineering verification we performed for the Guggenheim Museum Bilbao because of its ability to handle complex shapes and free-form surfaces."

Michael Stadler
Research Scientist
NInsight

Simulation-Driven Product Development™ relies on design process compression, using solutions that fully automate your simulation workflow — so you can focus on your engineering goals. Our advanced simulation workflow and application technologies are key to accelerating processes and gaining the necessary insight into your product so you can quickly make the right design decisions. With unified ANSYS Workbench workflows, there is no need to purchase, administer or configure third-party coupling software, nor pre- and post-processing software.

Multiphysics

The ANSYS portfolio of high-fidelity simulation tools enables you to accurately predict real-world, multiphysics behavior of industrial designs. Phenomena such as flow-induced vibration and material deformation induced by fluid can be readily captured using our multiphysics tools. We provide comprehensive technologies for all physics disciplines — structural mechanics, heat transfer, fluid flow and electromagnetics. By combining these, you can solve complex industrial engineering challenges to optimize your entire product.

ANSYS provides a powerful integrated solution for automating your different physics simulations. Data transfer to and from multiple physics eliminates the time required to manually convert output from one stage of your work process into input for another.

Optimization

The inherent and integrated parametric capabilities of our simulation framework enables optimization for all types of engineering applications.

Third-Party Tools

ANSYS provides support within the ANSYS Workbench platform for third-party tools such as CAD software. This extends to simulation capabilities as well.

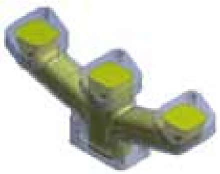
Post-Proccessing and Archival

The powerful post-processor for ANSYS CFD provides advanced quantitative and high-quality visual post-processing capabilities, including easy creation of charts, high-quality images and videos.

Simulation engineers generate large volumes of data that must be archived in a searchable format. ANSYS Engineering Knowledge Manager™ (EKM) captures simulation data to greatly improve long-term simulation and efficiency.

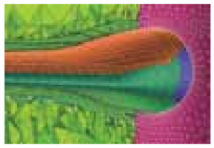
ANSYS Fluent

Geometry



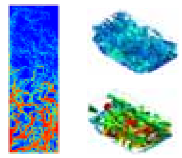
Flow volume automatically extracted

Mesh



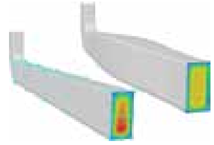
Multi-zone meshing combines the strength of various meshing tools to automatically generate this hybrid grid for a tidal turbine.

Comprehensive Models



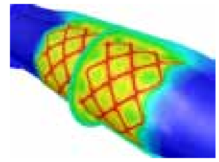
Multiphase and turbulence models, among other advanced technologies

Mesh Morpher



Automatic geometry adjustment of velocity distribution of L-shaped duct

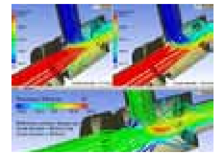
Customization



User-defined function, aniso-tropic diffusion of drug from blood vessel stent

Post-Processing

Geometry courtesy CADFEM GmbH.



ANSYS CFD-Post provides powerful post-processing and reporting capabilities.

Pre-Processing → **Simulation** → **Post-Processing** → **Archive**

Other ANSYS Engineering Simulation Capabilities

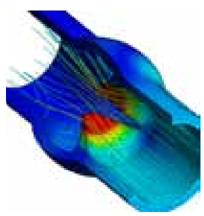
CAD

ANSYS DesignModeler™ and ANSYS SpaceClaim DirectModeler provide modeling and geometry creation functions for fluid dynamics analysis. The entire ANSYS suite is CAD independent, enabling data import from various sources. In addition, we collaborate with leading CAD developers to ensure an efficient workflow.

Integration

ANSYS Workbench is the framework for the industry's broadest and deepest suite of advanced engineering simulation technology. It delivers unprecedented productivity, enabling Simulation-Driven Product Development™.

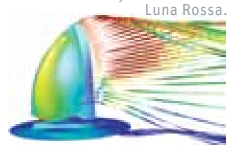
Multiphysics



To help ensure a successful product, R&D teams must accurately predict how complex products will behave in a real-world environment. The ANSYS suite captures the interaction of multiple physics: structural, fluid dynamics, electromechanics, and systems interactions. A single, unified platform harnesses the core physics and enables their interoperability.

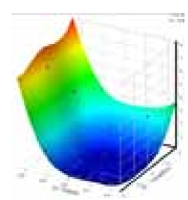
HPC

Courtesy I.M. Viola and Luna Rossa.



High-performance computing enables creation of large, high-fidelity models that yield accurate and detailed insight. ANSYS offers scalable solutions and partners with hardware vendors to ensure that you get the power and speed you need.

Design Optimization



Good design starts with identifying the relationship between performance and design variables. ANSYS DesignXplorer™ enables engineers to perform design of experiments (DOE) analyses, investigate response surfaces, and analyze input constraints in pursuit of optimal design candidates.

Data Management

ANSYS EKM addresses critical issues associated with simulation data, including backup and archival, traceability and audit trail, process automation, collaboration and capture of engineering expertise, and IP protection.

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 over 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.