

## ANSYS FLUENT

### Powerful Computational Fluid Dynamics Software for Optimization of Product Development and Processes

Companies throughout the world benefit from the extensive range of physical modeling capabilities available in ANSYS® FLUENT® software. The broad physical modeling capabilities of this important engineering design and analysis tool have been successfully applied to industrial applications ranging from flow over an aircraft wing to combustion in a furnace, from bubble columns to glass production, from blood flow to wastewater treatment plants. The ability of the fluid dynamics software to model internal combustion engines, aero-acoustics, turbomachinery and multiphase systems has broadened its reach across the product engineering sector.

With its longstanding reputation of being user friendly and robust, ANSYS FLUENT makes it easy for new users to come up to productive speed. The integration of ANSYS FLUENT into the ANSYS® Workbench™ environment combined with the ability to use ANSYS CFD-Post software for post-processing creates a comprehensive fluids simulation software solution available to the engineering analysis community. In conjunction, ANSYS technical support offers comprehensive user training focused on making customers successful. Throughout the years, these important components — comprehensive models, usability and technical support — have combined to accelerate the adoption of ANSYS FLUENT across a broad spectrum of industries.

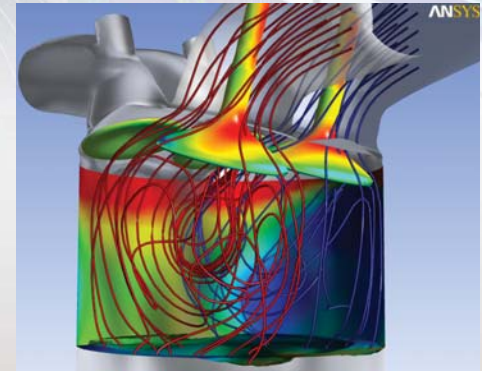
#### ANSYS FLUENT and the ANSYS Workbench Environment

ANSYS FLUENT software is fully integrated into the ANSYS Workbench environment, the framework for the full engineering simulation suite of solutions from ANSYS. Its adaptive architecture enables users to easily set up any problem from standard fluid flow analyses to complex interacting systems with simple drag-and-drop operations. Users can easily assess performance at multiple design points or compare several alternative designs. Within ANSYS Workbench, applications from multiple simulation disciplines can access tools common to all, such as CAD connection, geometry and meshing tools. ANSYS CFD-Post software can be used to compare results and to perform final data analysis. Data transfer from ANSYS FLUENT to the ANSYS® Mechanical™ program permits one-way FSI calculations.

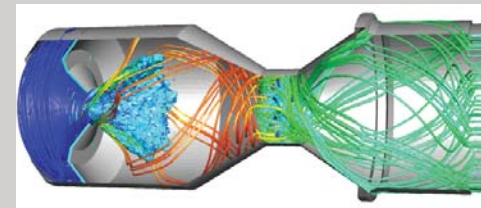
#### Technology

ANSYS FLUENT software leads the engineering simulation industry in the number of complex physical models offered for solution on unstructured meshes. Combinations of elements in a variety of shapes are permitted, such as quadrilaterals and triangles for 2-D simulations and hexahedra, tetrahedra, polyhedra, prisms and pyramids for 3-D simulations. Meshes can be created using ANSYS or third-party meshing products and, in the case of polyhedra, via automatic cell agglomeration directly within ANSYS FLUENT. Meshes containing many cells, even over a billion, can quickly be automatically partitioned when read into ANSYS FLUENT software running on a compute cluster. Additional built-in tools can be used to further manipulate meshes.

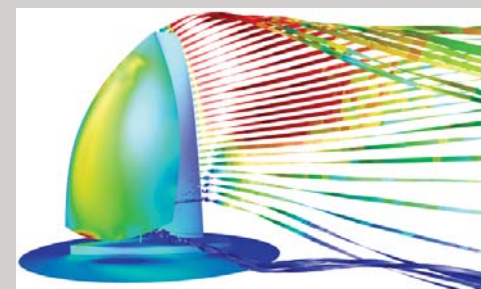
## Industry Solutions



Internal combustion engine and the flow inside modeled using ANSYS FLUENT software

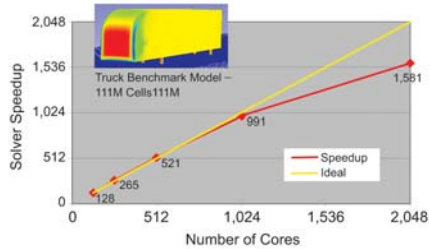


Evaporating diesel fuel inside an autothermal reformer mixing chamber  
*Courtesy Forschungszentrum Julich GmbH.*



1 billion cells were used to model the fluid flows around the spinnaker and main sail of a racing yacht design  
*Courtesy Ignazio Maria Viola.*

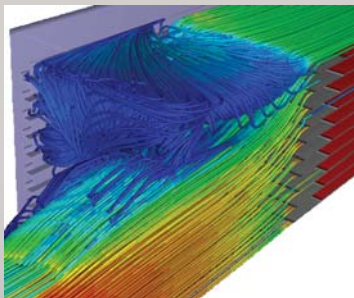
## Industry Solutions



Scaling of ANSYS FLUENT software is nearly ideal up to 1,024 processors and 78 percent of ideal at 2,048 processors. Data courtesy SGI, based on the SGI Altix® ICE 8200EX using quad-core Intel® Xeon® processor E5472 with Infiniband®



Vortical structures generated by an aircraft landing gear visualized using deformation isosurfaces colored by velocity magnitude



Pathlines in a nuclear reactor showing a vortex observed in the space between the diffuser and plate-region

*Courtesy ANOVA-CAD-CAE TEST and Turkish Atomic Energy Authority.*

**Numerics and Parallel Processing:** Inside ANSYS FLUENT, sophisticated numerics and robust solvers — including a pressure-based coupled solver, a fully segregated pressure-based solver and two density-based solver formulations — help to ensure robust and accurate results for a nearly limitless range of flows. Advanced parallel processing numerics can efficiently utilize multiple multi-core processors in a single machine and in multiple machines on a network. Dynamic load balancing automatically detects and analyzes parallel performance and adjusts the distribution of computational cells among the processors so that a balanced load is shared by the CPUs even when complex physical models are in use. ANSYS FLUENT is available on Windows®, Linux® and UNIX® platforms.

**Turbulence:** ANSYS continually defines the cutting edge of turbulence modeling in commercial CFD software and offers the engineering community an unparalleled breadth of models. Inside ANSYS FLUENT, several popular  $k$ -epsilon and  $k$ -omega models are available, as is the Reynolds stress model (RSM) for highly swirling or anisotropic flows. Advanced computing power at reduced cost is making large eddy simulation (LES) and the more economical detached eddy simulation (DES) turbulence models attractive for industrial applications. Also available are innovative new models such as those for predicting laminar-to-turbulent transition and the novel scale-resolving Scale-Adaptive Simulation™ (SAS) model for flows in which steady-state turbulence models are insufficient. Wall functions and enhanced wall treatment options allow for the best possible representation of all wall-bounded flows. The range of turbulence options and the ability for further customization ensure that turbulence for any flow condition can be simulated using ANSYS FLUENT software.

**Acoustics:** Aero-acoustics is an important focus for many industrial applications. In ANSYS FLUENT, the noise resulting from unsteady pressure fluctuations can be computed in several ways. Transient LES predictions for surface pressure can be converted to a frequency spectrum using the built-in fast Fourier transform (FFT) tool. The Ffowcs Williams–Hawkings acoustics analogy can be used to model the propagation of acoustic sources for objects ranging from exposed bluff bodies to rotating fan blades. Broadband noise source models allow acoustic sources to be estimated based on the results of steady-state simulations and, as a result, are practical tools for quickly evaluating design modifications.

**Dynamic and Moving Mesh:** The dynamic mesh capability within ANSYS FLUENT allows engineers to model the arbitrary, complicated motion of parts in challenging applications — such as in internal combustion engines, valves, store separation, ships moving through waves and rocket launches. Dynamic meshing is compatible with a host of other models, including the ANSYS FLUENT suite of spray breakup and combustion models, multiphase flow, free-surface prediction and compressible flow. The sliding mesh and multiple reference frame models have a proven track record for representing the periodic motion inside mixing tanks, pumps and turbomachinery. These moving mesh models are fully compatible with complex models, such as LES, reactions and multiphase flow.

**Heat Transfer, Phase Change and Radiation:** ANSYS FLUENT offers engineers a comprehensive suite of options for modeling convection, conduction and radiation. Models are available for analyzing radiation in environments with optically thick (participating) media, and a view factor-based, surface-to-surface model is available for environments with transparent non-participating media. The discrete ordinates (DO) model is suited for any medium, including glass. A solar load model permits more meaningful climate control simulations. Other capabilities closely associated with heat transfer include models for cavitation, compressible liquids, heat exchangers, shell conduction, real gases, wet steam, and melting and freezing. Evaporation from droplets or wet particles and devolatilization from coal are available with the discrete phase model (DPM). The straightforward addition of heat sources and a complete set of thermal boundary condition options round out the capabilities, making heat transfer modeling in ANSYS FLUENT a mature and reliable tool for any set of needs.

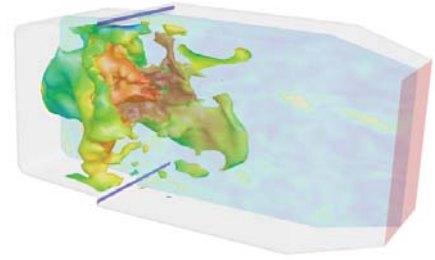
**Reacting Flow:** Comprehensive chemical reaction modeling, especially in turbulent conditions, has been a hallmark of ANSYS FLUENT software since its inception. The eddy dissipation concept, PDF transport and stiff finite rate chemistry models, paired with the proven workhorses of ANSYS FLUENT technology — the eddy dissipation, equilibrium mixture fraction, flamelet and premixed combustion models — are useful for tackling a vast array of gaseous, coal and liquid fuel combustion simulations. Models for reactions between gas and surface species and the prediction of the formation of NO<sub>x</sub>, SO<sub>x</sub> and other pollutants are also widely used and customizable. Reaction models in ANSYS FLUENT can be used in conjunction with the LES and DES turbulence models. When these transient turbulence models are coupled with the reacting flow models, the power to predict flame stabilization and burnout becomes possible.

**Multiphase:** ANSYS FLUENT multiphase modeling technology allows engineers to gain insight into equipment that is often difficult to probe. The Eulerian multiphase model makes use of separate sets of fluid equations for interpenetrating fluids or phases. Special physics are available if one of the fluids is granular as well as to model an interface between fluids. In many cases, the more economical mixture model can be used for both granular and nongranular mixtures. An unlimited number of phases including any combination of liquid, solid and gas can be modeled. As a result, simulations of slurry bubble columns and trickle bed reactors are possible. Heat and mass transfer between phases can be accounted for, making homogeneous and heterogeneous reactions possible. Bubble size distributions can be tracked using integrated population balance models. Both the volume of fluid model and the coupled level-set model are available for free surface flows, such as bubble flow, in which the prediction of the interface is of interest.

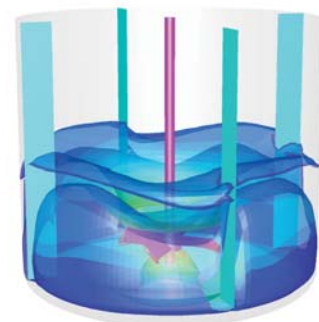
The Discrete phase model, a Lagrangian model, is appropriate for some multiphase applications — such as spray dryers, coal furnaces, continuous fiber drawing and liquid fuel sprays. Injections of particles, bubbles or droplets can undergo heat, mass and momentum transfer with the background fluid.

**Post-Processing:** User-defined functions allow analysts and engineers wanting to customize ANSYS FLUENT software. Add-on modules for applications such as PEM and solid oxide fuel cells as well as

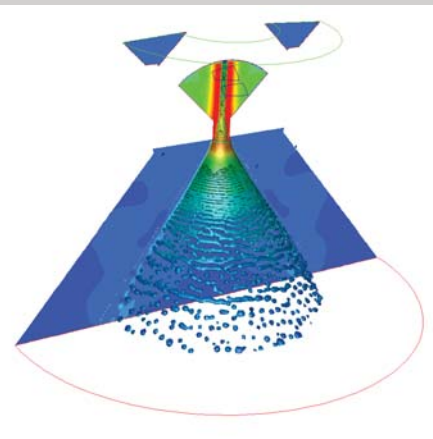
## Industry Solutions



Temperatures on flame surface modeled using LES and state-of-the-art combustion models inside ANSYS FLUENT software



An isosurface of solid concentration in a stirred tank during solids suspension



The predicted spray resulting from atomization. Unstable waves, called Kelvin–Helmholtz waves, are apparent previous to the areas of disintegration ligaments and further into droplets

*Courtesy Bend Research.*

## Industry Solutions

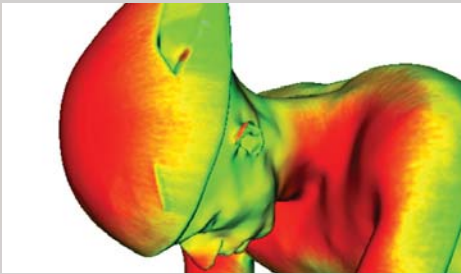


Pathlines of blood flow in an aneurysm during time of average inflow (left). The 3-D model of the cerebral vasculature showing the location of the aneurysm (right)

*Courtesy of The Methodist Hospital Research Institute.*



Contours of pressure on a helicopter in hovering flight mode

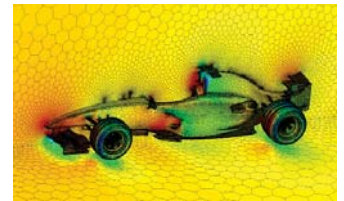


Static pressure contours on a bicycle helmet

*Courtesy: University of Sheffield.*

magnetohydrodynamics are available for many special applications.

**Customized Tools:** Within ANSYS FLUENT, a full suite of qualitative and quantitative post-processing tools can be used to check solution progress and to generate meaningful graphics, animations and reports that make it easy to convey fluids simulation results to engineers and nonengineers alike. The embedded post-processing capability within ANSYS FLUENT works in parallel and can handle even the largest data sets. Solution data can be exported to ANSYS CFD-Post software, third-party graphics packages or CAE packages for additional scrutiny. Solution data also can be mapped to ANSYS® Mechanical™ APDL and third-party FEA meshes for FSI simulation. ANSYS CFD-Post provides extended capabilities including a powerful expression language to derive further quantities from the calculated results, session files and scripting for automation, and templates for automatic report generation incorporating charts, tables and 2-D or 3-D images. Results from multiple simulations can be compared via either side-by-side examination or by calculating differences.



Polyhedral mesh and pressure distribution on an F1 car post-processed using ANSYS CFD-Post software

### Summary

ANSYS FLUENT software provides fast, accurate and robust CFD solutions. Built-in physical models are available for directly simulating a multitude of complex processes. Add-on modules and customization tools allow for more specialized applications. ANSYS is eager to continue its long history of meeting customer needs for a variety of industries and a myriad of applications.

### The ANSYS Advantage

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products 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®

[www.ansys.com](http://www.ansys.com)

ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto: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](mailto:eu.sales@ansys.com)



GSA Contract Holder