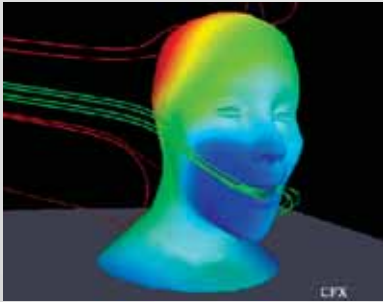
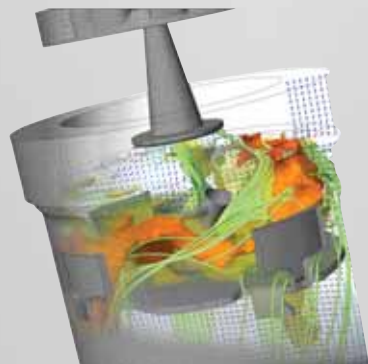


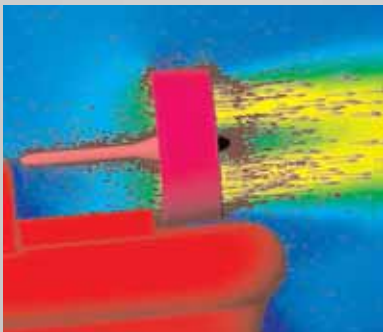
INDUSTRY SOLUTIONS



Ventilation around a human head.



Auxiliary automobile heating.
Courtesy of Webasto

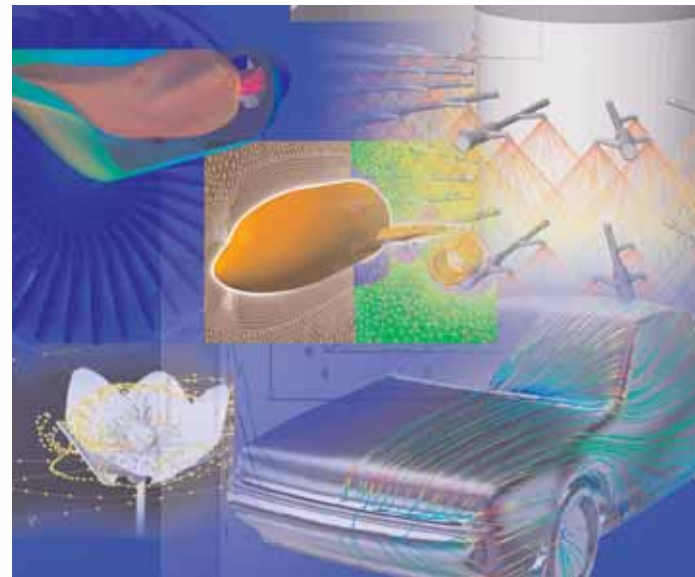


Fan for air cushion vehicle.
Courtesy of CDI Marine

Powerful computational fluid dynamics software for optimization of product development and processes.

Understanding the motion of liquids and gasses is crucial in many branches of engineering. Until recently, studies of fluids in motion were confined to the laboratory, but with the rapid growth in processing power of the personal computer, software applications now bring numerical analysis and solutions of flow problems to the desktop. In addition, the use of common interfaces and workflow processes make fluid dynamics accessible to designers as well as analysts.

Computational Fluid Dynamics (CFD) has become an integral part of the engineering design and analysis environment of many companies that require the ability to predict the performance of new designs or processes before they are ever manufactured or implemented. CFD solutions from ANSYS, Inc. are based on the proven technology of ANSYS® CFX® software. Companies around the world have trusted CFX computational fluid dynamics software to contribute to their success for more than 20 years.



Some pictures courtesy of URS Corporation and University of Canterbury

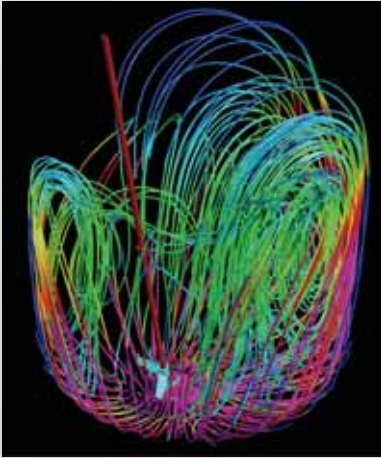
Fluid dynamics is used in industries including aerospace, automotive, chemical processing, power generation, heating, ventilation, air conditioning, biomedical, oil and gas, marine and many others. From ventilation comfort in large buildings to the tiniest scale in micro-pumps and nanotechnology, a wide range of problems can be addressed due to the scalable nature of fluid dynamics. Expertise in assisting companies increase performance through simulation-driven design for pumps, fans, turbines, compressors and other rotating machinery has been incorporated in all elements of ANSYS CFX software, making it a leader in this demanding field. Specialized models for combustion, reacting flows and radiation, among others, help provide the insight into equipment and processes required to increase production, improve longevity and decrease waste.

ANSYS CFX in Workbench

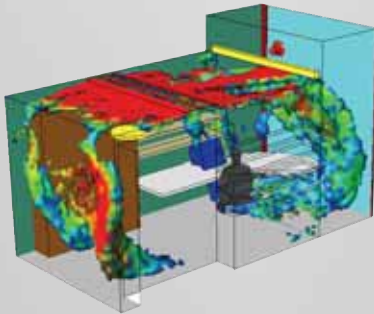
Now CFX technology is available in the ANSYS® Workbench™ interface. Geometry creation, meshing, physics definition, solution and post-processing for CFD are available in a single simulation environment. The project page steps the user through the CFD work flow and makes it easy to learn.

CFX computational fluid dynamics technology is available from ANSYS, Inc. through the CFX product suite. CFD within the Workbench solution provides a common user interface and file structure, allowing easy performance of simulation from geometry to post-processing.

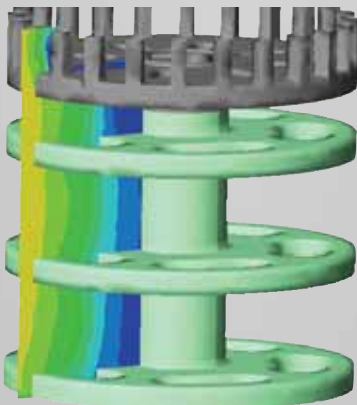
INDUSTRY SOLUTIONS



Mixing vessel.



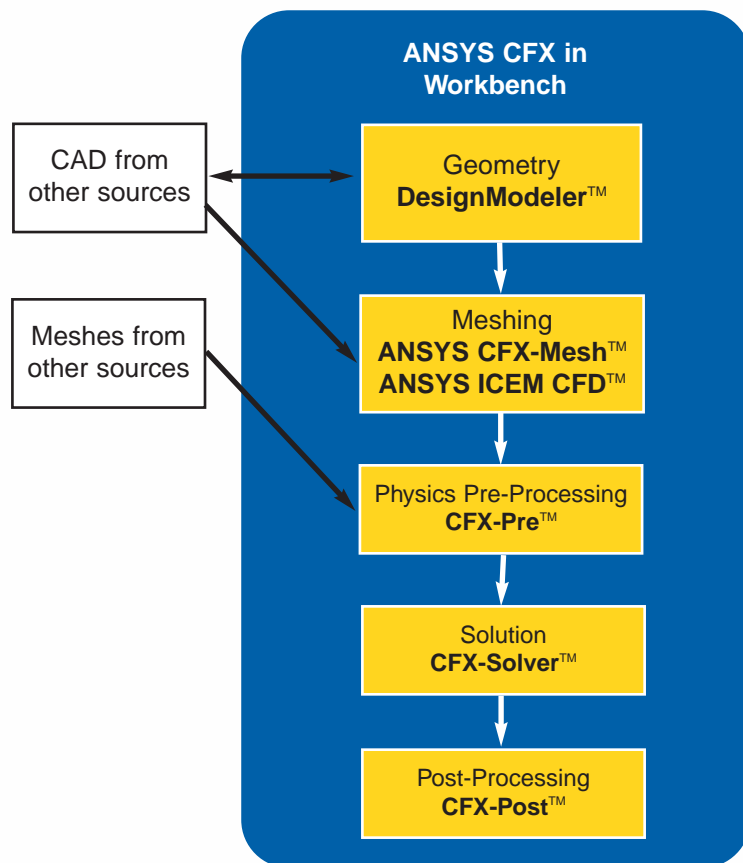
Office ventilation.
Courtesy of Olof Granlund Oy



Grinding mill.
Courtesy of Hatch Australia

Geometry

ANSYS® DesignModeler™ software is a geometry tool specifically designed for the creation and modification of geometry for analysis. Using an advanced system of interfaces, the DesignModeler software gives a direct, bi-directional link to geometry models created in a wide variety of existing CAD packages and is itself an easy-to-use, fully parametric CAD tool. As the geometry portal for all ANSYS products, the DesignModeler software provides a single geometry source for a complete range of engineering simulation tools. The DesignModeler software will help create the detailed geometry required for engineering simulation, minimize geometry rework and simplify interdisciplinary analyses.



Meshing

To provide accurate CFD results requires superior meshing technology. ANSYS, Inc. provides two choices for CFD meshing requirements within the ANSYS® Workbench™ solution, the ANSYS® CFX®-Mesh™ and ANSYS® ICEM CFD™ products.

CFD Pre-Processing

The CFX®-Pre™ module is a modern, consistent and intuitive interface for the definition of the complex physics sometimes required for a CFD analysis. In addition, this tool reads one or more meshes from a variety of sources and provides the user with options for assigning domains.

CFD Solver

The heart of advanced CFD within the ANSYS Workbench interface is the CFX coupled algebraic multigrid solver. Simply put, it achieves reliable and fast convergence by solving the equations well. The solver is fully scalable achieving linear increase in CPU time with problem size, is easy to set up in both serial and parallel run-modes and is representative of true physics. The Solver Manager™ provides feedback on convergence progress, allows dynamic display of many criteria and, when necessary, parameters can be adjusted without stopping the solver so convergence can be accelerated. The ANSYS® CFX® solver runs in the high accuracy mode by default, achieving accurate flow predictions robustly and reliably.

Interoperable Models

The fidelity of simulation is linked directly to the choice of physical models available. The ANSYS CFX software contains a large number of physical models to provide accurate simulation for a wide variety of industrial applications. Because almost all physical models interoperate with each other and in conjunction with all element types, across all grid interface connection types, using the coupled multigrid solver, in parallel, with accurate numerics, the ability to obtain an accurate solution is enhanced greatly. Unique to CFX is the ability to use the 2nd order accurate numerical scheme by default, to deliver more accurate solutions on a given mesh for every simulation.

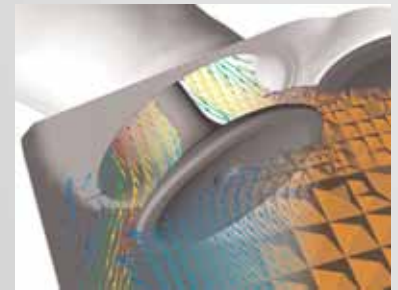
Multiphase: More than 20 years of experience in multiphase modeling is incorporated including models to allow the simulation of multiple fluid streams, bubbles, droplets and free surface flows. The Particle Transport model allows the solution of one or more discrete particle phases within a continuous phase. Transient particle tracking enables the simulation of fire suppression, particulate settling and spray deposition. Particle secondary break-up models capture fragmentation of droplets under the action of external forces. A general framework for Interphase Mass Transfer is also incorporated. The homogeneous MUSIG (multiple size group) model simulates bubble size growth and decay through a series of size groups. Fluidized beds are modeled using a kinetic theory model that includes the effects of inter-particle collisions within dispersed solid phase.

Rotating Machinery: ANSYS CFX software also is recognized as the technology leader in CFD simulation for rotating machinery. All appropriate models for that industry are incorporated and continuously enhanced. In addition, specialized turbomachinery pre- and post-processing ease setup and analysis of results.

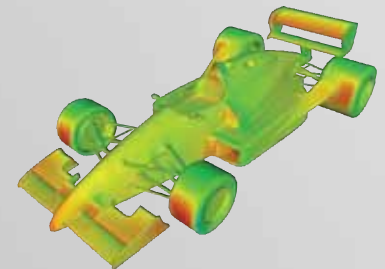
Turbulence: Most industrial flows are turbulent and ANSYS CFX software sets the standard for state-of-the-art turbulence modeling capabilities. A variety of well-established models are available such as $k-\epsilon$ and SST that include the scalable wall function model that ensures solution accuracy is improved with mesh refinement. CFX includes the first commercially available predictive laminar to turbulent flow transition model, known as the Menter-Langtry $\gamma-\theta$ model, that does not require provisions for geometry or grid topology.

Heat Transfer: Optimizing heat transfer between fluids and solids is critical in many types of industrial equipment. ANSYS CFX features the latest technology for solving fluid flows in 3-D domains including conjugate heat transfer for calculation of thermal conduction through solid materials.

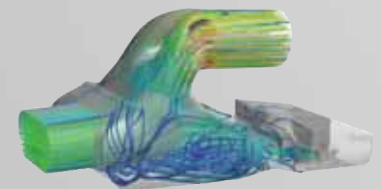
INDUSTRY SOLUTIONS



Automotive intake manifold.
Courtesy of BMW

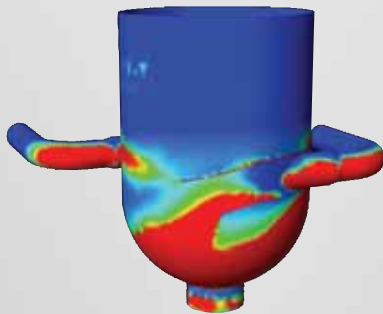


Indy car.



Airplane nacelle.
Courtesy of Bombardier Aerospace

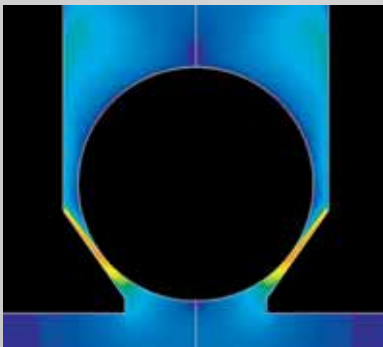
INDUSTRY SOLUTIONS



Feed nozzle for distribution column.
Courtesy of Petrobras



Separator.
Courtesy of Twister BV



Fluid structure interaction in a ball valve.

Porous Media: A true volume porous media model captures the velocity and pressure discontinuity at the interface and produces more accurate and robust solutions than using just a momentum loss model.

Radiation: A wide class of radiative heat transfer models, from transparent to participating non-gray media, are available and include applications such as combustion, heating and ventilation and radiation through solid materials, among others.

Combustion: All species are solved as a single coupled system, accelerating convergence especially for complex reaction mechanisms. Models include multi-step Eddy Break-up, finite rate chemistry, NOx and soot models, as well as state-of-the-art flamelet and Zimont models.

Fluid Structure Interaction: ANSYS, Inc. provides the world's most advanced fluid structure interaction software, permitting combined fluid and solid physics analysis. Our FSI approach preserves the individually validated specialized software components in the computational fluid dynamics and stress analysis disciplines, while at the same time permitting state-of-the-art interaction between the fluid and solid. See the ANSYS FSI Solution brochure for more information.

Moving Mesh: When fluid simulations involve changing geometry (for example, devices such as screw compressors, gear pumps, blood pumps and internal combustion engines), moving mesh may be required. Mesh movement strategies cover almost every conceivable mesh movement need.

CFD Post-Processing

CFX-Post, the powerful CFD post-processing tool, uses an intuitive user interface to represent both graphical and quantitative results. The powerful visualization capabilities of CFX-Post can quickly provide insight into flow field behavior with features such as isosurfaces, slices, vectors, surface plots, animations and streamlines. The quantitative capability allows the user to easily extract values of interest to the designer and analyst that can be used to increase performance and obtain a better understanding. Turbo-Post mode simplifies post-processing for turbomachinery applications. CFX-Post is one of the most powerful CFD post-processors available.

Parallelization

By combining the memory and CPU resources of multiple processors, parallelization allows the user to reduce computation time and perform larger simulations. All physical models, features, modes and options in ANSYS CFX work in parallel, no exceptions.

The superior performance of ANSYS CFX is not based on any single product feature alone. It is the combination of proven, leading-edge technology in all elements of the software that provides the accuracy, reliability, speed and flexibility that companies trust to make them successful.