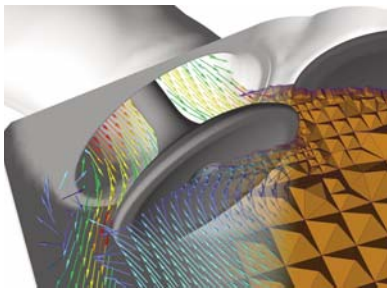
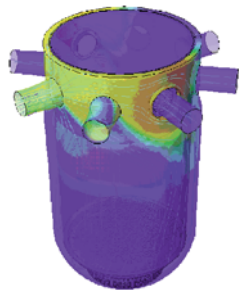


## Industry Solutions



Automotive intake manifold  
*Courtesy BMW AG*



Pressure vessel  
*Courtesy FZ Rossendorf*



Compressor  
*Courtesy Siemens AG*

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

Understanding the motion of liquids and gases 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 computer processing power, software applications now bring numerical analysis and solutions of flow problems to the desktop. In addition, the use of common interfaces and workflow processes makes 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 at many companies that need to predict the performance of new designs or processes before they are manufactured or implemented. For more than 20 years, companies around the world have trusted ANSYS® CFX® technology to contribute to their successes.

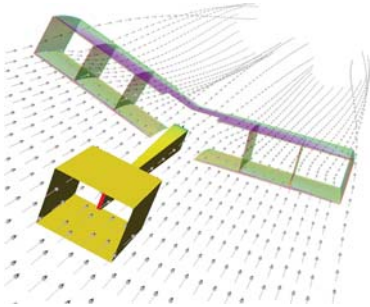
Fluid dynamics is used in industries including aerospace, automotive, chemical processing, power generation, heating, ventilation, air conditioning, biomedical, oil and gas, marine and many others. From ventilation comfort in large buildings to the tiniest scale in micro-pumps and nanotechnology, a wide range of applications can be addressed due to the scalable nature of fluid dynamics. ANSYS has considerable expertise in using simulation-driven design to improve performance for pumps, fans, turbines, compressors and other rotating machinery, and the company has incorporated this into 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 product longevity and decrease waste.

### One Environment

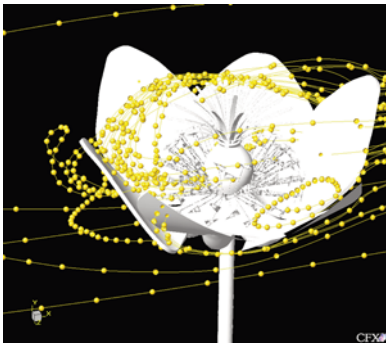
The ANSYS® Workbench™ environment provides a single setting for simulation from start to finish, enabling users to perform more product development tasks faster. ANSYS Workbench delivers the basis for a full computer-aided engineering (CAE) solution from ANSYS, providing access to a wide variety of simulation technologies.

All settings are persistent and are connected back to the parametric computer-aided design (CAD) model, from analysis-specific modifications made to the geometry through the application of physics, solver control parameters, graphic objects created during post-processing and quantitative expressions evaluating performance.

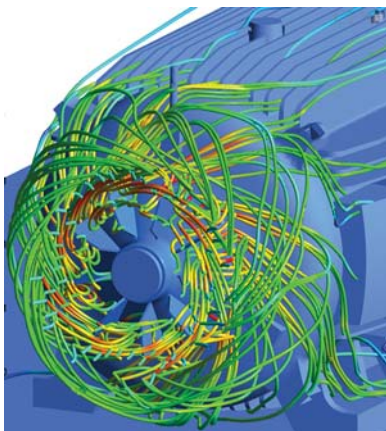
### Industry Solutions



Historic aircraft  
Courtesy CFD 14-Bis project



Kiwi flower pollen  
Courtesy University of Canterbury



Electric motor  
Courtesy WEG Motors

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

**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 and NO<sub>x</sub> and soot models; in addition, state-of-the-art flamelet and Zimont models are available.

**Fluid Structure Interaction:** ANSYS, Inc. provides the world's most advanced fluid structure interaction (FSI) software, permitting combined fluid and solid physics analysis. The ANSYS FSI approach preserves the individually validated specialized software components in CFD and stress analysis disciplines while, at the same time, permitting state-of-the-art interaction between the fluid and solid. Both one-way and two-way FSI simulations are possible, from problem setup to post-processing, all within the ANSYS Workbench environment. (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

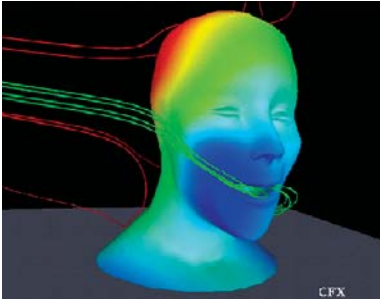
The CFD post-processing tool uses an intuitive user interface to represent both graphical and quantitative results. The powerful visualization capabilities of ANSYS CFX software can quickly provide insight into flow field behavior with features such as isosurfaces, slices, vectors, surface plots, animations and streamlines. The quantitative capability allows the user to easily extract values of interest to the designer and analyst, which can be used to increase performance and obtain better understanding. The Turbo-Post mode simplifies post-processing for turbomachinery applications and includes a set of template reports based on machine type. The post-processor includes automated report generation. Report templates can be established that include all charts, tables and figures, which then can be re-used for each design to allow easy comparison of design alternatives. ANSYS CFX technology includes one of the most powerful CFD post-processors available.

### Parallelization

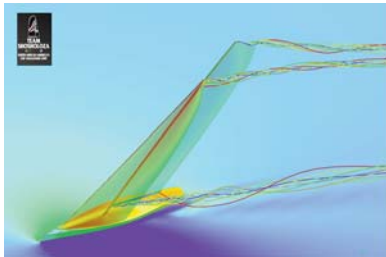
By combining the memory and CPU resources of multiple processors, parallelization allows the user to reduce computation time and perform larger simulations. All physical models, features, modes and options in ANSYS CFX software work in parallel, without exception.

*The superior performance of ANSYS CFX software is not based on any single product feature. 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.*

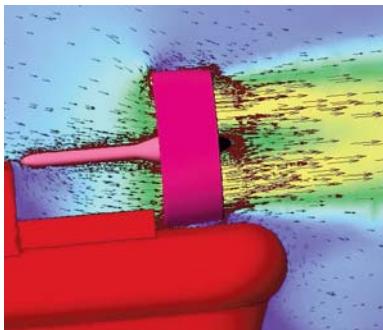
## Industry Solutions



Ventilation around a human head



Yacht



Fan for air-cushion vehicle  
Courtesy CDI Marine

## ANSYS CFX in ANSYS Workbench Delivers Fast Design Iteration and Parametric Studies

The parameter manager of the ANSYS Workbench environment enables setup of a series of simulations to study the operating range of a product or to investigate and compare several alternative designs. The parameterized geometry and physics description, combined with the automatic calculation of performance metrics, allows the operating range of the product or process to be established quickly. The geometry, meshing, physics specification, solution and report generation sequence is run automatically over the range and sampling frequency of the defined parameters. Designers and analysts now are able to minimize physical prototyping and deliver better, more innovative products faster than ever. Design of Experiments as well as deterministic and robust design optimization techniques now can be applied to fluids analysis using ANSYS® DesignXplorer™ technology.

### 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, ANSYS DesignModeler software gives a direct, bi-directional link to geometry models created in a wide variety of existing CAD packages. It is an easy-to-use, fully parametric CAD tool. As the geometry portal for all ANSYS products, ANSYS DesignModeler software provides a single geometry source for a complete range of engineering simulation tools. ANSYS DesignModeler helps to create the detailed geometry required for engineering simulation, minimizes geometry rework and simplifies interdisciplinary analyses.

### Meshing

Providing accurate CFD results requires superior meshing technology. The meshing application within the ANSYS Workbench environment provides access to swept, hex-dominant, tetrahedral and prism meshing technologies in a single location that can be applied on a part-by-part basis. ANSYS® ICEM CFD™ meshing tools also are available and include mesh editing capabilities as well as structured hexahedral meshing.

### CFD Pre-Processing

The ANSYS CFX physics pre-processor is a modern, consistent and intuitive interface for the definition of the complex physics sometimes required for 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 ANSYS CFX within the ANSYS Workbench interface is the coupled algebraic multigrid solver. In brief, it achieves reliable and fast convergence by solving the equations accurately. 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 and allows dynamic display of many criteria. When necessary, parameters can be adjusted without stopping the solver so convergence can be accelerated. The ANSYS CFX solver runs in 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. 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 in conjunction with all element types and across all grid interface connection types, the ability to obtain an accurate solution is greatly enhanced. Unique to ANSYS CFX is the ability to use the second-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, resulting in 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 breakup models capture fragmentation of droplets under the action of external forces. A general framework for Interphase Mass Transfer also is included. The homogeneous multiple size group (MUSIG) 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.

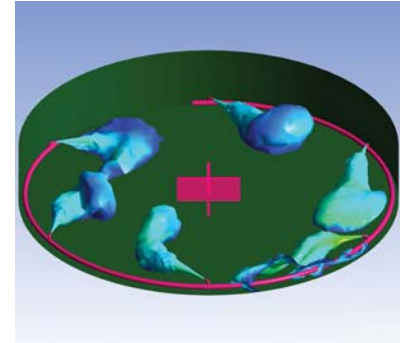
**Rotating Machinery:** ANSYS CFX software is recognized as the technology leader in CFD simulation for rotating machinery. All appropriate models for that industry are incorporated and enhanced continuously. In addition, specialized turbomachinery pre- and post-processing ease setup and analysis of 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, which both include the scalable wall function model, ensuring that solution accuracy is improved with mesh refinement. ANSYS CFX software 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 and the Scale Adaptive Simulation™ (SAS) model for unsteady CFD simulations with flows in which steady-state simulations are not sufficiently accurate and do not properly describe the true nature of the physical phenomena.

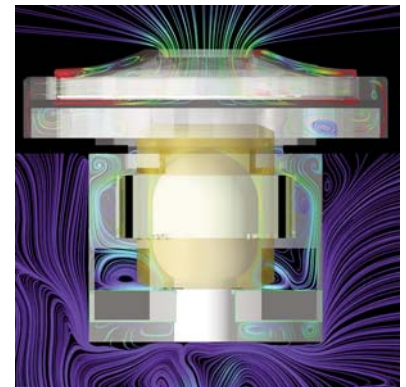
**Heat Transfer:** Optimizing heat transfer between fluids and solids is critical in many types of industrial equipment. ANSYS CFX software features the latest technology for solving fluid flows in 3-D domains, including conjugate heat transfer for calculation of thermal conduction through solid materials.

**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 a momentum-loss model.

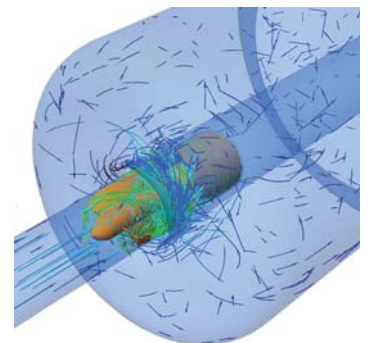
### Industry Solutions



Mixer



Vacuum cleaner fan  
*Courtesy LSTM-Erlangen*



Cardiac pump  
*Courtesy Department of Biomedical Engineering/Lerner Research Institute/The Cleveland Clinic Foundation*