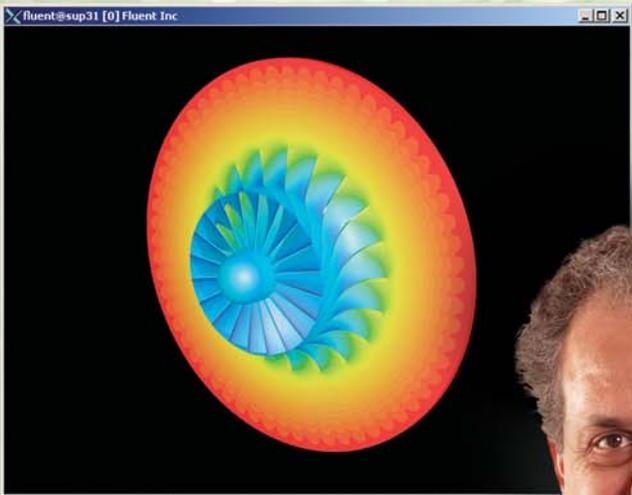


CFD

# FLUENT Flow Modeling Software



FLUENT@sup31 [0] Fluent, Inc

File	Grid	Define
225612	hexahedr	
130092	mixed ce.	
8982	mixed int	
375847	mixed int	
2469	mixed wa.	
3996	quadrilat	
8550	quadrilat	
8550	quadrilat	
1216	quadrilat	
182	quadrilat	
240	quadrilat	
2062	quadrilat	
3064	quadrilat	
9570	quadrilat	
662528	mixed int	
288	quadrilat	
271	quadrilat	
3064	quadrilat	
2062	quadrilat	
8982	interfac	
864	child anc	
813	child anc	
8699	shape noc	
8699	shape noc	

- ▼ Laminar
- ▼ Spalart-Allmaras (1 eqn)
- ▼ k-epsilon (2 eqn)
- ▼ k-omega (2 eqn)
- ▼ Reynolds Stress (7 eqn)
- ▼ Detached Eddy Simulation
- ▼ Large Eddy Simulation

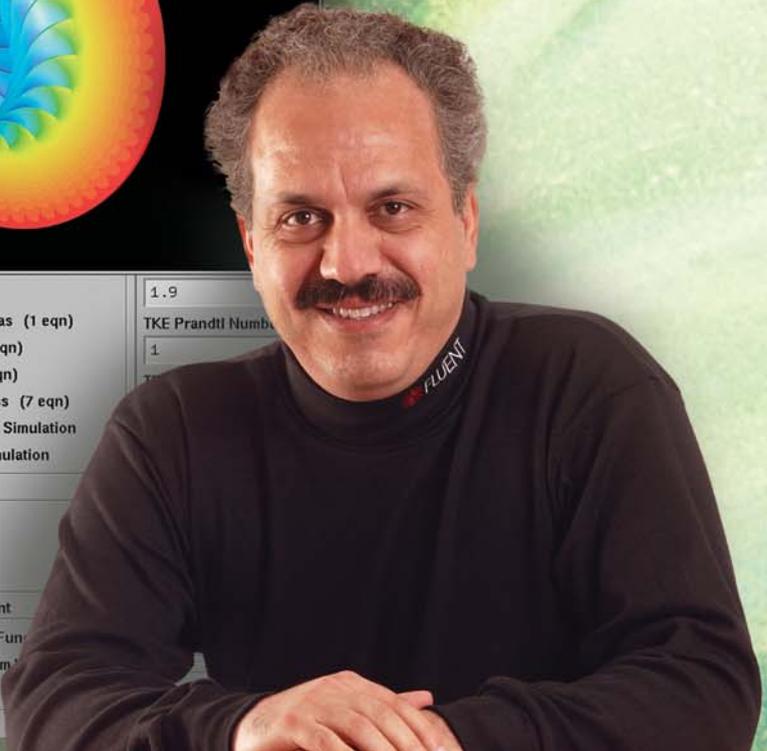
k-epsilon Model

- ▼ Standard
- ▼ RNG
- ▼ Realizable

Near-Wall Treatment

- ▼ Standard Wall Fun
- ▼ Non-Equilibrium
- ▼ Enhanced Wall

Options



0110 0010101 0010101001 001 11001011  
0110 0010101 0010101001 001 110010110



**FLUENT**  
The Right Answer in CFD

FLUENT. For over twenty years the name has been synonymous with the world's leading commercial software for solving fluid flow problems. Based on a numerical technique called computational fluid dynamics, or CFD, FLUENT has a proven track record for delivering fast, accurate, and meaningful solutions to engineers in dozens of industries worldwide.

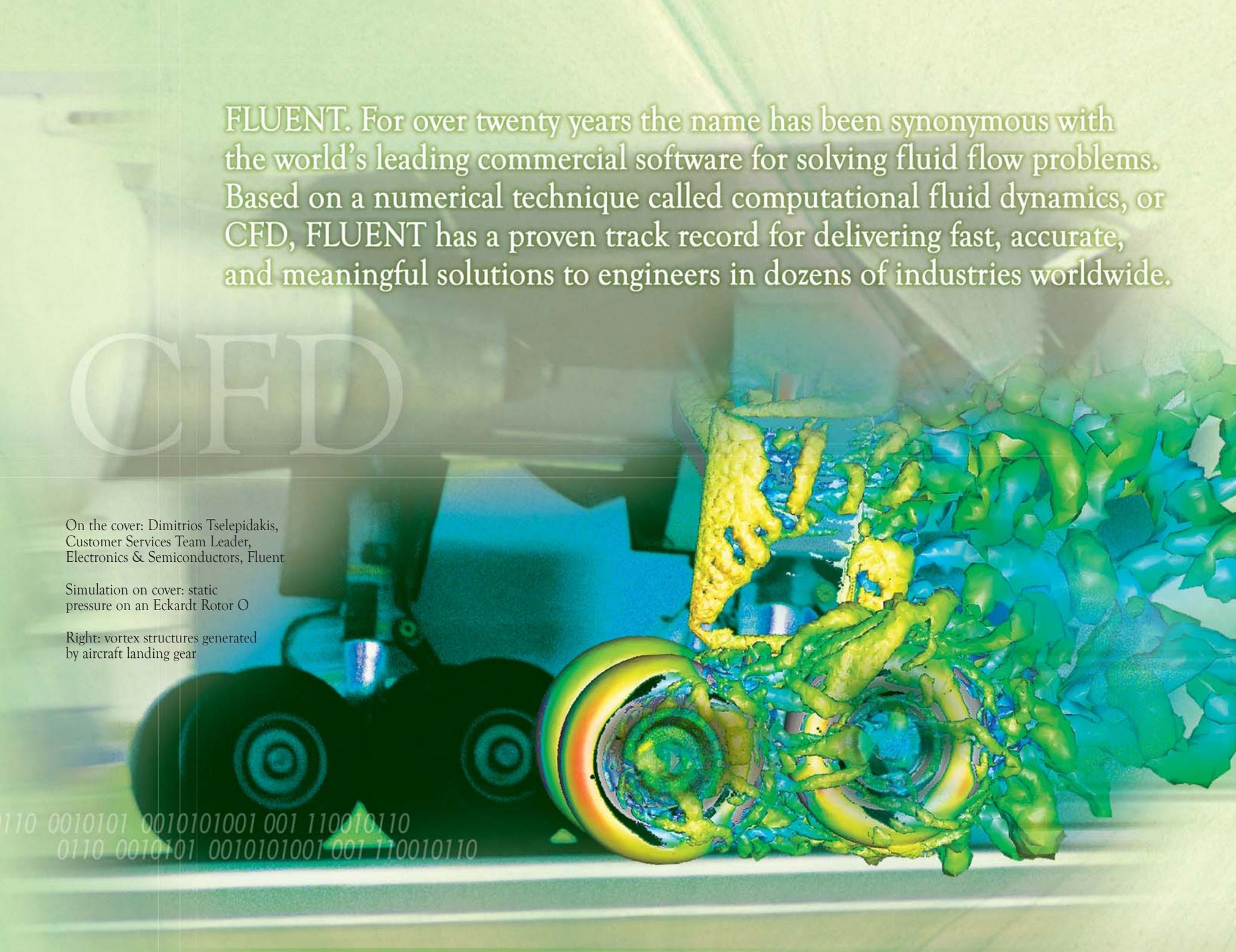
# CFD

On the cover: Dimitrios Tselepidakis,  
Customer Services Team Leader,  
Electronics & Semiconductors, Fluent

Simulation on cover: static  
pressure on an Eckardt Rotor O

Right: vortex structures generated  
by aircraft landing gear

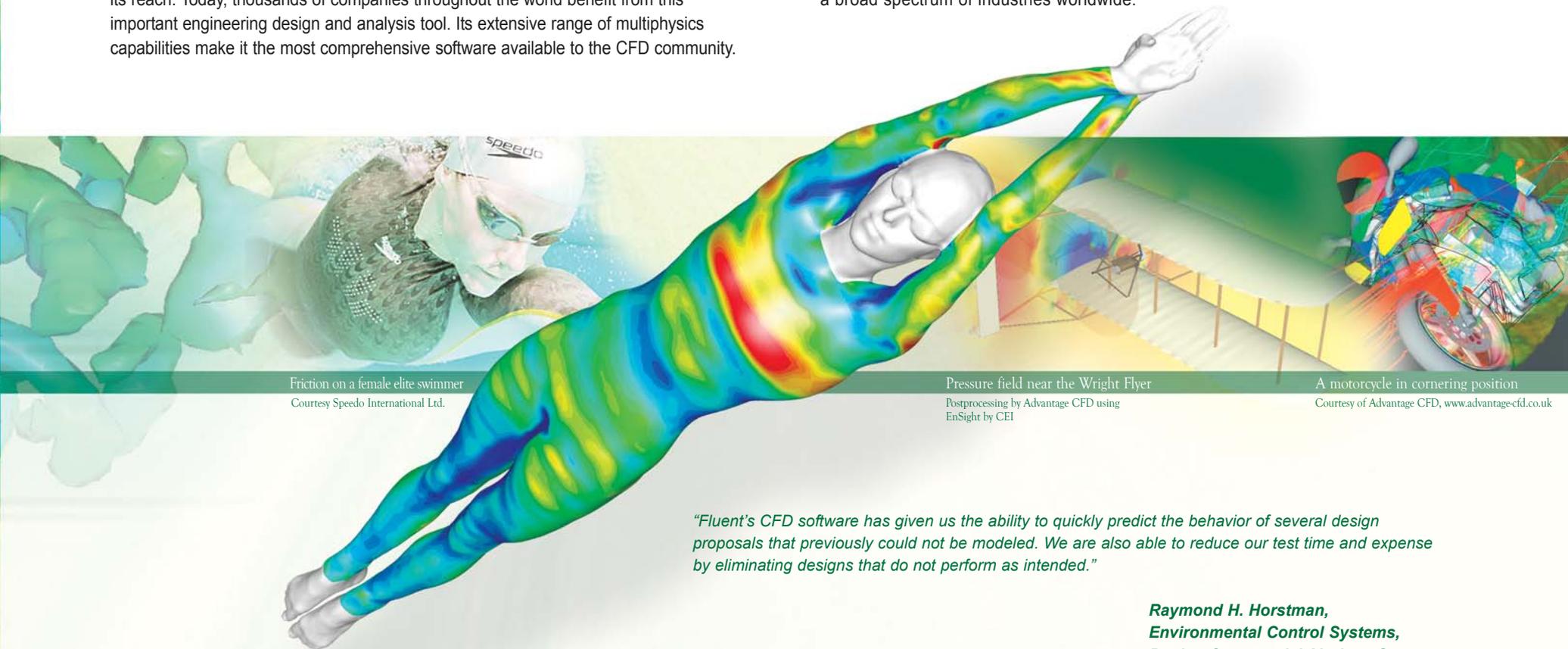
110 0010101 0010101001 001 110010110  
0110 0010101 0010101001 001 110010110



# The Right Answer in CFD

When people think of CFD, the name FLUENT comes to mind. The broad physical modeling capabilities of FLUENT have been applied to industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to glass production, from blood flow to semiconductor manufacturing, from clean room design to wastewater treatment plants. The ability of the software to model in-cylinder engines, aeroacoustics, turbomachinery, and multiphase systems has served to broaden its reach. Today, thousands of companies throughout the world benefit from this important engineering design and analysis tool. Its extensive range of multiphysics capabilities make it the most comprehensive software available to the CFD community.

With its long-standing reputation of being user-friendly and robust, FLUENT makes it easy for new users to come up to productive speed. Most importantly, technical support is top-notch, from the initial training to the ongoing relationship that develops between customers and Fluent staff. Throughout the years, these important components—comprehensive models, usability, and personalized technical support—have combined to make FLUENT the CFD software of choice across a broad spectrum of industries worldwide.



Friction on a female elite swimmer  
Courtesy Speedo International Ltd.

Pressure field near the Wright Flyer  
Postprocessing by Advantage CFD using  
EnSight by CEI

A motorcycle in cornering position  
Courtesy of Advantage CFD, [www.advantage-cfd.co.uk](http://www.advantage-cfd.co.uk)

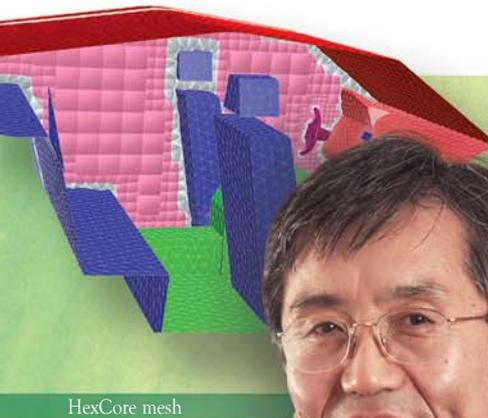
*"Fluent's CFD software has given us the ability to quickly predict the behavior of several design proposals that previously could not be modeled. We are also able to reduce our test time and expense by eliminating designs that do not perform as intended."*

**Raymond H. Horstman,**  
**Environmental Control Systems,**  
**Boeing Commercial Airplane Group**

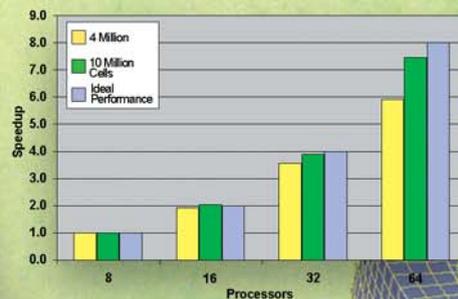
# Grids, Numerics, Parallel Processing

At the core of any CFD calculation is a computational grid, used to divide the solution domain into thousands or millions of elements where the problem variables are computed and stored. In FLUENT, unstructured grid technology is used, which means that the grid can consist of elements in a variety of shapes: quadrilaterals and triangles for 2D simulations, and hexahedra, tetrahedra, prisms, and pyramids for 3D simulations. These elements, created using automated controls in GAMBIT, FLUENT's companion preprocessor, form an interlocking network throughout the volume where the fluid flow analysis takes place.

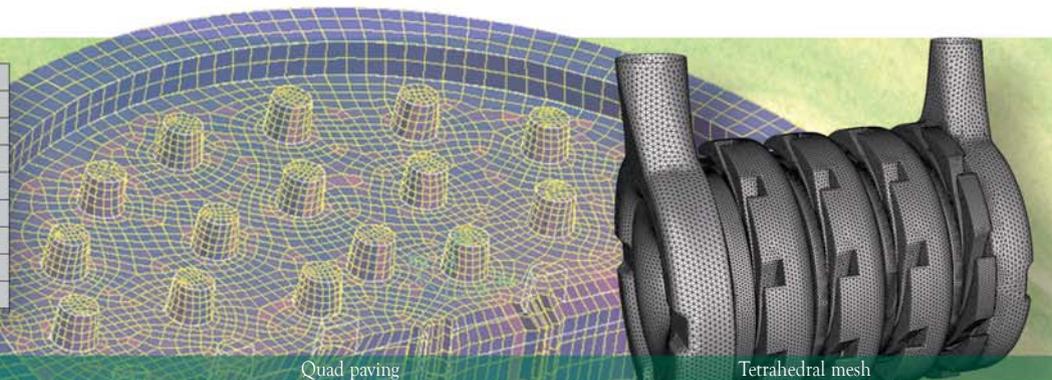
In the commercial CFD market today, no other provider offers more complex physical models on an unstructured grid than FLUENT. Sophisticated numerics and a robust solver ensure accurate results. The new non-iterative time advancement scheme has greatly reduced the time-to-solution for transient simulations. Mature parallel processing capabilities, available on NT, Linux, and Unix platforms, can be used on multiple processors of a single machine or 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.



HexCore mesh

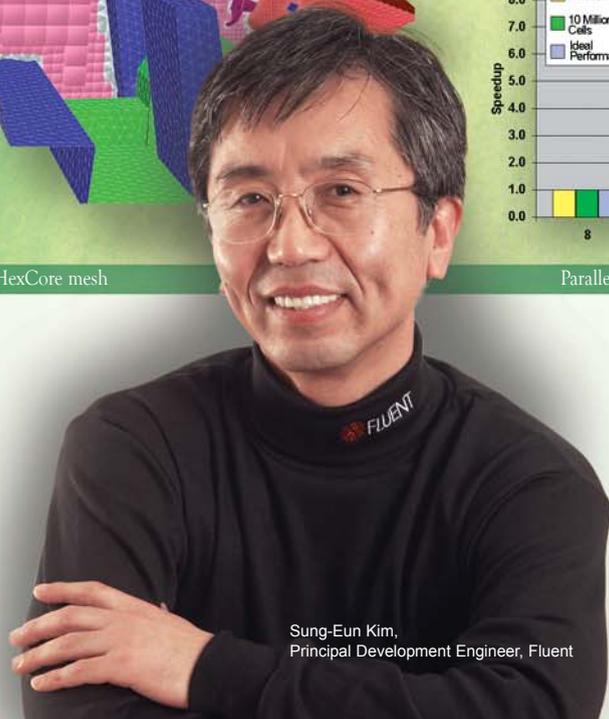


Parallel scalability for two FLUENT test cases



Quad paving

Tetrahedral mesh



Sung-Eun Kim,  
Principal Development Engineer, Fluent

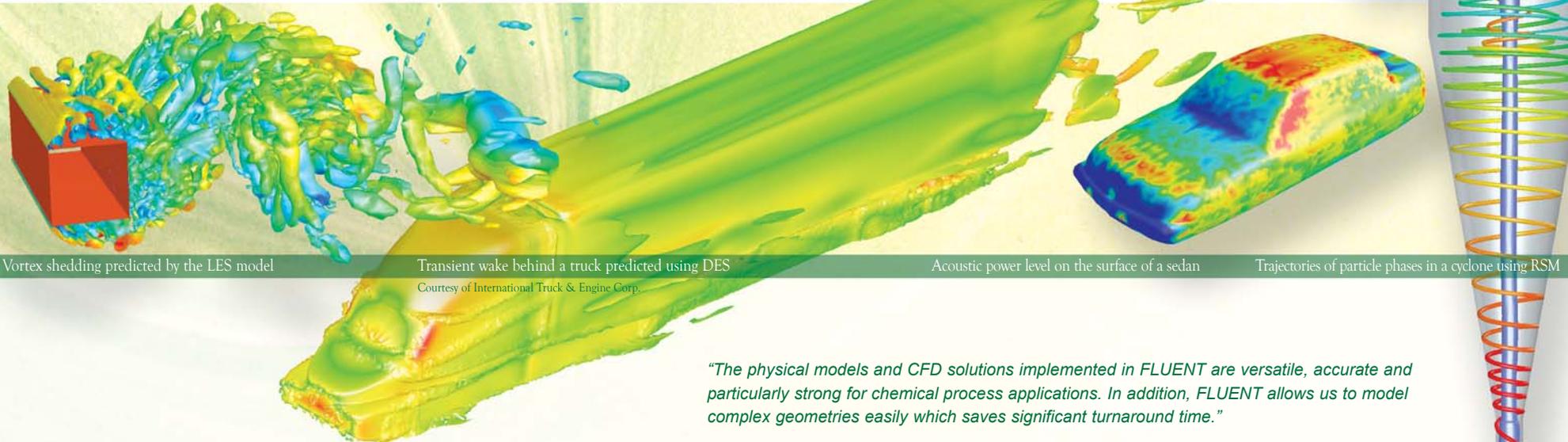
*"Using Fluent's preprocessing software, we made light work of our client's complex building geometry in a 2 million cell simulation. Solving this CFD model on an NT Cluster allowed us to predict the effects of solar radiation on occupant comfort. Using FLUENT, we have been able to concentrate more on analyzing results than ever before."*

**Matthew Kitson,**  
**Environmental Modeling,**  
**Hilson Moran Partnership Ltd.**

# Turbulence and Acoustics

FLUENT has always defined the cutting edge of turbulence modeling in commercial CFD software, offering an unparalleled breadth of models. Several versions of the heavily used and time-honored k- $\epsilon$  model are available, as is the Reynolds stress model (RSM) for highly swirling or anisotropic flows. Recent increases in computer power, coupled with decreases in cost, have combined to make large eddy simulation (LES) and the more economical detached eddy simulation (DES) turbulence models very attractive for industrial applications. Other models round out the offerings so that any flow condition can be simulated. Wall functions and enhanced wall treatment options allow for the best possible representation of all wall-bounded flows.

Aeroacoustics is an important focus for many industrial applications, but has traditionally been difficult to simulate. In FLUENT, the noise resulting from unsteady pressure fluctuations can now 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, so are practical tools for quickly evaluating design modifications.



Vortex shedding predicted by the LES model

Transient wake behind a truck predicted using DES  
Courtesy of International Truck & Engine Corp.

Acoustic power level on the surface of a sedan

Trajectories of particle phases in a cyclone using RSM

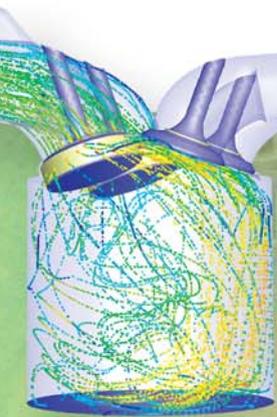
*"The physical models and CFD solutions implemented in FLUENT are versatile, accurate and particularly strong for chemical process applications. In addition, FLUENT allows us to model complex geometries easily which saves significant turnaround time."*

**Hiroaki Takehara,**  
Science and Technology Research Center,  
Mitsubishi Chemical Corporation

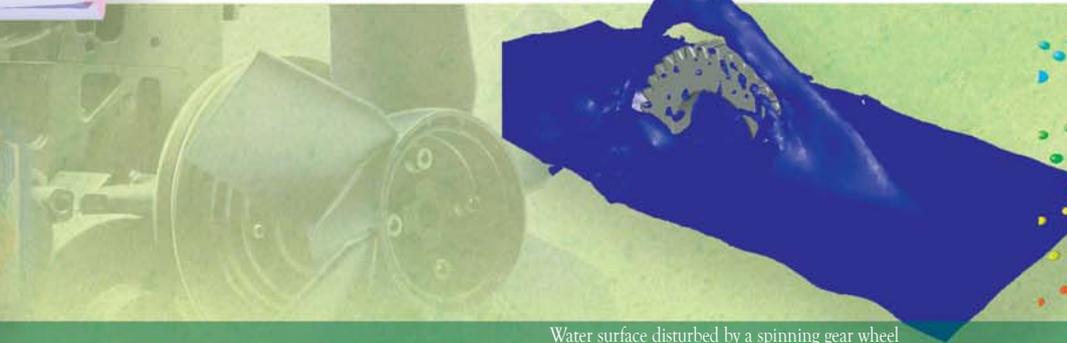
# Dynamic and Moving Mesh

Internal combustion engines, valves, store separation, and rocket launches are examples of problems where moving parts define the fluid flow. In FLUENT, the dynamic mesh capability meets the needs of these challenging applications. It has several different mesh rebuilding schemes, which can be used for different moving parts within the same simulation, as needed. Only the initial mesh and a description of the boundary movement are required. Dynamic meshing is compatible with a host of other models, including FLUENT's suite of spray breakup and combustion models, multiphase flow, free surface prediction, and compressible flow.

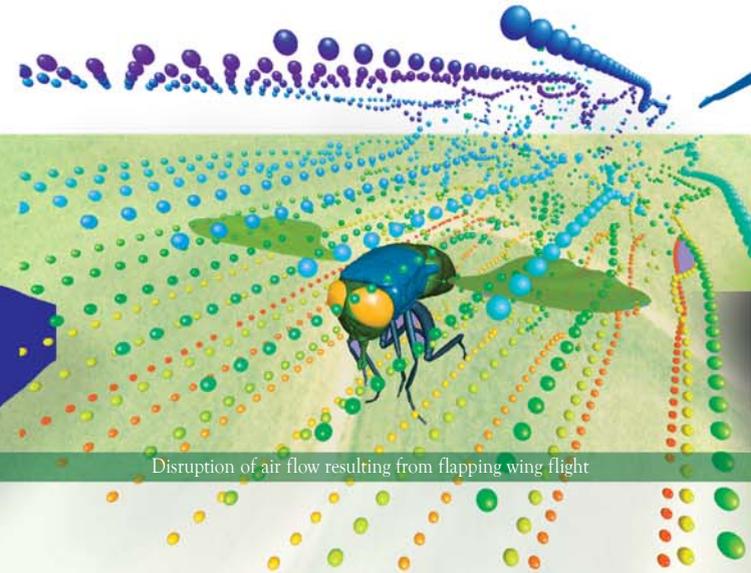
The periodic motion inside mixing tanks, pumps, and turbomachinery has traditionally been simulated with moving mesh models in FLUENT. The sliding mesh and multiple reference frames models have a proven track record and are fully compatible with other relevant models, such as LES, reacting, and multiphase flow.



Internal combustion engine



Water surface disturbed by a spinning gear wheel



Disruption of air flow resulting from flapping wing flight

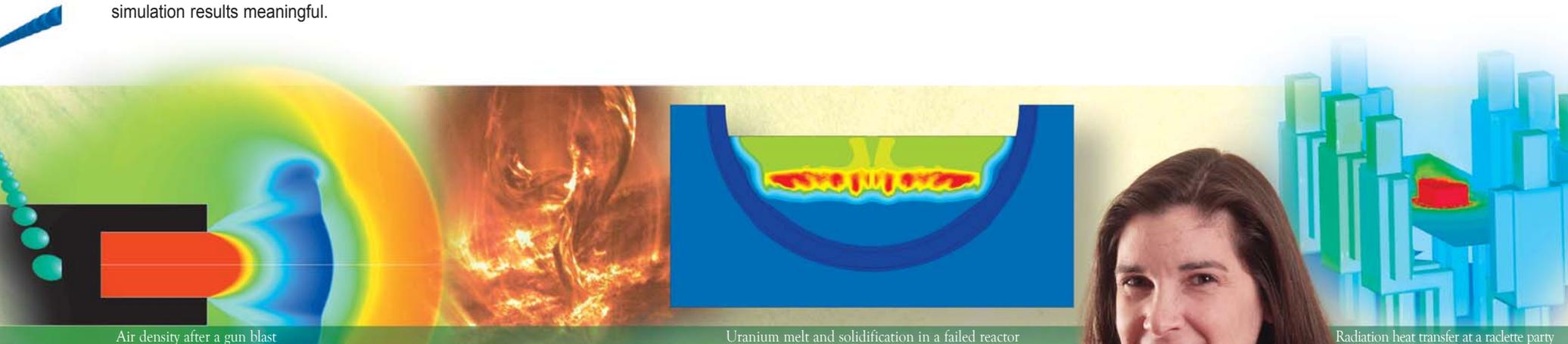
Background photo courtesy of Deutz AG

In-Cylinder . Store Separation . Passing Vehicles . Valves  
Pumps . Turbomachinery . Mixing Tanks

# Heat Transfer, Phase Change, Radiation

Heat transfer accompanies many fluid flow phenomena and FLUENT offers a comprehensive suite of options for convection, conduction, and radiation. For radiation, the P1 and Rosseland models are available for environments with optically thick (participating) media, and a view-factor-based surface-to-surface model is available for environments with non-participating media. The discrete ordinates (DO) model is suited for any medium, including glass. A general-purpose discrete ray tracing model (DTRM) is also available. A solar load model makes use of a ray-tracing algorithm and includes a solar calculator. It allows for visualization of illuminated and shadowed areas, making climate control simulation results meaningful.

Other capabilities closely associated with heat transfer include models for cavitation, compressible liquids, heat exchangers, shell conduction, real gas, and wet steam. The phase change model tracks melting and freezing in the bulk fluid. 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 FLUENT a mature and reliable tool to meet any set of needs.



Air density after a gun blast  
Courtesy of Benet Laboratories

Courtesy of TRACE/NASA Small Explorer Program

Uranium melt and solidification in a failed reactor

Radiation heat transfer at a raclette party

Heat Exchangers . Compressible Flow . Melting  
Freezing . Evaporation . Boiling

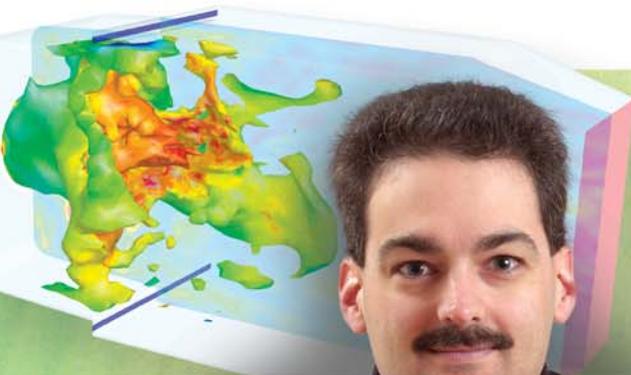
Nicole Diana,  
FLUENT Product Manager, Fluent

# Reacting Flow

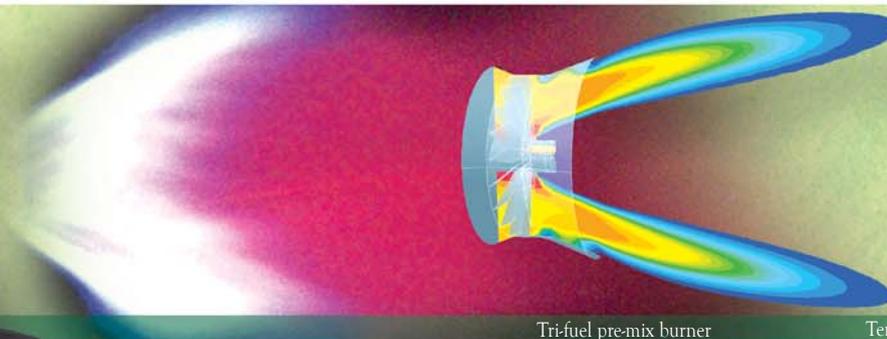
Chemical reaction modeling, especially in turbulent conditions, has been a hallmark of FLUENT software since its inception. Over the years, the comprehensive reaction modeling capabilities in FLUENT have helped engineers meet the diverse challenges of difficult combustion processes. The eddy dissipation concept, PDF transport, and stiff finite rate chemistry models have recently joined FLUENT's proven workhorses: the eddy dissipation, equilibrium mixture fraction, flamelet, and premixed combustion models for tackling a vast array of gaseous, coal, and liquid fuel combustion simulations. Models for the prediction of NOx formation are also widely used and customizable.

Many industrial applications involve reactions that take place on solid surfaces. FLUENT's surface reaction capability allows for reactions between gas and surface species as well as between different surface species, so that deposition and etching can be rigorously predicted. Simulations of catalytic converters, gas reformers, pollution control devices, and semiconductor manufacturing have all benefited from this technology.

FLUENT's reaction models 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.



Temperatures on a flame surface



Courtesy of Astec Industries

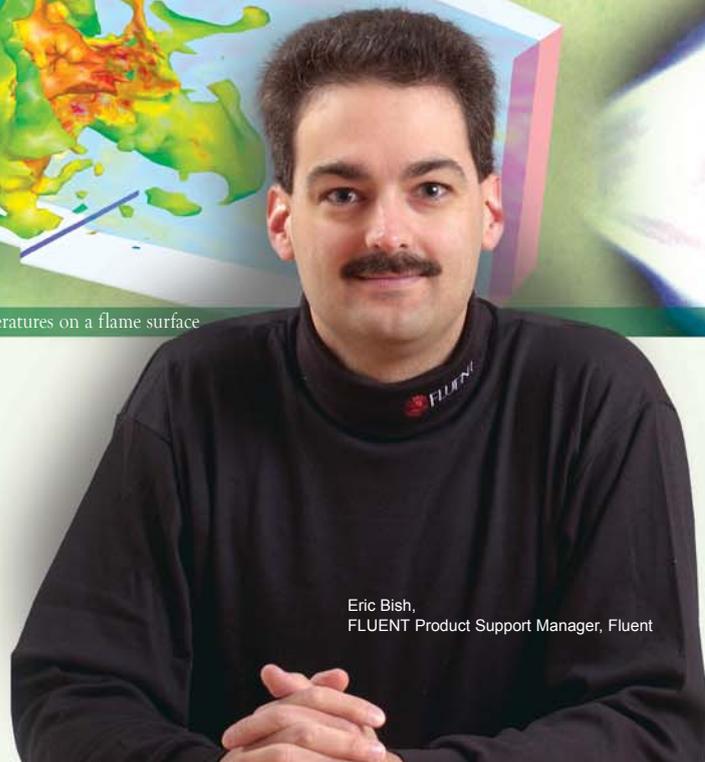
Tri-fuel pre-mix burner

Courtesy of Astec Industries



Temperatures in selective catalytic reduction ductwork

Courtesy of Babcock Power



Eric Bish,  
FLUENT Product Support Manager, Fluent

Gas Turbines . Boilers . Furnaces . Chemical Vapor Deposition  
Erosion . Emissions . Flares . Etching . Catalytic Converters

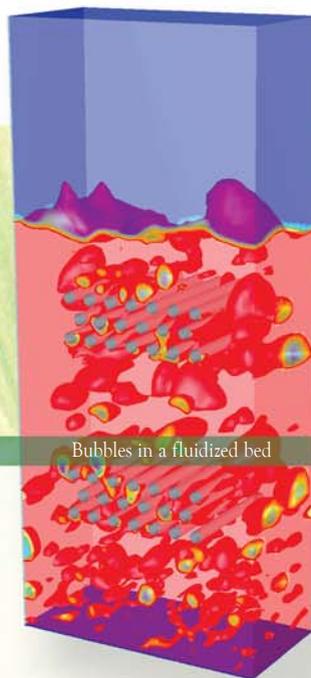
# Multiphase

Multiphase mixtures are widespread in industrial applications, and FLUENT is a leader in multiphase modeling technology. Its varied capabilities allow 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. In many cases, the more economical mixture model can be used for granular and non-granular mixtures as well. Three-phase mixtures (liquid, granular, and gas) can be modeled, so simulations of slurry bubble columns and trickle bed reactors are possible. Heat and mass transfer between phases can take place, making homogeneous and heterogeneous reactions possible.

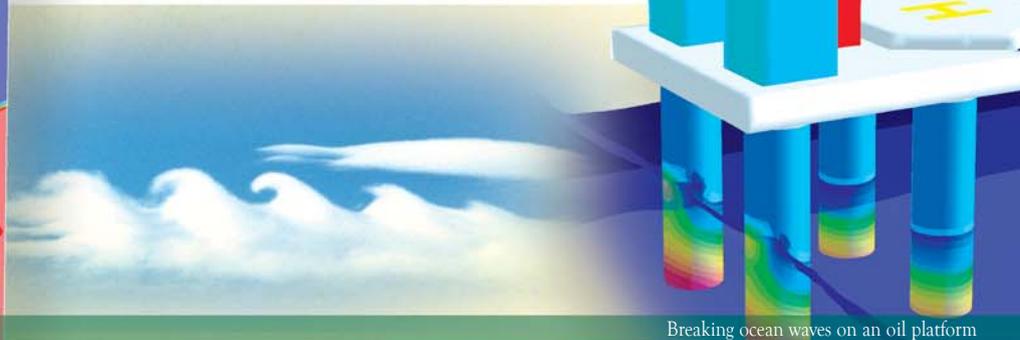
Several other multiphase models are also standard in FLUENT. For some multiphase applications, such as spray dryers, coal furnaces, and liquid fuel sprays, the discrete phase model (DPM) can be used. Injections of particles, bubbles, or droplets can undergo heat, mass, and momentum transfer with the background fluid. The volume of fluid model is available for free surface flows, such as ocean waves, where the prediction of the interface is of interest. The cavitation model has proven useful for modeling hydrofoils, pumps, and fuel injectors. Boiling can be implemented through readily available user-defined functions.



Cavitation in an automotive fuel pump



Bubbles in a fluidized bed



Copyright photo by Brooks Martner, NOAA/ETL

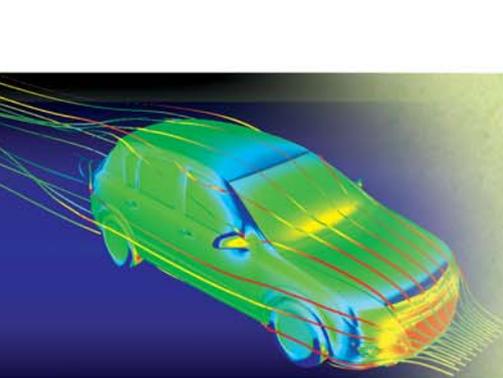
Breaking ocean waves on an oil platform

Bubble Columns . Fluidized Beds . Risers . Water Waves . Tank Sloshing . Separators  
Spray Breakup . Cavitation . Boiling . Coal Burners . Spray Dryers

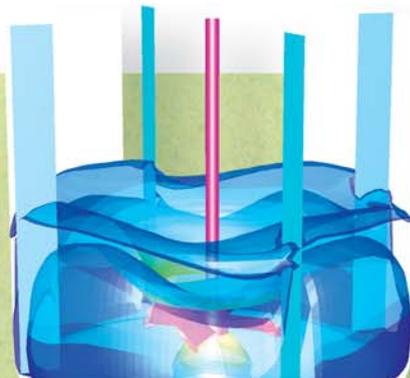
# Pre- and Postprocessing

Fluent offers software for the creation of problem geometries and grids. GAMBIT allows users to build geometries using basic construction tools. Alternatively, it can be used to read CAD files and condition the imported geometry for CFD analysis. For fast, flexible meshing, Fluent offers GAMBIT as well as TGrid, a state-of-the-art volume mesher. Both tools have many automated features for building or joining hybrid meshes with attention to boundary layers, nonuniform sizing, and core regions of hexahedral cells. For turbomachinery applications, G/Turbo is available. This preprocessing tool uses familiar terminology and parametric templates to assist in problem setup.

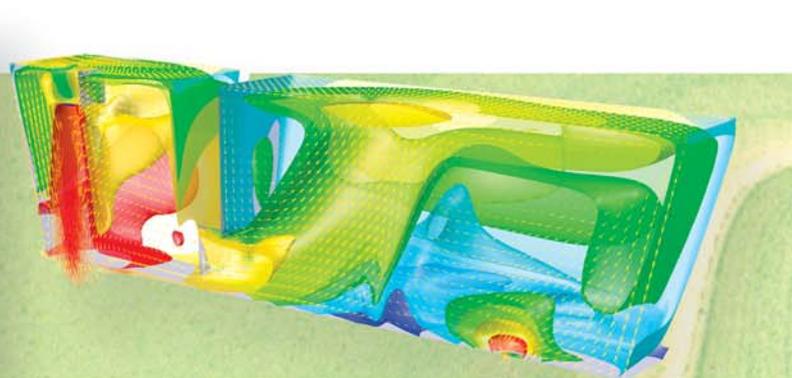
FLUENT's postprocessing tools can be used to generate meaningful graphics, animations, and reports that make it easy to convey CFD results to engineers and non-engineers alike. Shaded and transparent surfaces, pathlines, and scene construction are just some of the postprocessing features that make FLUENT graphics unique. Solution data can be exported to third party graphics packages, or can be exported to CAE packages for additional analysis.



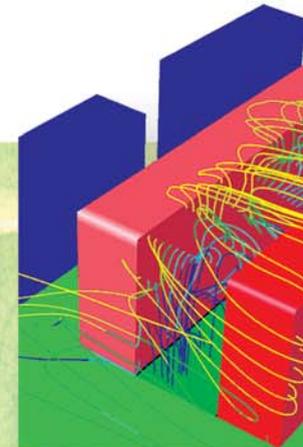
Flow around an Opel Astra  
Courtesy of Adam Opel AG



Solids suspension in a stirred tank



Turbulent kinetic energy in a steel tundish



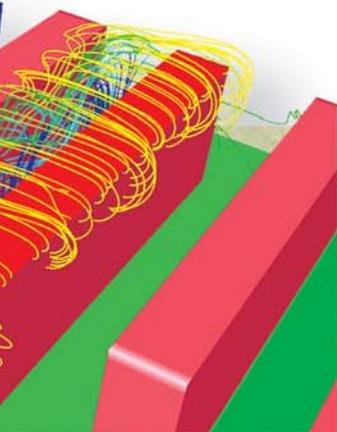
*"For a technology-based company like Norsk Hydro, CFD analysis using FLUENT has become an essential and accepted tool for problem analysis and optimization. FLUENT is used in all kinds of studies, from ventilation simulations via mixing analysis to complex multiphase separator calculations. We have come to appreciate the variety of models and the flexibility of the code. For us, FLUENT has proven to bring added value to the modeling process."*

**Eirik Manger,**  
Corporate Research Centre,  
Norsk Hydro ASA

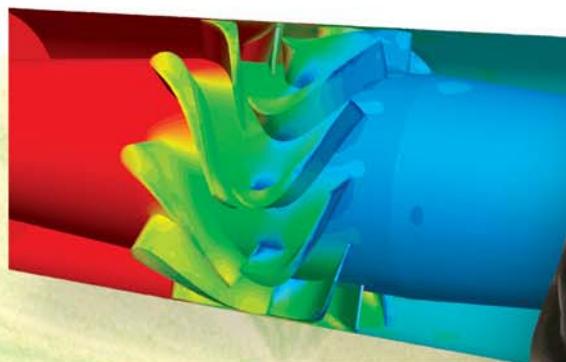
# Customized Tools

User-defined functions are a popular option for users wanting to customize FLUENT. Comprehensive documentation and a number of tutorials are available, as is full technical support. Fluent's global consulting network can provide or help create templates for the repeated setup of any type of equipment, such as cyclone separators, automotive HVAC systems, and furnaces. Add-on modules for many special applications are available, such as PEM and solid oxide fuel cells, magnetohydrodynamics, and continuous fiber drawing.

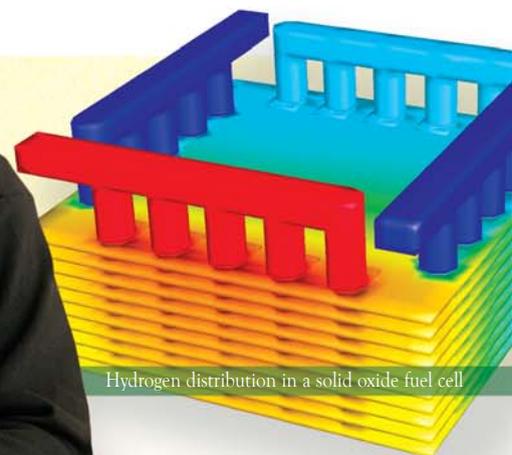
Fluent is proud to continue its long history of meeting customer needs across a breadth of industries and a myriad of applications. Clients receive technical support from some of the most knowledgeable flow modeling specialists in the business and can rely upon a wealth of core competencies as their needs dictate. Call your local office. See what FLUENT can do for your engineering process.



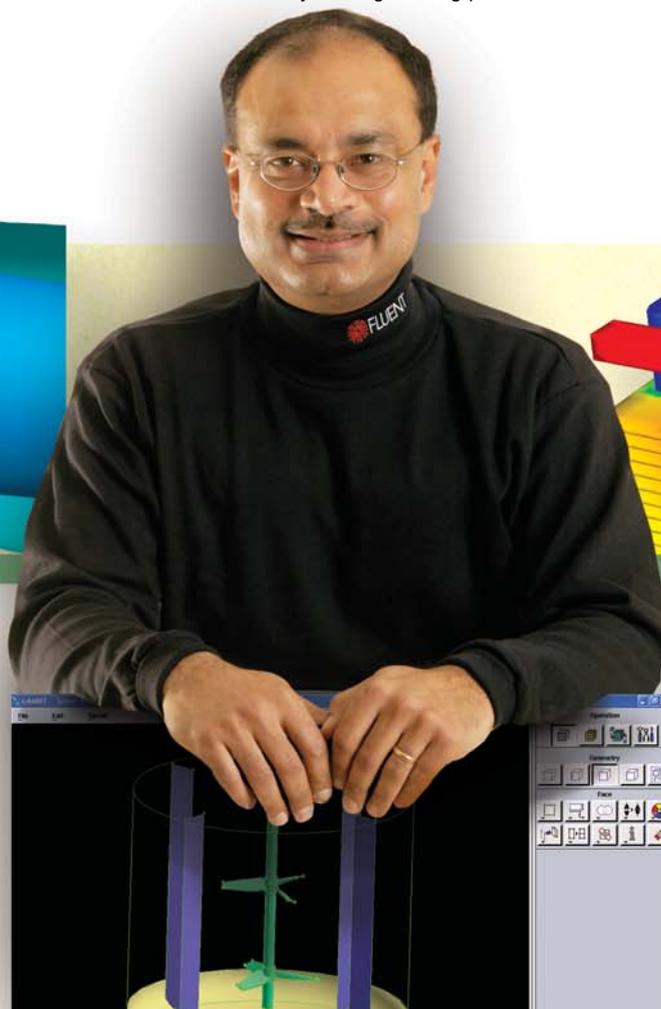
Data center cooling  
Courtesy of Facilities Engineering Associates



Pressure contours on a stator and rotor in a down-hole turbine  
Courtesy of Schlumberger



Hydrogen distribution in a solid oxide fuel cell



*"The FLUENT code is characterized above all by its user-friendliness and by its intuitive environment. Thanks to those features and its robust, accurate, and efficient solver, FLUENT has been readily adopted into our existing design process at the engine department. We now routinely use the code for simulation of engine related flows such as intake and exhaust systems."*

**Per Jonsson,  
Scania CV AB**

Dipankar Choudhury,  
Chief Technology Officer, Fluent

0110 0010101 0010101001 001 11001011  
0110 0010101 0010101001 001 110010110

# Fluent Worldwide

## Corporate Headquarters

Fluent Inc.  
10 Cavendish Court  
Lebanon, NH 03766, USA  
Tel: 603 643 2600  
Fax: 603 643 3967  
Email: info@fluent.com

## US Regional Offices

Ann Arbor, MI 48104  
Tel: 734 213 6821

Austin, TX 78746  
Tel: 512 306 9299

Evanston, IL 60201  
Tel: 847 491 0200

Morgantown, WV 26505  
Tel: 304 598 3770

Santa Clara, CA 95051  
Tel: 408 522 8726

## European Regional Offices

Fluent Benelux  
Wavre, Belgium  
Tel: 32 1045 2861  
Email: info@fluent.be

Fluent Deutschland GmbH  
Darmstadt, Germany  
Tel: 49 6151 36440  
Email: info@fluent.de

Fluent Europe Ltd.  
Sheffield, England  
Tel: 44 114 281 8888  
Email: info@fluent.co.uk

Fluent France SA  
Montigny le Bretonneux, France  
Tel: 33 1 3060 9897  
Email: info@fluent.fr

Fluent Italia  
Milano, Italy  
Tel: 39 02 8901 3378  
Email: info@fluent.it

Fluent Sweden AB  
Göteborg, Sweden  
Tel: 46 31 771 8780  
Email: info@fluent.se

## Asian Regional Offices

Fluent Asia Pacific Co., Ltd.  
Tokyo, Japan  
Tel: 81 3 5324 7301  
Email: info@fluent.co.jp

Osaka, Japan  
Tel: 81 6 6445 5690

Fluent China Holdings Ltd.  
Shanghai, China  
currently being established

Fluent India Pvt. Ltd.  
Pune, India  
Tel: 91 20 414 2500  
Email: info@fluent.co.in

## Distributors

For a full listing of our worldwide distributor network, log on to:  
[www.fluent.com/worldwide/dist](http://www.fluent.com/worldwide/dist)



 **Fluent Headquarters**

 **Offices**

 **Development Staff**

 **Distributors**

LEBANON ANN ARBOR AUSTIN EVANSTON SANTA CLARA MORGANTOWN WAVRE DARMSTADT SHEFFIELD MONTIGNY LE BRETONNEUX  
MILAN GÖTEBORG TOKYO OSAKA SHANGHAI PUNE SYDNEY MELBOURNE SEOUL BEIJING TAIPEI HSIEN CHIHUAHUA SAO PAULO RISHON LE ZION  
ISTANBUL GREENFORD BUDAPEST HAIFA TURKU KHARKOV KARACHI THESSALONIKI WARSZAWA BUCHAREST PRAHA PRETORIA



[www.fluent.com/software/fluent](http://www.fluent.com/software/fluent)

0410-10000, © 2004 Fluent Inc. All rights reserved. FLUENT, GAMBIT, TGrid, and GITurbo are registered trademarks of Fluent Inc. All other products or brands are trademarks of their respective owners.