



# 2005 CFD User of the Year Awards

**NOW IN THEIR SOPHOMORE YEAR**, the annual CFD User of the Year Awards were established to recognize excellence in applied CFD studies as demonstrated by conference presentations or papers published by users of Fluent software. Due to the many applications and benefits to be gained from the escalating use of CFD software and technology, Fluent Inc. once again honored five award categories. The 2005 award winners were chosen by a panel of Fluent executives and external CFD industry experts out of a pool of 65 candidate submissions from all over the world. According to the judges, all of the winners

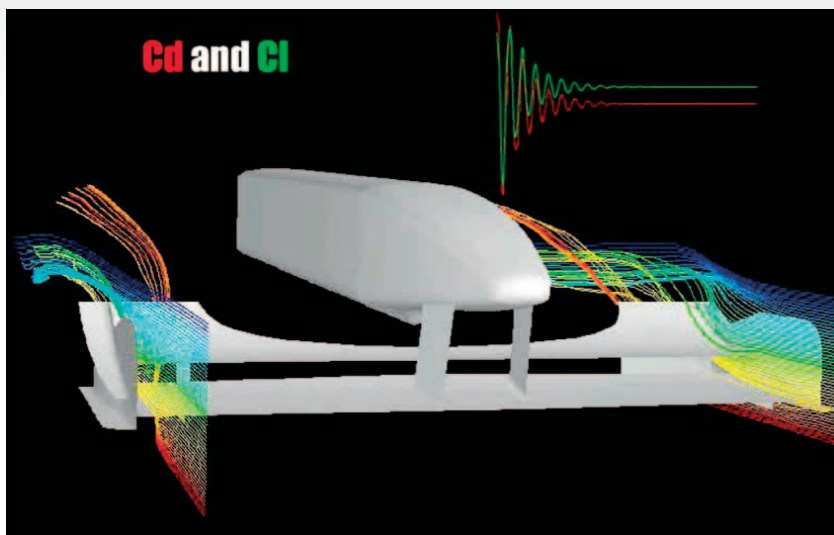
“demonstrated pragmatic solution strategies to real-world engineering situations, especially with the way they used CFD as a tool to answer complex problems.” In addition to recognition from Fluent this past March, each recipient was presented with a prize of \$1000 for their efforts.

ANSYS CFD software users are invited to submit entries for the 2006 CFD User of the Year Awards by visiting [www.fluent.com/events/cfd\\_user\\_awards\\_06/cfd\\_user\\_form\\_06.htm](http://www.fluent.com/events/cfd_user_awards_06/cfd_user_form_06.htm). Entries will be accepted through December 15, 2006.

## Most Innovative Use of CFD Technology

“Modeling FSI Problems in FLUENT using UDFs”

Riccardo Baudille, University of Rome, Italy



Deforming front wing panel on a Formula 1 race car, computed using Riccardo Baudille's FSI solver

Fluid-structure interaction (FSI) analysis is the much sought-after fusion of fluid and structural dynamics simulation (i.e., the ability to model both phenomena simultaneously), and is useful when heavy fluid actions result in deforming structural boundaries. Riccardo Baudille's research caught the attention of the judges both because of its depth and because he was able to create an FSI software toolkit that was subsequently validated against

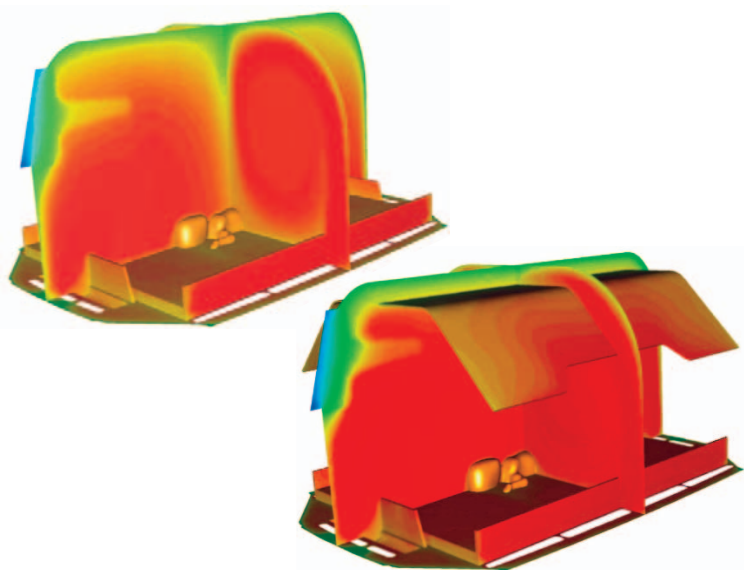
practical industrial test cases. The toolkit consists of a general finite-element method (FEM) solver that is hooked to FLUENT via user-defined functions (UDFs), a dedicated graphical user interface (GUI), and linkages to data import and export from other structural analysis codes. In addition, his tool streamlines solvers and algorithms so that the numerical overhead for his FSI calculations is as low as possible. This approach allows FLUENT to be coupled with any commercial or academic structural analysis code, and helps a CFD user to simulate the presence of structures in a very familiar way.

To test his toolkit, Baudille applied his models to the deformation of a thin front wing from a Formula 1 race car and the simulation of aero-flutter on a solid aircraft wing. In each case, the wing displacement and aerodynamic coefficients were the quantities of interest. The judges found that the FSI postprocessing and animations were very illuminating and easily understood by the viewer, which helped clinch the award. ■

## Most Impact on Society

### “Analysis of Combined Heat and Fluid Flow Processes in an Infant Incubator”

Maciej Ginalski, Silesian University of Technology, Gliwice, Poland (now at Fluent Europe Ltd., Sheffield, UK)



Temperatures on two cross planes in an incubator without (left) and with (right) an overhead plastic screen. The screen was found to reduce radiative losses from the child

Premature babies generally enter the world with little protection from the harsh environment. Thermal comfort therefore plays a crucial role in their survival and health condition. To provide the optimal environment for these infants, incubators are widely used. However, the design of modern incubators is still a developing science, with factors like airflow patterns, local oxygen concentration, and heat transfer inside the chamber being critical to their performance.

As part of his research, Mr. Ginalski developed a CFD model of conjugate fluid flow and heat transfer in an incubator to support infant health care and improve medical equipment design. Accurate geometric models representing the human body in a variety of postures were created, and these models, in combination with the associated physiological boundary conditions caused the judges to consider Ginalski’s research both pathfinding and comprehensive. His results delineated existing incubator performance and suggested enhancements to their design. His ability to incorporate a wide range of software products, including CATIA, GAMBIT, TGrid, FLUENT with UDFs, and 3D STUDIO MAX to create his CFD simulations and to visualize his flow modeling results in a high-impact way was also considered impressive. In particular, the panel noted Ginalski’s outstanding use of animations to communicate his findings to wider medical audiences. ■

## Best Industrial CFD Study

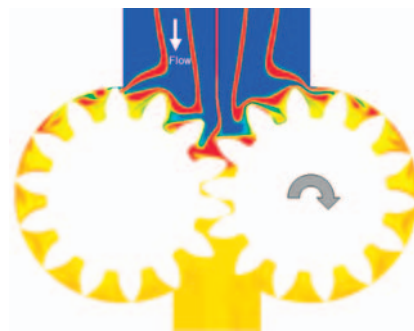
### “CFD Investigation of Gear Pump Mixing”

Wayne Strasser, Eastman Chemical Company, Kingsport, Tennessee, USA

Wayne Strasser’s industrial application of CFD at Eastman Chemical detailed the systematic modeling of fluids mixing in a moving gear pump – typical of those found in plastics manufacturing – by the use of moving and deforming meshes in FLUENT. He examined the unsteady laminar multi-phase flow of several viscous materials through an intermeshing industrial-scale metering gear pump. The process fluid being studied had a viscosity between 10 and 100 times larger than that of the additive fluid, which was fed separately to the pump intake in relatively low mass concentrations. He considered various scaled pump dimensions and throughputs to obtain coefficient of variation (COV) values for mixing that were below 5%, which is the

value typically sought in static mixer applications. Additionally, he included viscous heating and thermal/shear-thinning (VHTST) physics in his simulations, which showed an even further reduction in the COV.

In particular, Strasser was interested in preventing the buildup of high-concentration zones in the teeth cavity floors. Subsequent measurements revealed COV values much higher than those predicted by the CFD simulations. He was able to attribute this discrepancy to experimental difficulties with trying to maintain separate additive inlet streams. Despite these results, the judges commended his paper because of the systematic way Strasser evaluated a type of pump that was previously not well understood. ■

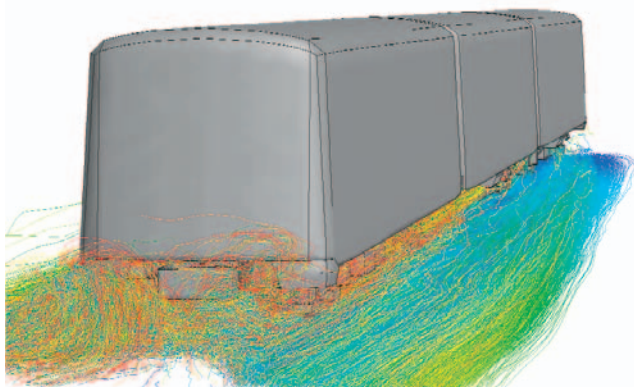


Contours of normalized mass concentration for five injection streams into a fluid in a gear pump

## Most Impact of CFD on a Business Process

### “Numerical Simulation of Snow Entrainment with Application to Train Undercarriage Design”

Markus Trenker and Wolfgang Payer, Arsenal Research, Vienna, Austria



Snow particle tracks, colored by local air velocity, collected by compressor intakes around the undercarriage of a train

In their submission, Trenker and Payer used customized CFD models to implement particle transportation and accretion on train undercarriages in cold weather. Both train manufacturers and operators view this information as pivotal, since the blocking of air inlets and vents in the undercarriages can have a deleterious effect on both ventilation systems and engine cooling for trains operating in sub-freezing temperatures.

During the course of their work, Trenker and Payer developed user-defined functions (UDFs) for discrete particle movement that they hooked to FLUENT to accurately model snow buildup. They used approximately seven million computational cells to simulate a complete train to study the buildup phenomenon, and thus were able to capture the external aerodynamics and air vent flow features. The localized flow velocities near the undercarriage surfaces were necessary for determining snow accretion zones, especially those close to vents and ducts. This work has led to the retrofitting of existing trains and the addition of new fixtures on undercarriages, thus preventing extensive (and expensive) redesign of the trains being assessed – with all of the potential downtime and even train breakdowns – if these changes had not been effected. ■

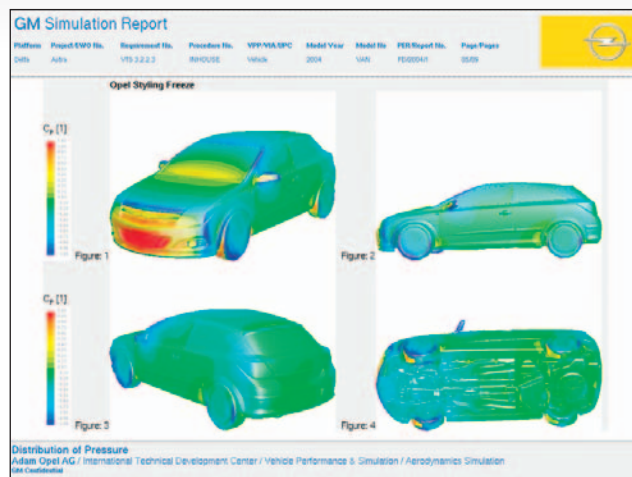
## Best Use of CFD as a Design Tool

### “Customizing FLUENT to Speed Up Aerodynamic Vehicle Development”

Silvestre Artiaga-Hahn, Adam Opel GmbH, Rüsselsheim, Germany

Silvestre Artiaga-Hahn’s work illustrated how Opel was able to optimize its development process for the external aerodynamic design of production sedans. Car developers at Opel used the internally produced Opel Virtual Aero Lab (OVAL) tool to produce custom aerodynamic reports that answer crucial “What if...?” questions, which invariably occur during the development process.

OVAL incorporates customized GUIs with an encapsulation of Opel’s CFD design process in a set of automated steps. For every one of Opel’s new simulations, a survey found that up to 90% of the CFD tasks were repetitive manual steps. It was found that much of this work, such as the manual selection of zones needed during the meshing process, could be done automatically. OVAL thus helped reduce design time for a complete external aerodynamic calculation from 20 hours to 12. This time saving is now enabling CFD simulation to better compete with traditional wind tunnel tests and compute a variety of design variants within the timeline of a styling period. In the bigger picture, OVAL has provided Opel with an improvement in its overall development timeframe for a new car such that it has fallen from 30 to 20 months, with the biggest savings coming from the automation of the mesh generation and postprocessing steps of the CFD design process. ■



A sample of flow visualization that is part of the OVAL Simulation report