ANSYS®

ADVANTAGE

EXCELLENCE IN ENGINEERING SIMULATION

VOLUME I   ISSUE 1   2007

USING MULTIPLE ANALYSIS TOOLS
NEXT-GENERATION SUBMERSIBLES
PAGE 3

**GAS TURBINE
BLADE COOLING**
PAGE 6

**FLEXIBLE RACE
CAR WINGS**
PAGE 9

**DEFORMING
BLOOD VESSELS**
PAGE 12

Welcome to **ANSYS Advantage!**

With the strategic acquisition of Fluent Inc. and blending of their leading-edge CFD technologies with its existing core software offerings, ANSYS, Inc. has further strengthened its position in having one of the broadest, most comprehensive, independent engineering simulation software offerings in the industry. The combined user base is vast, comprising one of the world's largest simulation communities with commercial seats at more than 10,000 companies, including 94 of the top FORTUNE 100 industrial companies.

One of the best ways to serve this growing simulation community is with a single publication providing a forum for the exchange of ideas, a conduit for technology transfer between disciplines and a common framework for integrating so many diverse areas of interest. With this in mind, the former *Fluent News* and *ANSYS Solutions* publications have been merged into the new quarterly *ANSYS Advantage* magazine covering the entire range of ANSYS technologies and applications.

One of the greatest benefits of a single magazine is the opportunity for readers to become familiar with software and applications beyond their usual fields of interest. Mechanical engineers accustomed to using ANSYS primarily for structural analysis may see how the use of CFD could be used in their development efforts, for example. Likewise, CFD analysts can better understand the tools used to gain insight into the mechanical behavior of products.

Our editorial team is proud to present this premier issue. The feature articles highlight applications in which multiple simulation technologies are used. For example, our cover story discusses how Hawkes Ocean Technologies used ANSYS CFX software to minimize drag in the design of an innovative two-man oceanographic craft and ANSYS Mechanical tools to ensure that composite parts withstand underwater pressure without being overdesigned with excess material. At the center of the magazine, a 16-page supplement shines a spotlight on applications in the sports and leisure industry that range from the design of alpine skis to fitness equipment.

We invite you to consider *ANSYS Advantage* your magazine, not only providing information about software products and technology applications but also giving you a way to share your work with colleagues in the simulation community. We welcome your feedback and ideas for articles you might want to contribute. Most importantly, we hope you find the publication to be a valuable asset in implementing simulation-based product development in your own workplace. ■

Liz Marshall and John Krouse, Editors

Contents

FEATURES

(Continued on next page)

Spotlight on Engineering Simulation in the Sports and Leisure Industry

Taking **Next-Generation Submersibles** to New Depths

ANSYS simulation tools help minimize drag and reduce weight by half in two-man oceanographic craft.

By Adam Wright
Hawkes Ocean Technologies
California, U.S.A.

The world beneath the ocean surface is teeming with most of earth's animal and plant species. While three-quarters of our planet lies under water, less than 5 percent has been explored, mainly because of shortcomings in today's research equipment. Scuba limits divers to the topmost slice of the oceans. Conventional submersibles, on the other hand, are designed to drop like bricks into the ocean depths using variable buoyancy to control dive depth with bulky air tanks, compressors, pumps and piping. As a result, they have limited maneuverability and need a dedicated mother ship to transport and maintain them. Furthermore, the loud operational noise and bright lights associated with these crafts scare away many sea organisms.

Hawkes Ocean Technologies has come up with a solution to move beyond these constraints: a new class of small, highly maneuverable craft that can be piloted through the water to a desired depth using controls, wings and thrusters for undersea flight similar to that of a jet aircraft.
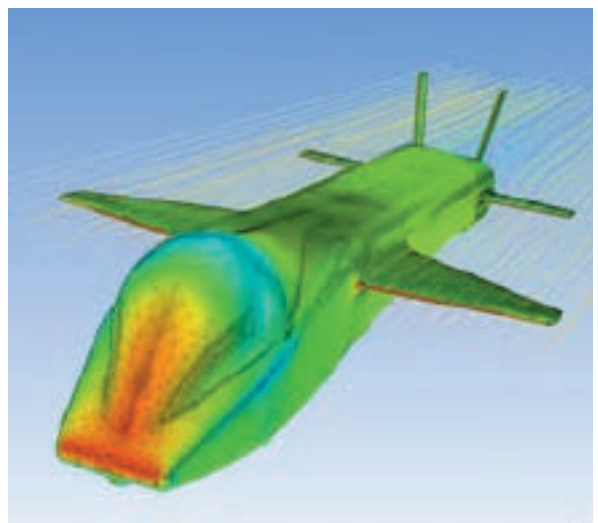
ANSYS CFX computational fluid dynamics software helped develop the overall streamlined shape of the external fairing to minimize underwater drag.
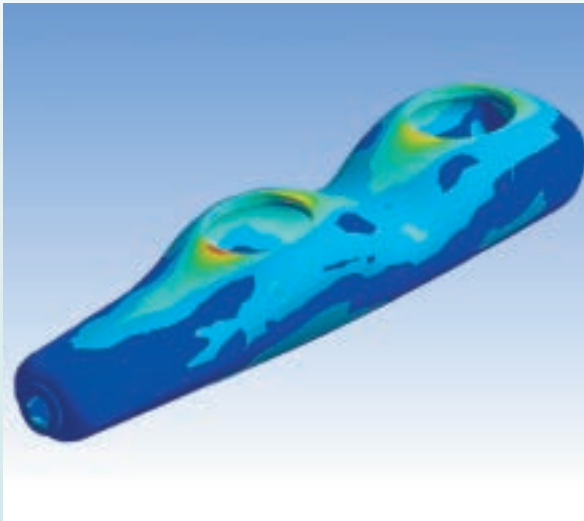
ANSYS Mechanical software was used extensively for stress analysis in ensuring that the pressurized pilot compartment hull could safely withstand 700 psi at quarter-mile depths without overdesigning components with excess material.

The Wet Flight is a high performance one-person sub designed for underwater filming.

In this way, the company's winged-submersible concept combines the vision and low-intrusiveness of scuba diving with the depth capability of a conventional submersible.

An internationally renowned ocean engineer and explorer, company founder Graham Hawkes holds the world record for deepest solo dive of 3,000 feet and has been responsible for the design of hundreds of remotely operated underwater vehicles and manned underwater craft built for research and industry worldwide. The Deep Rover submersible, for example, is featured in James Cameron's 3-D IMAX film "Aliens of the Deep," and the Mantis craft appeared in the James Bond film "For Your Eyes Only."

Based near San Francisco Bay, California, U.S.A., Hawkes Ocean Technologies is an award-winning design and engineering firm with a small staff of dedicated professionals who use ANSYS software to help them develop their innovative craft. Hawkes' winged submersibles, which are based on the concept of underwater flight, are rated for a depth of 3,000 feet; the next-generation submersibles already have been tested down to 20,000 feet. The model currently being designed and built is a next-generation two-man craft with lightweight carbon-reinforced composite material replacing the aluminium parts of the previous model. A pressurized pilot compartment hull and electronic equipment housings are made of a filament-wound composite, while the streamlined exterior skin of the craft is made of layered fabric composite. Transparent acrylic domes provide 360-degree visibility and minimize distortion due to water boundary refraction.

Challenges of Withstanding Pressure

One of the most difficult aspects of designing the new craft involved the determination of stresses in the complex geometries of the composite parts that must withstand pressures of nearly 700 psi. In particular, the compartment hull protecting pilots from this crushing pressure is a cocoon-like contoured structure designed to maximize space in order to maintain comfort: a significant design factor because an occupant tends to become cramped and possibly claustrophobic after an hour or two beneath the great mass of water above. Another complicating factor in determining component stress distribution was the anisotropic nature of the composite material properties, which have different strengths in each direction depending on the orientation of the carbon fiber.

In addition to ensuring adequate strength of the craft, designers had to optimize tradeoffs between power and weight. One problem to be addressed was that of minimizing the underwater drag of the external fairing to achieve maximum speed with minimal power consumption. The right balance allows the craft to sustain the speed needed by the airfoils to overcome positive buoyancy while extending the range. Since the winged craft must keep moving at about two knots to remain submerged, this was a critical consideration.

The Solution

To address these design issues, Hawkes engineers turned to simulation tools within the ANSYS Workbench environment. To minimize drag, ANSYS CFX computational fluid dynamics software was used to develop the overall streamlined shape of the external fairing. The analysis defined the flow around the fairing and enabled researchers to readily pinpoint any areas of excessive turbulence. The results helped them configure the shape for minimum hydrodynamic resistance and maximum lift and effectiveness of the airfoil surfaces for allowing the craft to dive and maneuver underwater.
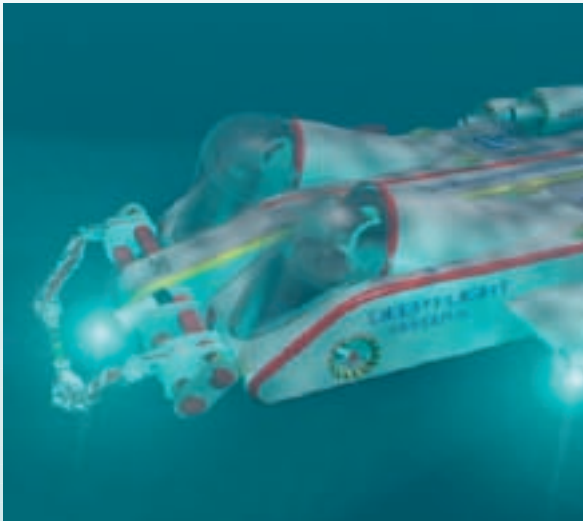
The Deep Flight II can house one or two persons in a prone position and can travel for up to eight hours.

The Wet Flight submersible rises to the surface.

When diving to 1,500 feet and deeper depths, there is no room for error, so Hawkes used ANSYS Mechanical software for stress analysis to ensure that composite parts could withstand underwater pressure without being overdesigned with excess material. The program readily accounted for the anisotropic material properties of the composite parts and clearly showed directional stresses graphically as well as numerically with precise von Misses values. The capability helped engineers determine the proper carbon fiber orientation and wall thickness needed to strengthen high-stress areas of composite parts, particularly the pressurized pilot hull.

The stress levels of assemblies of individual parts made of different materials also were analyzed. For example, one assembly included the metal locking ring that clamps the fittings and seal of the acrylic dome to the composite hull, along with the dome and hull. In generating these assembly models, the ANSYS surface-to-surface contact element feature automatically detected the contact points, allowed for different material properties and adjusted mesh densities instead of requiring users to perform these tasks manually. Moreover, convenient element-sizing functions enabled engineers to readily increase mesh density in localized regions in which they wanted to study stresses in greater detail.

Easy access to computer-aided design (CAD) software and simulation applications through the integrated ANSYS Workbench platform allowed Hawkes engineers to become productive on the first day. Simulation models were created based on part geometry from the Autodesk® Inventor™ design system. Direct associativity with the CAD system enabled engineers to readily change the design based on

an analysis and quickly perform another simulation on the new part geometry without having to re-apply loads, supports and boundary conditions. For some cases, more than 40 design iterations were tested. The approach saved considerable time and effort, allowed numerous alternative configurations to be studied, guided engineers toward the uniquely contoured compartment hull shape, and, perhaps most importantly, minimized mistakes. In this way, the researchers were able to quickly arrive at a not-intuitively-obvious optimal design for a craft that could withstand prescribed pressure limits with minimal weight and fit within the tight space constraints of the two-man submersible.

Significant Weight Reduction

By using ANSYS software in the design of components to be made with composites instead of aluminium, engineers were able to reduce the overall weight of the craft by 50 percent. This significant weight reduction is expected to increase maximum underwater speed and save battery life to increase the time the craft can spend underwater. Because the lightweight submersible does not need a dedicated mother ship, operational costs are reduced by 70 percent and the craft can operate freely worldwide off of a variety of launch platforms. This greatly expands the underwater exploration possibilities of the craft. Furthermore, these next-generation submersibles hold the potential of unlocking new biotechnology from the ocean depths that may help cure disease, discovering new aquatic species, finding new mineral and food reserves, studying weather, and providing a means to monitor and prevent further pollution at sea. ■

Fluid Structure Interaction Makes for **Cool Gas Turbine Blades**

An integrated simulation process improves performance without sacrificing longevity.

By Michel Arnal, Christian Precht and Thomas Sprunk, Wood Group Heavy Industrial Turbines AG, Switzerland
Tobias Danninger and John Stokes, ANSYS, Inc.

The blade geometry

The internal features of the blade geometry include the plenum (blue) and the cooling channels (gold).

In gas turbines, hot gas from the combustion system flows past the rotating turbine blades, expanding in the process. In order to reach desired levels of efficiency and power output, advanced gas turbines operate at very high temperatures. As a result, the components subjected to these high temperatures often require cooling. One method of cooling the turbine blades involves extracting air from a compressor and forcing it through a plenum and into channels inside the blade. While effective cooling of the blades can increase their lifespan, it can also reduce the thermal efficiency of the engine. It is therefore important to develop designs that extend component life while having a minimal effect on engine thermal efficiency. Numerical simulations that accurately capture the interaction between the fluid and thermal effects can play an important role in the design process.

Wood Group Heavy Industrial Turbines provides a comprehensive range of support solutions, including re-engineered replacement parts and maintenance, repair and overhaul services for industrial gas turbines and related high-speed rotating equipment used in the global power generation and oil and gas markets. One example of the work done by Wood Group is a recent project involving the re-engineering of the blade from the first stage of a gas turbine. The goal of the project was to optimize the blade design and improve its longevity. The numerical simulation process coupled ANSYS CFX software for the fluid flow, ANSYS Mechanical software for the

structural response of the blade, and the 1-D thermal and fluid flow simulation package Flowmaster2. This set of simulation tools provided an efficient virtual prototype that was used to assess the performance of the turbine blade under actual operating conditions.

CFD Model

The original 3-D CAD geometry, which is intended for manufacturing, was extended for the purpose of the simulation using ANSYS DesignModeler. The extensions served to better represent the true operating conditions of the rotor. For example, gaps not present under normal operating conditions were closed. This extended CAD model then served as the basis for the CFD mesh.

The two fluid domains (the hot gas flow around the blade and the coolant airflow in the plenum) and one solid domain (the blade itself) were meshed independently using ANSYS ICEM CFD meshing software. Generalized grid interfaces (GGIs) were used to connect the non-matching mesh topologies of the individual domains. The cooling channels were modeled using Flowmaster2, and the result of this 1-D simulation was connected to ANSYS CFX using the standard CFX Expression Language (CEL), which requires no user programming. Taking advantage of CEL callback functions, the coolant air flow in the plenum, the hot gas around the blade and the heat conduction through the solid blade can be solved for in a single ANSYS CFX simulation. At the same time, the CFD simulation can use the unique ANSYS CFX model for laminar to turbulent transition, a key feature that properly captures heat transfer rates from the hot gas to the blade surface as the boundary layer develops. The temperature field in the solid blade as computed by ANSYS CFX software was then directly written out in a format appropriate for the subsequent ANSYS Mechanical calculation.

FE Model

For the simulation using ANSYS Mechanical software, the 3-D temperature field in the solid blade, calculated in

Overview of the interaction between the simulation tools used to provide a virtual prototype of the gas turbine blade.

A schematic of the two fluid (blue and red) and one solid (gray) domain used to perform the analysis. Boundary conditions are defined at locations indicated with arrows. The green lines indicate generalized grid interface connections. The inflow to the hot gas domain from the cooling channels is described by CFX Expression Language callbacks to 1) the flow out of the plenum domain and 2) the heat transfer from the solid domain to the cooling channels.

ANSYS ICEM CFD mesh of the hot gas fluid domain at the tip of the gas turbine blade

the ANSYS CFX conjugate heat transfer analysis, was used as input for the thermal load. This, along with the rotational load on the blade at operating conditions, determined the stress distribution. Together, the resulting thermal and mechanical stress distributions in the blade were used to determine component life. Applying these loads, life-limiting elements of the blade design could be determined and new design alternatives evaluated.

The ability to combine the entire fluid and thermal analysis through the use of standard functionality, especially the powerful CFX Expression Language and its callback functions, are key to making simulations such as this feasible. By combining both CFD and structural analysis with a 1-D thermal simulation, this virtual prototype has provided a more complete understanding of the performance of each blade design in a given set of operating conditions. This allows modifications to be made early in the design process, and therefore is essential in the efforts to help improve efficiency and increase longevity. ■

Suggested Reading

Arnal, Michel; Precht, Christian; Sprunk, Thomas; Danninger, Tobias; and Stokes, John: Analysis of a Virtual Prototype First-Stage Rotor Blade Using Integrated Computer-Based Design Tools. Proceedings of ESDA2006 8th Biennial ASME Conference on Engineering Systems Design and Analysis, Torino, Italy, July 2006.

The finite element mesh

a)

b)

c)

Stress distribution in the directionally solidified blade due to a) temperature variations (but not including rotational effects), b) centripetal forces (assuming a constant temperature) and c) the combination of temperature variations and centripetal forces
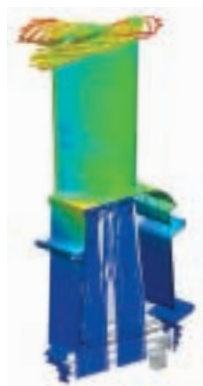
ANSYS CFX temperature simulation of the blade surface. Streamlines show flow from the inlet into the plenum and from the cooling channel outlets into the hot gas. Where the internal cooling channels are close to the blade surface on the suction side of the blade near the trailing edge, areas of lower temperature are shown in blue.
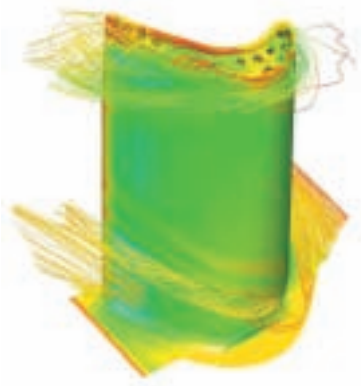
CFD simulation of heat flux distribution. The heat transfer is from the hot gas to the blade surface in most areas, but in the tip region the heat transfer is positive corresponding to where the cooling air from the cooling holes comes into contact with the blade surface.
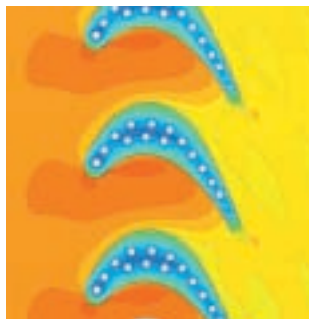
Temperature contours in the flow field and through the blade at a radial location near the blade platform (left) and outer casing (right)

Race Cars **Flex Their Muscle**

An Indy car rear wing is designed for aeroelastic response using
multidisciplinary optimization.

*By David Massegur, Giuseppe Quaranta and
Luca Cavagna, Department of Aerospace Engineering
Politecnico di Milano, Italy*
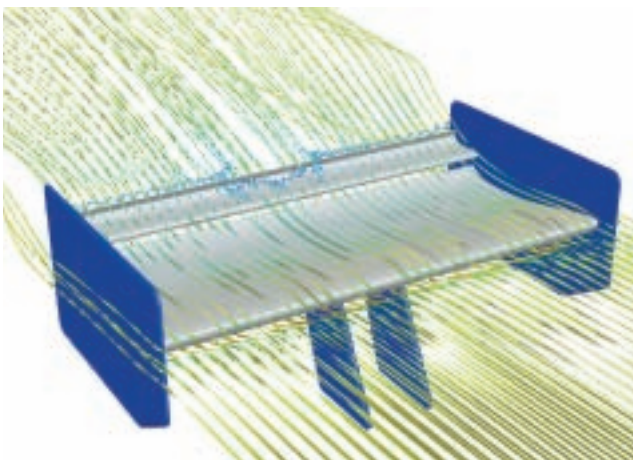
Pathlines illustrate the presence of the flow recirculation behind the gurney flap.

Aerodynamics play a crucial role in the performance of race cars, such as Indy and Formula 1, and for years, teams have spent a great deal of time and money on wind-tunnel testing. Nowadays, thanks to increases in computational power, CFD has become a valuable tool for fine-tuning both the external and internal shape of these cars. The goal is to maximize downforce, in order to increase cornering speeds, and to reduce drag to be faster on the straights. Thus, the highest aerodynamic efficiency is sought that represents the optimal trade-off between high downforce at low speeds (for cornering) and low drag at high speeds (for driving on the straights) [1].

To improve car performance at the different operating conditions, the flexibility of aerodynamic devices (aeroelastic effects) can be exploited. In fact, the changes in the shape of such devices due to deformation may cause a modification of the flow field around the car. Despite being severely restricted by technical regulations, this currently is the only way to optimize the car for different regions

1. Main wing
2. Flap wing
3. Gurney flap
4. End-plates
5. Supports
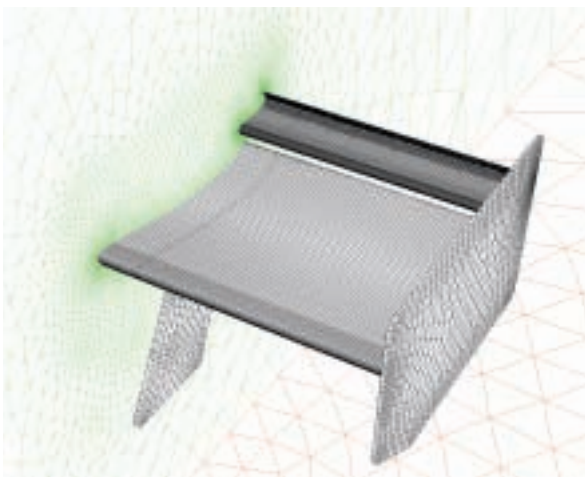
The main elements that form the rear wing (top) and their respective discretization (bottom)
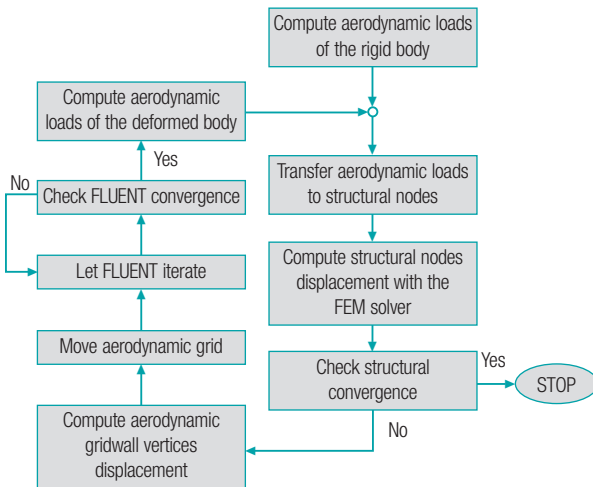
The iterative method used to solve the static aeroelastic problem

of the track because any servo-aided device aimed at moving an element of the car is strictly forbidden. Since aerodynamic loads increase quadratically with speed, aeroelastic phenomena can be exploited more easily at high speed, where the pressure loads cause larger deformations.

A multidisciplinary computational model has been developed at the Politecnico di Milano to evaluate, by means of numerical optimization, possible geometric and structural configurations of race car rear wings in order to tailor aeroelastic phenomena to maximize car performance.

The model has been applied to the DALLARA Indy car rear wing. The focus of the study has been on the influence of the structural deformation of the carbon-fiber flap on the aerodynamic loads of the whole rear wing assembly. Future applications will investigate the effects of other deformable parts, such as the pillar junctions.

The multidisciplinary optimization requires the development of a static aeroelastic algorithm to compute the correct loads. In this case, CFD is the perfect tool for predicting the complex flow phenomena around the wings, such as the flap influence on the main wing and the twin-vortex recirculation around the gurney flap [1]. FLUENT has been used for the CFD calculation, because it is able to run steady-state, Reynolds-averaged Navier-Stokes (RANS) models with relatively modest computational resources.

The aeroelastic problem is solved by applying an iterative procedure based on a sequence of load calculations by means of CFD for a given shape followed by an FEM calculation to compute the deformation based on the CFD loads. The software NASTRAN® is used for the structural calculation, and the sequence is run until convergence is reached. The adoption of two different solvers for the fluid and structure provides the freedom to choose the optimal discretization method for each, but it requires the implementation of an interface scheme to transfer the necessary information between the two calculations [2]. To transfer the structural displacements from the FEM model to the surfaces of the CFD grid and the aerodynamic loads to the structural nodes, an algorithm based on the weighed moving least squares (WMLS) method has been developed [3].

The aeroelastic solution is managed in FLUENT software through a number of user-defined functions (UDFs) that execute the different tasks required by the iterative solver. Of all the tasks, the most expensive one, requiring almost half of the solution time, is the deformation of the CFD grid, used to adapt to the new wing shape after each iteration. The spring-based method in FLUENT software is used for remeshing. It is based on the analogy between the computational grid and a network of linear elastic springs, with stiffness inversely proportional to the distance between the respective adjacent nodes. However, the deformation required by each time step typically is greater than the dimensions of the cells close to the walls, giving rise to the
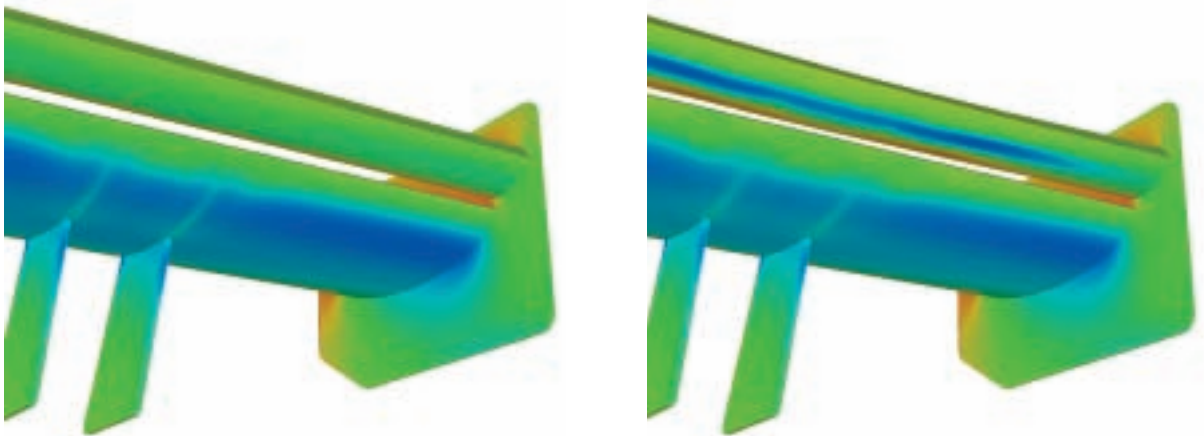
Pressure coefficient distribution comparison between the rigid (left) and deformed (right) wings

appearance of negative cell sizes. To improve the robustness of the method, an algorithm has been implemented to adaptively subdivide the required deformation into substeps with achievable deformations.

The net result of the approach has been a multi-objective optimization of both the wing geometry and structural characteristics to increase downforce at low speeds and decrease drag at high speeds. The response surface method (RSM), driven by the Design of Experiments (DOE) technique [4], has been used to run the cases. This approach limits the total number of required analysis yet allows up to 20 design variables, such as the composite material properties, number and orientation of the plies in different zones of the wing, the wing angle of attack, the wing sweep and spanwise twist. The optimization [5] is subject to design constraints relative to the fulfillment of flexibility tests required by regulations and material strength.

The results from the calculations show that variations of 25 percent on both downforce and drag can be obtained, depending on the aerodynamic configuration. Keeping the same level of downforce delivered while cornering, wing drag can be reduced by 3 percent. As a result, the car top speed can be improved by 1 km/hr, which represents a gain of half a tenth of a second per lap in tracks such as

Barcelona. This initial application has shown the high gains that can be potentially achieved by multidisciplinary optimization for race cars. Significant improvements are expected by applying the proposed method to other aerodynamic surfaces, such as the front wing and the diffusers. ∎

References

[1] Aerodynamics of Race Cars, Annual Review of Fluid Mechanics, University of California, U.S.A. (2006).

[2] L. Cavagna, G. Quaranta, P. Mantegazza, E. Merlo, D. Marchetti, M. Martegani: Flexible Flyers in the Transonic Regime, Fluent News, Spring 2006.

[3] G. Quaranta, P. Masarati, P. Mantegazza: A Conservative Mesh-Free Approach for Fluid-Structure Interface Problems, International Conference for Coupled Problems in Science and Engineering, Greece (2005).

[4] D. C. Montgomery: Design and Analysis of Experiments, Wiley International Edition, New York, U.S.A. (2001).

[5] I. Das, J. Dennis: Normal-Boundary Intersection: An Alternate Method for Generating {Pareto} Optimal Points in Multicriteria Optimization Problems},SIAM Journal on Optimization, 8 (1998).
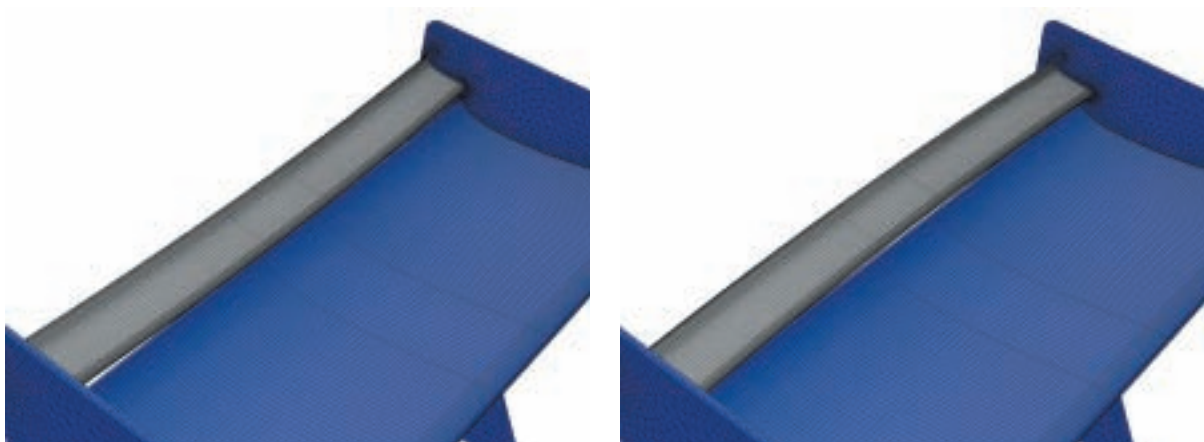
Two examples of a swept wing, introduced as a variable of the multi-objective optimization, are a back-swept (left) and a front-swept (right) wing.

Modern Medicine Takes Simulation to Heart

A fluid structure interaction simulation is performed to capture patient-specific modeling of hypertensive hemodynamics.

By Kendall S. Hunter, Department of Pediatrics, Section of Cardiology, University of Colorado Health Sciences Center, Colorado, U.S.A.

Could simulation technology more commonly associated with rocket science and race cars someday provide insight into the inner workings of the vascular system that would help doctors provide improved diagnosis treatment in clinical situations? Researchers at the University of Colorado Health Sciences Center (UCHSC) have taken the first steps toward that end, and the ANSYS fluid structure interaction (FSI) solution is proving to be a key enabling technology.

The pulmonary arteries are the blood vessels that carry oxygen-poor blood from the right ventricle of the heart to the small arteries in the lungs. For a healthy individual, the normal average pressure in the pulmonary artery is about 14 mm Hg. For individuals with pulmonary arterial hypertension (PAH), the average pressure is usually greater than 25 mm Hg. This increases the load on the right side of the heart and can lead to eventual heart failure and death.

Diagnosis and evaluation of PAH typically is accomplished with a combination of cardiac catheterization (in which a plastic tube is passed through the iliac vein in the leg and weaved up the body, through the right side of the heart, and out into the main pulmonary artery) and imaging techniques such as angiography

Mesh generated using ANSYS ICEM CFD Hexa; the CFD domain is bounded by the blue cells and the shell mesh, used for the structural calculation, is shown in lavender.

and magnetic resonance imaging (MRI). While these methods are effective in the diagnosis of vascular pathologies, they cannot currently provide enough detail or be performed with sufficient frequency to elucidate the causes of disease progression and are hard pressed to predict the outcome of clinical interventions. To date, clinicians have mainly characterized PAH by evaluating pulmonary vascular resistance (PVR), defined as the mean pressure drop divided by the mean flow rate. In considering only mean conditions, the effects of vascular stiffness are ignored; in patients with PAH, however, these effects can amount to 40 percent of the total right heart afterload. Over time, the vasculature can thicken in response to the increased pressure. Such proximal thickening and stiffening is believed to change distal flow and further increase pressures; thus, it may be part of a feedback loop by which PAH worsens.

At UCHSC, researchers are investigating the impact of proximal artery stiffness by using ANSYS software to simulate the transient fluid structure interaction of the blood flow and vascular walls of the pulmonary artery. By using numerical simulation, researchers can gain a better fundamental understanding of the physics involved in PAH and insight into the effects of vascular stiffness on proximal, and, perhaps more importantly, distal hemodynamics. Eventually, the regular clinical use of patient-specific

Contours of pressure on the vessel walls at peak systole

Contours of vessel wall displacement at peak systole

simulation, in which the vascular geometry is extracted from medical imaging, could provide better insight into the progression of PAH and improve predictions of the outcome of surgical intervention.

For the ANSYS FSI simulations reported here, geometry acquisition begins with bi-plane angiography of the proximal pulmonary tree performed during cardiac catheterization of an 18 month-old male patient. This provides data describing the vessel centerline and diameter. A CAD system is used to turn this skeletal data into a smooth representation of the vessel geometry. The geometry is imported into ANSYS ICEM CFD software and the Hexa meshing module is used to construct a high-quality hexahedral volume mesh. The resulting mesh uses an O-grid inflation layer from all walls so that the mesh is nearly orthogonal with excellent control over near-wall spacing. This mesh is used for the CFD component of the FSI simulation, solved using ANSYS CFX software. The quad surface elements from that same mesh are imported into ANSYS as a shell element representation of the vessel. This type of representation is a significant advan-

tage, since it allows investigations in which the vessel wall thickness is varied without the need for geometry modifications or re-meshing. A script is used to apply variable shell thickness on a node-by-node basis to the vessel mesh.

For these studies the Arruda–Boyce hyperelastic material model is used. The model parameters were suggested by biomechanical studies of the stress–strain properties of normotensive and hypertensive pulmonary arteries from a rat model and solid-only simulations of human pulmonary arteries. Residual stress is not considered here due to the difficulty of incorporating such effects in clinical models in which direct measurements within the artery cannot be obtained. The solid model was constrained on the inflow/outflow boundaries. The remaining nodes were allowed to deform in response to applied forces.

Blood is modeled as an incompressible Newtonian fluid with constant dynamic viscosity and the flow is assumed to be laminar. Using the CFX Expression Language (CEL), it was straightforward to implement a time-

varying mass flow boundary condition at the fluid inlet with a half-sinusoid profile. Exit boundary conditions were modeled using CEL and a resistive relationship in which the outlet pressure for each branch was determined by multiplying the local instantaneous flow rate by a resistance factor. [1,2]

The early results of this pilot study have confirmed the anticipated behavior of the system. Upcoming studies with improved clinical and imaging data will allow validation and refinement of the simulation methodology. Eventually, the clinical use of non-invasive, patient-specific simulation may provide better understanding of the progression of PAH and improved predictions of the potential outcomes of available treatments. ■

References

1   Vignon-Clementel, I.E.; Figueroa, C.A.; Jansen, K.E.; Taylor, C.A.: Outflow Boundary Conditions for Three-Dimensional Finite Element Modeling of Blood Flow and Pressure in Arteries. Comp. Meth. App. Mech. Engr. (CMAME) 2006; in press.

2   Olufsen, M.S.: Structured Tree Outflow Conditions for Blood Flow in Larger Systemic Arteries. Am. J. Physiol. (AJP) 276(1): H257-H268, 1999.

CAE Takes a **Front Seat**

Engineers use ANSYS software to meet complex and potentially conflicting requirements to design a chair for a wide range of body types and postures.

By Larry Larder and Jeff Wiersma
Herman Miller Inc., Michigan, U.S.A.

Herman Miller Inc. transformed the residential furniture industry as America's first proponent of modern design, beginning in the early 1930s through collaborations with iconic figures like Gilbert Rohde, George Nelson, Charles and Ray Eames, Isamu Noguchi and Alexander Girard. Later the company transformed the modern office with the world's first open-plan office systems in the 1960s and the concept of ergonomic office seating in 1976 with the introduction of the Ergon® chair, followed by the Equa® chair in 1984. In 1994, the company launched the groundbreaking Aeron® chair. Founded in 1923, the company is one of the oldest and most respected names in American design. It has been recognized as a design leader, receiving the Smithsonian's "National Design Award." Dozens of its designs are in the permanent collections at major museums worldwide, including the New York Museum of Modern Art, the Whitney Museum, the Henry Ford and the Smithsonian Institution.

As one of the leaders in high-performance office furniture, Herman Miller set its sights in 2000 on the long-neglected and potentially lucrative mid-priced segment of the market, representing half of all office chairs sold worldwide. The goal was to develop the Mirra™ chair as an entirely new reference point for mid-priced
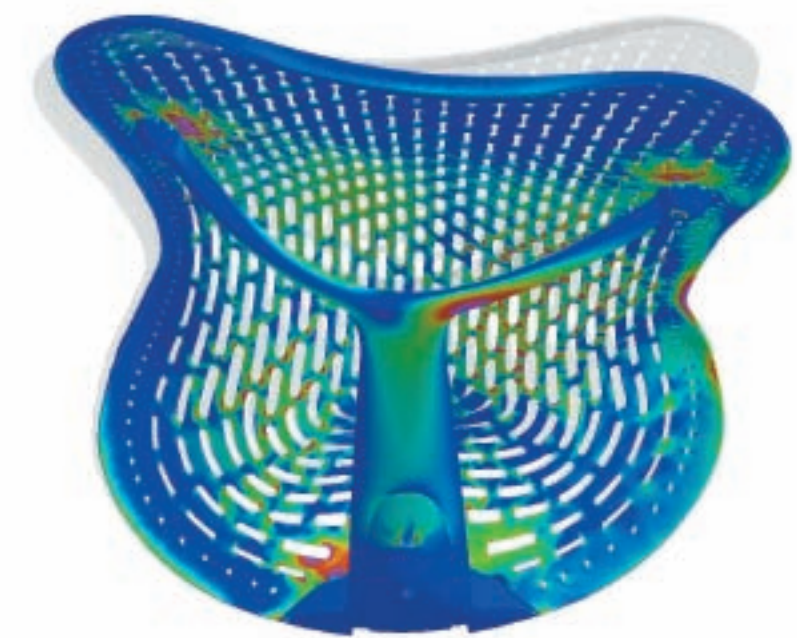
The TriFlex back that automatically adjusts to each user was developed as a single composite plastic structure using analysis to determine the coupled response of the back and its supporting spine.

office seating offering ergonomic comfort for a wide range of body types and postures and easy adjustability for fit and feel. The cost also needed to be kept as low as possible through reduced part counts and effective use of structural materials, developed completely under Design for the Environment (DFE) protocols.

Given these many complex and potentially conflicting requirements, developing the chair through cycles of trial-and-error physical testing was considered impractical because the approach is expensive, time-consuming, and limits the number of design alternatives to be evaluated. Engineers needed a way to optimize

Herman Miller's new award-winning Mirra office chair was developed through virtual prototyping using ANSYS software.

ANSYS Structural software played a key role in the development of the cantilever leaf spring and moving fulcrum tilt mechanism.

the design early in the development by investigating a wide range of possibilities at that stage. These challenges were met through the use of virtual prototyping, in which "what-if" scenarios can be readily studied in the computer and hardware testing is more of a verification of the design at the end of the cycle.

One of the key virtual prototyping technologies selected was ANSYS structural analysis software, used as the primary tool for determining stress and deflection on every part of the chair. Engineers routinely used ANSYS DesignSpace to develop major components such as the base, arms, and pedestal. For more complex analysis, ANSYS DesignSpace models were used by an analyst as the basis for detailed simulation with ANSYS Structural software.

ANSYS Structural played an important role in the development of one of the chair's key assemblies: a cantilever leaf spring and moving fulcrum tilt mechanism that provide resistive force so that a person can lean back comfortably. Torque curves were generated to represent the force required to support various body types in three seat positions: upright, fully tilted and midway. The analyst wrote a text script file to simulate a range of spring and fulcrum combinations to operate within this torque-curve design envelope. Output from ANSYS software included spring deflection and stress distributions, giving engineers insight into each design so that they could select and refine the configuration that worked best. The result was an optimal mechanism that provided the range of torque required with only a few simple adjustments. Guided by the simulation, the design met the company's objectives of comfort and adjustability. Moreover, the text script file will be used as a basis

for developing similar mechanisms in other chair models.

Another major feature of the chair is a "passively adjustable" polypropylene back. In contrast to conventional rigid-back chairs, the pliable TriFlex™ back design provides the proper deflection according to the user's posture and movements. This concept evenly distributes seating forces, thus reducing load concentrations and fatigue. Engineers used ANSYS Structural software to determine the coupled response of the back and its supporting spine based on the material characteristics of each part together with the size and geometric pattern of the perforated back. Analysis was used extensively to engineer a single composite plastic structure that delivered the required coupled deflection response, reduced the parts count for the assembly and conformed to the DFE environmental criteria.

With the aid of simulation, Herman Miller developed an optimal chair design that delivered the required functionality while maintaining the company's high quality standards of wear and reliability. Prototype testing time was minimized, with a physical mock-up used to verify the functional performance established through analysis. Simulation also enabled

engineers to consolidate parts into integral modules, thus minimizing part counts and lowering manufacturing costs significantly. Due to these and other cost efficiencies, product margins for the Mirra have met target objectives. In terms of market acceptance, the chair has consistently exceeded the company's targets for orders and shipments.

Introduced in 2003, the Mirra chair received the Gold Award in the Best of NeoCon industry competition. It was named by *FORTUNE* magazine as one of the "Best Products of the Year" and received the Chicago Athenaeum Museum of Architecture and Design's Good Design Award. The goal of the Mirra chair was to set a new reference point for the mid-price seating market in terms of ergonomics and adjustability. Simulation with ANSYS software certainly allowed Herman Miller to meet these objectives with advanced technology that could be integrated easily into its product development process. Rather than merely fix problems toward the end of the development cycle, simulation was used to guide the design. As a result, the Mirra is probably one of Herman Miller's most successful and highly engineered products. ■
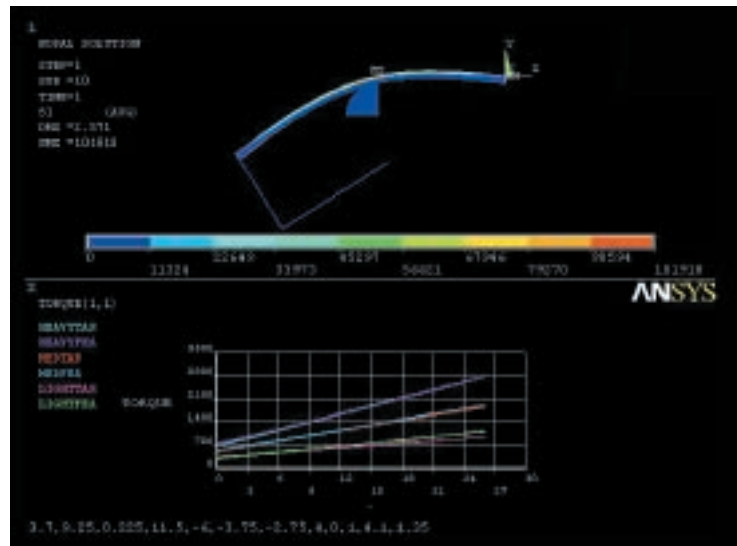
Transport of **Fragile Granules**

Pneumatic conveying systems in the pharmaceutical industry can lead to unwanted particle breakup.

By Pavol Rajniak and Rey Chern, Pharmaceutical Commercialization Technology, Merck & Company, Inc.; U.S.A., Kumar Dhanasekharan, ANSYS, Inc.; Csaba Sinka and Neil MacPhail, Merck Sharp and Dohme, U.K.

Pneumatic conveying systems are used at pharmaceutical manufacturing sites to transport granular materials. These materials — the active ingredient and various inactive ingredients — are combined to produce granules and then are transported to tablet presses, where pills are formed. Granule attrition, in which the particles suffer wear as a result of collisions and friction, can occur during the transport of materials. Even for a dilute mixture, attrition can reduce characteristic particle sizes by as much as 50 percent [1] leading to a deterioration in the granule properties and potentially compromising the quality attributes of the pharmaceutical product. Experimental and theoretical studies to understand the mechanical impact of conveying on granules are needed so that formulations, the processing parameters and the pneumatic conveying systems can be optimized to avoid problems at the large scale. To address attrition phenomena, different experimental and theoretical approaches have been followed. One experimental approach has been carried out at various bends, providing stress conditions closely related to industrial processes [2].

The Malvern Mastersizer particle size analyzer and its dry powder feeder (DPF)

Contours of the solid phase volume fraction with solid phase velocity vectors (left), and Sauter mean diameter (right) in the vicinity of the bend for an inlet jet pressure of 1.5 barg, inlet gas velocity of 30 m/s, average inlet granule diameter of 57.8 μm and granular material density of 1200 kg/m3. The results show the solids collecting near the outer wall of the bend and lowering the Sauter mean diameter because of increased attrition.

Another mimics well-defined stress conditions in simple setups to identify basic attrition mechanisms [3–5].

The current study demonstrates the use of a more fundamental and scientific approach to study particle attrition [6]. It incorporates an experimental program with CFD modeling of the gas–solid system. Experiments are carried out on the Malvern Mastersizer DPF, a laboratory-scale dry powder feeder and particle size analyzer from Malvern Instruments, U.K. The Eulerian granular multiphase model is used with the new population balance (PB) module in FLUENT 6.3 software to simulate the motion of the solid and gas phases and attrition within the device. The numerical model makes use of a semi-empirical expression for computing the breakage of solid particles. This expression involves the impact velocity of solid phase particles as they strike the wall and a small set of parameters that are obtained by fitting the model to experimental data.

The particle size distribution (PSD) is an important characteristic of a powder system because it plays a role in the final product quality. It is routinely measured by laser diffraction using bench-top equipment such as the Malvern Mastersizer. The powder under test is fed using a vibrating feeder and then suspended by a jet of compressed air whose pressure can be varied. Increasing the air jet pressure produces a finer PSD as a consequence of more extensive attrition. Below the suspending jet but upstream of the laser diffraction measurement chamber is a pipe bend, a key part of this lab-scale pneumatic conveying system. It generally is recognized that during pneumatic conveying, the particles experience extensive impact loads at the bends because the flow direction is changing [2]. For this reason, the initial CFD calculations were of the bend where the particle size distribution could be computed using the population balance module.

For the experiments, granule samples were analyzed at different inlet air jet pressures ranging from 0.5 to 2 barg

(bars gauge). Moments m_0, m_1, m_2, and m_3 of the original PSDs were evaluated using the relationship:

$$m_k = \sum_{i=1}^{100} n_i L_i^k$$

in which n_i is the volumetric number of particles in class i having characteristic size (diameter) L_i. The moments are compared in Table 1 for the range of jet pressures tested. All of the moments increase with increasing inlet jet pressure as a consequence of attrition, with the exception of m_3. This moment is a relative measure of the preserved volume of granules, so should be independent of the inlet pressure. A comparison of the Sauter mean diameter, $d_{32} = m_3/m_2$, widely used to characterize a PSD, also is presented in the table. As expected, the increased attrition at higher jet pressures is in evidence as the Sauter mean diameter steadily drops.

P_{in} (barg)	0.5	1.0	1.3	1.5	2.0
$m_0 \cdot 10^{-15}$ [#/m^3]	7.744	9.982	13.010	14.442	20.468
$m_1 \cdot 10^{-9}$ [m/m^3]	6.164	8.301	11.198	12.884	17.762
$m_2 \cdot 10^{-4}$ [m^2/m^3]	3.304	3.978	4.672	5.120	5.950
m_3 [m^3/m^3]	1.910	1.910	1.910	1.910	1.910
$d_{32} \cdot 10^6$ [m]	57.81	48.01	40.88	37.30	32.10

Table 1: First moments of the particle size distribution for a range of experimental inlet pressures

CFD results from a 2-D model of the bend were used to provide insight on the behavior of the solids as they travel through this region of the dry powder feeder. In particular, they show that the velocity gradients are highest in the bend. Contours of the Sauter mean diameter and volume fraction of solids, also computed using the CFD model, show a significant decrease of the particle diameter at the bend, suggesting increased attrition in this region. X-Y plots of the solid velocity magnitude and of d_{32} along the lower wall provide further insight into the flow and breakage phenomena in the process. These plots also indicate breakage around the bend of the transport pipe, suggesting that improvements to the transport system are needed. Currently, Merck and ANSYS are continuing studies that incorporate multiple breakage in 3-D geometries to further evaluate both lab-scale and plant-scale powder handling equipment.

Parametric studies also were performed to investigate the impact of different model parameters on the extent of breakage and resulting shape of the PSDs, as characterized by the PSD moments. The particle breakup model satisfactorily predicted the experimental Sauter mean diameter, but the lower moments, m_0 and m_1, were under-predicted. This could be due to breakage that results from an erosion mechanism similar to that reported in the literature for a stirred ball milling application [7].

The methodology illustrated here allows engineers to correlate the observed changes in particle size with the shear forces or impact velocities within the system. It is assumed that analogous physically based models combining properties of the gas–solid flow with the PB
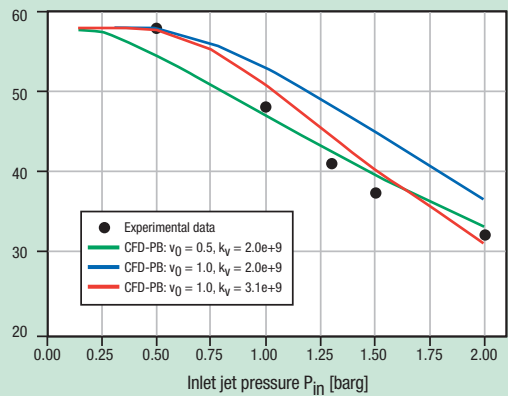
Comparison of the Sauter mean diameter computed from experimental data with that predicted by the CFD model using three different sets of fitting parameters
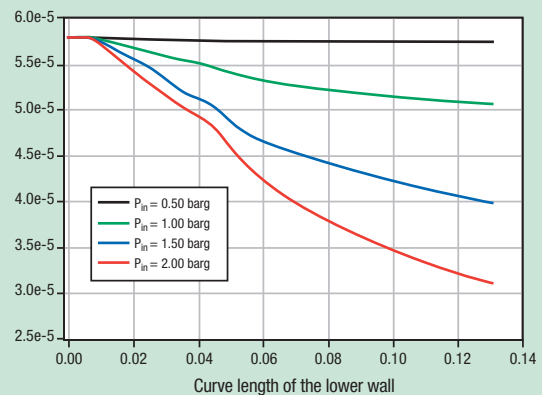
Calculated Sauter diameter of solid particles along the lower wall of the powder feeder at different inlet jet pressures, illustrating the increased attrition at higher pressures

models can be employed after fitting to experimental data for predicting attrition and breakage in large-scale pneumatic conveying systems and to assess the suitability of a given batch of granular material for larger scale processing. ■

References

[1] Chapelle, P.; Abou-Chakra, N.; Christakis, N.; Bridle, I.; Patel, M. K.; Baxter, J.; Tuzun, U.; Cross, M.: Numerical Predictions of Particle Degradation in Industrial-scale Pneumatic Conveyors. Powder Technology 143-144: 321–330, 2004.

[2] Kalman, H.: Attrition of Powders at Various Bends During Pneumatic Conveying. Powder Technology 112: 244 – 250, 2000.

[3] Salman, A. D.; Hounslow, M. J.; Verba, A.: Particle Fragmentation in Dilute Phase Pneumatic Conveying. Powder Technology 126: 109–115, 2002.

[4] Zhang, Z.; Ghadiri, M.: Impact Attrition of Particulate Solids: Part 2. Experimental Work. Chem. Eng. Sci. 57: 3671–3686, 2002.

[5] Gentzler, M.; Michaels, J. N.: Impact Attrition of Brittle Structured Particles at Low Velocities: Rigorous Use of a Laboratory Vibrational Impact Tester. Chem. Eng. Sci. 59: 5949–5958, 2004.

[6] Rajniak, P.; Dhanasekharan, K.; Sinka, C.; MacPhail, N.; Chern, R.; Fitzpatrick, S.. Modeling and Measurement of Granule Friability. Fifth World Congress on Particle Technology (Conference Proceedings CD Vol.2): 23-27, Orlando, FL, U.S.A., April 2006.

[7] Diemer, R. B.; Spahr, D. E.; Olson, J. H.; Magan R. V.: Interpretation of Size Reduction Data via Moment Models. Powder Technology 156: 83–94, 2005.

Solid Suspensions Get a Lift

A high efficiency hydrofoil is designed using CFD and multi-objective optimization software.

By Nicolas Spogis, Engineering Simulation and Scientific Software (ESSS), International Trade Center, Brazil
and José Roberto Nunhez, Department of Chemical Engineering, University of Campinas (UNICAMP), Brazil

Mixing vessels are widely used in the chemical, petrochemical, pharmaceutical, biotechnology and food processing industries to optimize mixing and/or heat transfer. Mixing must be efficient, precise and reproducible to ensure optimum product quality. Quantities of interest may include mixing times, power draw, local shear and strain rates, and solids distribution. In the chemical industry, for example, fast mixing of reactive substances is desired to achieve an efficient reaction. The optimum impeller should produce a highly turbulent flow to reduce segregation and minimize the mass and energy transport limitation for

The flow generated by one of the initial impeller blades (left, similar to a PBT blade only with a wing profile cross-section) shows tip vortices, which increase drag and lead to increased power consumption. The optimized impeller blade (right) produces no such flow pattern, so it is more economical to use.

Solids distribution in a stirred tank driven by a standard PBT (left) is compared to that in the same tank driven by the optimized impeller (right); the cloud height is greater when the optimized impeller is used because the solids that collect under the PBT impeller are more uniformly distributed.

the chemical reaction. For biochemical applications, on the other hand, there is a need to carefully suspend microorganisms in bioreactors so the cells are not exposed to high shear rates, which can lead to their destruction. Over the years, the wide variety of mixing applications has led to a wide variety of impellers and vessels, making the choice of the right mixing equipment a challenge for the process engineer. In a project recently completed at the University of Campinas (UNICAMP) in Brazil, an optimization procedure was applied to the design of an impeller to illustrate how this approach can lead to more efficient mixing processes in general.

The suspension of solid particles in a stirred tank was used to illustrate the methodology. Solid–liquid mixtures appear in applications ranging from the mining industry to the pharmaceutical industry. The parameters that affect solid suspensions are the shape and size distribution of the solid particles, the solid concentration and density, and the liquid density and viscosity. When choosing the right equipment for solid–liquid mixing, it is important to understand how the flow pattern generated by the impeller affects the solids distribution within the vessel. Abrasion and impeller wear also are important factors to consider. On the economic side, minimization of the power required to achieve a desired distribution is important, as is the cost and expected lifetime of the impeller and vessel materials. All of these aspects of mixing depend on the geometry of the equipment, the solid and liquid properties, and the impeller speed.

The engineers at UNICAMP used ANSYS CFX software for the project along with modeFRONTIER™, a multi-objective design optimization tool from

A close-up view of the optimized impeller and the flow it produces shows the wing profile and a variable pitch angle from the root to the tip.

A prototype of the optimized impeller

the definition of the Pareto frontier, a set of equally optimal designs, at the end of the procedure. Traditional optimization algorithms transform multi-objective problems into mono-objective ones using weighted sums of objective functions. This research did not select the best impeller shape in these terms. Rather, the optimization algorithm allowed for the entire Pareto frontier, that is, the complete set of acceptable solutions, to be determined, so that all designs that were not dominated by others could be analyzed.

The optimization process was divided into two main steps:

1. A real optimization step, in which the objective functions and constraints were evaluated by the CFD approach.
2. A virtual optimization step, in which well-behaved response surfaces were used to extrapolate the initial results, saving computational time.

The PBT45 has a low discharge angle and, hence, a low pumping effectiveness and poor solid suspension. The optimized impeller, on the other hand, determined by the process described above, was found to have a high discharge angle (parallel to the shaft), resulting in both a higher pumping effectiveness an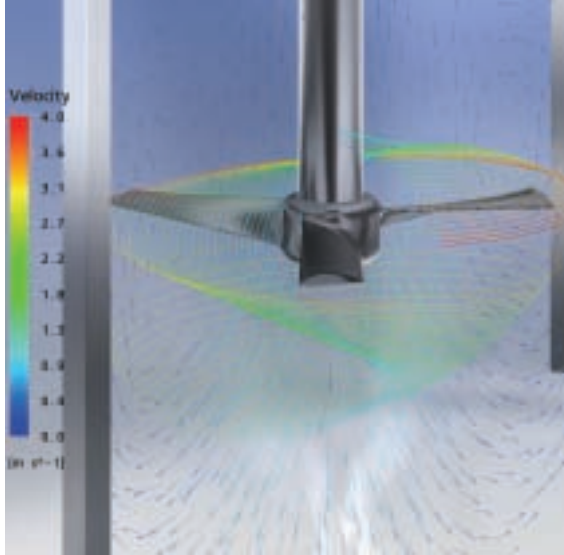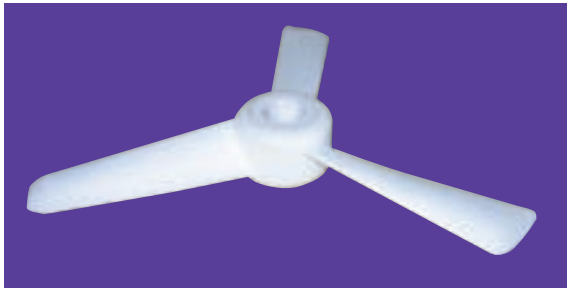d a more uniform solid suspension. When compared to the PBT45, the optimized impeller has a reduced solid accumulation at the bottom of the vessel, even directly below the shaft. The variance of the concentration is low as well, indicating a more homogenous suspension. Specifically, for the optimized impeller, the solid concentration variance was reduced by 48.5 percent and power consumption was reduced by 84.4 percent while pumping effectiveness was increased by 410.2 percent. In addition, an experimental validation was carried out to validate the numerical results, and very good agreement was obtained. ■

www.feq.unicamp.br/~nunhez

the Italian company ESTECO. CFD has long been used in the process industries for mixing analysis because it is more cost-effective than experimental work and provides a reliable alternative to the guess-work associated with process scale-up. The use of CFD for impeller optimization makes the technology even more powerful for delivering benefits ranging from cost-savings to improved product quality.

For the system of interest, a 120° sector tank model was created and a hybrid mesh of approximately 700,000 cells was created. Turbulence was taken into account through the shear–stress transport (SST) k-ω model coupled with the streamline curvature option. These modeling choices were made so that the impeller blade flow separation could be captured in an accurate manner. The SST k-ω model combines the advantages of both the k-ε and standard k-ω approaches, ensuring a proper relationship between the turbulent stress and turbulent kinetic energy. The multiple frames of reference option was used for the steady-state model of the rotating impeller, with a frozen rotor reference frame change applied at the interface.

A robust, stochastic algorithm in modeFRONTIER called MOGAII was used for the automated optimization process. This multi-objective, constrained shape approach allowed for seven design variables with two nonlinear constraints. The designs were compared in their ability to meet two objectives: to increase the impeller effectiveness, defined as the ratio of pumping number to power number (in other words, the ratio of the pumping capacity to power consumed, normalized to be dimensionless), and to improve the homogeneity of the liquid–solid mixture (by increasing the cloud height). The initial group of impeller shapes had well-known profiles, such as the NACA0012. The performance of the final optimized design was compared to that of a standard four (flat)-bladed, 45˚ pitched blade turbine (PBT45).

MOGAII's search method has two desirable aspects. First, it allows global solutions to be found and second, it guarantees an actual multi-objective optimization and allows for

The Many Colors of Glass

Numerical simulation helps guide the color change process in the glass industry.

By Mustafa Oran, SISECAM Research Center, Turkey

Colored glass products have many commercial applications. One way to change the color of molten glass is to add a colorant material to one particular channel of the glass furnace called the forehearth (F/H). The forehearth is where molten glass is conditioned while being transported to the downstream forming machines. By adding colorant to one of the forehearth channels, the original color of glass can remain unchanged in the melting tank. A CFD project has been under way at the Research Center of SISECAM to develop a numerical model for the coloration of glass melt in an F/H for the production of table-

ware products. The model simulates the coloration phenomenon of the glass melt by calculating the distribution of the colorant agent in the glass melt as it flows through the channel.

Coloration is essentially an unsteady mixing process of two or more fluids resulting from natural (diffusion) and forced (advection) mechanisms. Molecular diffusion can be from a point source in a static field or from a point source in a velocity field in which relative motion exists between the source and the field. Therefore, the spread of a colorant, referred to as "frit," in molten glass can be obtained by solving Fick's second law for

diffusion when the source and other boundary conditions are defined. The advection process is driven by the movement of the fluid, which is molten glass in the case of the forehearth. Because the colorant is carried in all directions by the flow in the F/H, a 3-D time-dependent species transport equation must be solved to track its distribution throughout the glass melt.

In addition to the CFD work, a set of experimental studies also has been performed at SISECAM to obtain the diffusion coefficient of the frit in the molten glass. Measurements made use of a laboratory setup based on image processing of a time-lapsed
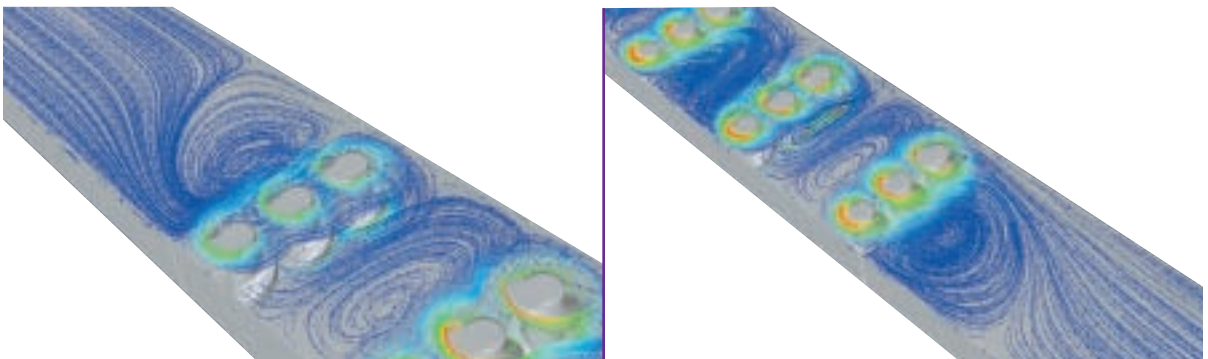
Two views showing the pathlines of molten glass flow in the vicinity of the stirrers in the forehearth

video record. The raw image data representing the rate of change of area occupied by the frit on the molten glass surface for different temperature values are digitized and transformed to a curve representing the diffusion coefficient of the frit as a function of temperature. This value is used in the numerical model in the form of a polynomial function.

The frit is fed to the glass surface from the top of the F/H through a hole. The F/H has a mixing zone, where 12 stirrers are located in four banks. A rotating tube is located in the spout section at the downstream end of the F/H to generate a gob for the production of the glass item at the forming station. One of the main aspects of color control is the requirement of homogeneity of the frit in the glass melt to obtain color uniformity in the end product. Another goal is to achieve on-time delivery of the end product with a target color value. Because the frit is added near the end of the entire glass process, a strong stirring action is required to create uniform mixing in a short time. Two different configurations of screw stirrers that rotate in the clockwise and counter-clockwise directions are used in the numerical model.

A typical forehearth was chosen for the numerical solution, which was carried out using FLUENT software.

For the first phase of the simulation, the initial velocity distribution of the glass melt was obtained using a steady-state approach. The multiple reference frames (MRF) model was used to simulate the rotational motion of the stirrers, and the rotation of the tube in the spout region also was taken into account. These results show that strong vortices occur between adjacent banks of stirrers. In general, the glass melt is pumped upward along the axis of the stirrers and downward in the mixing vortices between the stirrers. The up-pumping action of the stirrers is necessary because the frit used in the process is denser than the glass melt. The results reveal that the two different configurations of stirrers create the same circulation effect in the glass melt in the vertical direction. This flow pattern enhances the mixing between the molten glass and frit so that a homogenous blend can be generated.

In the second part of the numerical study, the transient tracking of the frit concentration was performed. The sliding mesh model was used to capture the motion of the stirrers. The results show

30 minutes

60 minutes

120 minutes

180 minutes

300 minutes

600 minutes

Concentration distributions of the frit along the forehearth at different times

30 minutes   180 minutes

300 minutes   600 minutes
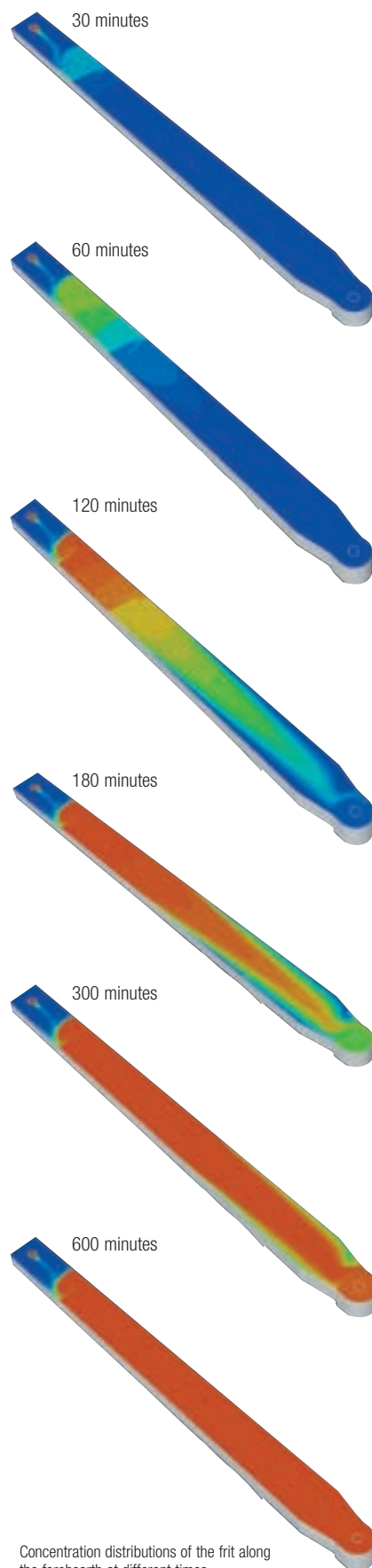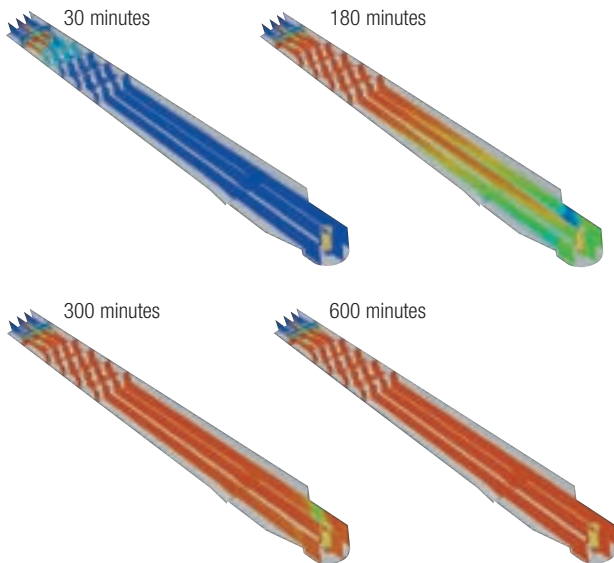
The vertical spread of the frit below the glass surface at different times

Developing Power

Integrating ANSYS technology with other software enabled researchers to efficiently assess component reliability for ceramic microturbine rotors.

By Stephen Duffy, Connecticut Reserve Technologies Inc. Ohio, U.S.A.

that a considerable amount of frit reaches the stirring zone 30 minutes after feeding, following the flow pattern of the glass melt. Axial slices of frit concentration show that the initial direction of the frit motion is toward the bottom of the channel while only a small amount of frit travels near the glass surface. This result occurs because of the high density of the frit, which tends to sink toward the bottom of glass immediately after feeding. As the time proceeds, most of the frit is pumped upward as it passes through the stirring zone. There, the frit and molten glass are progressively mixed and a uniform distribution is gradually achieved.

Mixing between the glass melt and frit is accelerated in the stirring zone and, after one hour, a cross-sectional view of each stirrer bank shows a more or less homogenous frit distribution. As the coloration process continues, the target concentration of frit is obtained homogenously in the stirring zone before three hours have passed. The simulation shows that the target value of frit (0.5 percent of the pull rate) is uniformly distributed along the F/H well before 10 hours.

The color of the final glass gob does not change during the first 90 minutes. After that point the glass gradually changes color, but the production glass is not discarded because the early color changes are not visible and the product can still be accepted commercially. The simulations show that the target value of frit concentration at the end of the forehearth is obtained after nine to 10 hours, whereas the real process in the plant starts to accept the new color value after eight to nine hours. Since the variation in frit concentration during the final hour is very small, the numerical simulations can be safely accepted for practical use. ■

www.sisecam.com

Microturbines a few inches in diameter are critical components in compact co-generation units that produce electrical power. These modular distributed power systems are intended to operate on-site at manufacturing plants and other facilities as a source of economical and reliable electrical power, thus avoiding the high cost and vulnerability to power outage of public utility lines.

Advanced structural ceramics such as silicon nitride enable microturbines to operate at higher temperatures than conventional metal alloys, which translates into significant fuel savings and emissions reductions. However, ceramics exhibit large variations in fracture strength, particularly with inherent flaws resulting from various surface treatments. Accounting for these complex statistical strength distributions will lead to more accurate predictions of expected component life, expressed as component reliability as a function of time.

Two algorithms work in conjunction with one another to provide the probabilistic design approaches required to determine ceramic reliability predictions. The ceramic analysis and reliability evaluation (CARES) algorithm originally was developed at NASA Glenn Research Center to determine component reliability based on temperature and stress fields. The CRT WeibPar algorithm was developed at Connecticut Reserved Technologies Inc. to determine the probability of failure for ceramic components.

These algorithms were upgraded under the U.S. Department of Energy (DOE) Distributed Energy Program to specifically utilize features of ANSYS Structural analysis software. As part of the program, which is administered by Oak Ridge National Laboratory, engineering consulting firm Connecticut

Systems that Can **Take the Heat**

Surface Type 2
As-fired surfaces

Surface Type 2
Ground surfaces
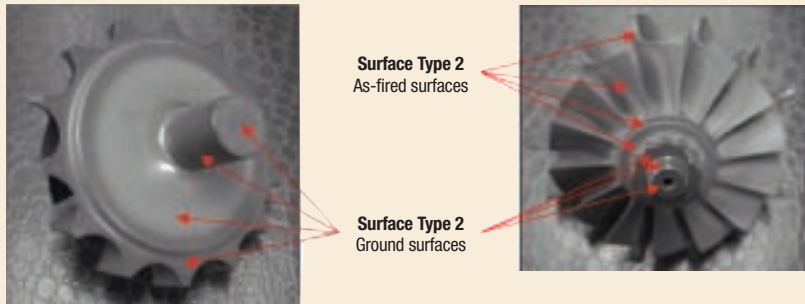
Photos of the Ingersoll-Rand microturbine rotor shown from the bottom (left) and top (right). All surfaces on the bottom and three surfaces on the top are ground. All others are as-fired.

Reserve Technologies used the software in a project to determine the material requirements for a blade that was being developed for a microturbine manufactured by Ingersoll-Rand Company.

One of the challenges in the project has been defining and implementing a method that establishes Weibull distribution metrics for silicon nitride suppliers based on the particular component. In establishing these metrics, service stress states from the various treated surfaces of a rotor blade must be combined with a stipulated component reliability to develop material performance curves. These curves must be scaled to standard ceramic bend bar test specimens, making component requirements more readily understood by material suppliers.

Through the use of ANSYS Parametric Design Language (APDL), surfaces of a rotor component with specified finishes are identified, the ANSYS results file is queried and stresses are mapped to the relevant element surfaces. Failure data is analyzed using WeibPar. Using information generated by ANSYS (geometry and stress state), the CARES algorithm computes component reliability. The openness of ANSYS technology and the ease of integration with other software enabled the ANSYS, CARES and WeibPar programs to operate together in a smooth and efficient manner.

The resulting design approach has allowed changes and improvements in system requirements to take place readily in parallel with enhancements in material properties. In the past, this was typically a series process in which system engineering would follow improvements in ceramic materials. Now material characterization maps can be quickly generated for a given component under specified operating conditions. The information can influence the goals of a ceramic materials development program and better guide engineers toward an optimal design.

"ANSYS continues to be critical iterative software for design optimization and probabilistic lifing of structural ceramic components under consideration for advanced turbine engines," notes Dr. Andrew Wereszczak, senior staff scientist at Oak Ridge National Laboratory. "Ultimately, the ever-increasing versatility and capabilities of ANSYS are allowing structural designers to increase their confidence in, and rate component designs." ■
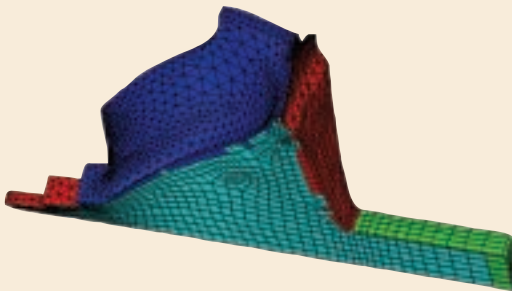
The mesh used for the rotor vane analysis
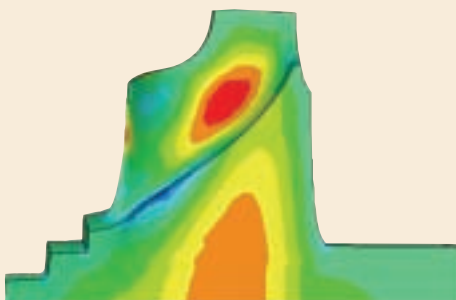
First principle stress (psi) from a cold start on Ingersoll-Rand's ceramic microturbine vane

ANSYS

ADVANTAGE

Spotlight on Engineering Simulation in the
Sports and Leisure Industry

Sporting Swifter, Higher and Stronger Performances with Engineering Simulation

Computer-aided engineering plays a major role
in the world of sports.

By Keith Hanna, ANSYS, Inc.
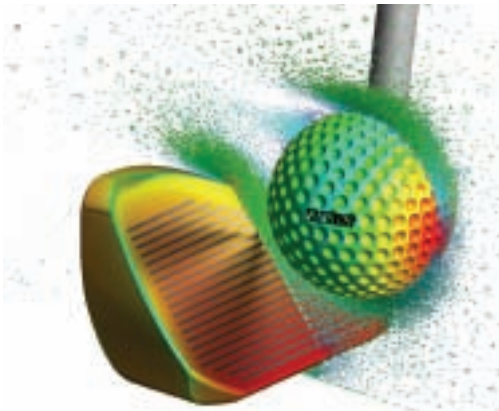
Close up of airflow vectors and surface pressure predictions on a golf ball
and #3 iron just after being struck, computed using ANSYS CFX software

Close up of a golf ball deforming after being struck in a fluid structure
interaction simulation, computed using ANSYS AUTODYN software

The sports and leisure Industry has seen some profound changes during the last 25 years, especially in the areas of new product design, innovation and development. Indeed, there has been an explosion of professional sporting and leisure activities, driven by consumers having more disposable income to spend and multi-channel 24-hour TV, hungry for content and information. In the last decade alone, the amount of money pouring into elite sport has hit staggering heights. The seven-time German Formula 1 Motor Racing World Champion, Michael Schumacher, has been estimated to have earned $1 billion throughout his 15-year career, and the American golfer, Tiger Woods, is not that far behind. Major sporting events are now linked with major business opportunities, and the worldwide sports and leisure industry is estimated to be worth about $500 billion per year while growing at 3 percent per annum.

The push to involve science and engineering in sports has been led by motor racing in a quest for that elusive fraction of a pecentage point improvement in performance that can lead to victory. New engineering tools and disciplines like computer-aided engineering (CAE) are now major transforming agents for this industry. CAE allows for virtual design and testing techniques to be applied to all aspects of sport and leisure equipment development. Modern CAE software tools provide a cost-effective way of assessing new products and product innovations in what were previously lengthy product design turnaround times.

Many elite athletes, teams and sports equipment manufacturers now are realizing that they can derive competitive aerodynamic and structural advantages from advanced fluid flow and structural modeling technologies. Computational fluid dynamics (CFD) in particular is an integral part of the CAE process in many sports today, where the technology leads to performance gains that easily justify the financial outlays for hardware and software.

It is generally known that cyclists riding in a pack expend significantly less energy when drafting behind another rider. In a recent study by Dale Apgar at Dartmouth College, CFD was used to study four cyclists in a line (left) and in a small peloton (right). The results showed that in the line configuration, the last cyclist experiences 46 percent less drag than the lead rider. In the four-man peloton, the two side riders experience more drag than the first rider, suggesting difficulty for other cyclists to overtake the leader. The result may not be true for larger pelotons. Pressure contours on the individual cyclists, shown above, were used to compute the drag.

An increasing drive toward cheaper, easier to use CAE software coupled with the ready availability of increasingly more powerful computers have led to an expansion in numerical simulation for numerous sporting applications. CAE now is being used routinely to help explain physical phenomena in both competition and training scenarios. It is indispensable in the design of better equipment, where it is used to improve safety, comfort and efficiency.

In the world of Formula 1 racing, for example, the leading teams are pushing toward once unimaginable 1 billion cell CFD calculations. The BMW Sauber F1 Team recently announced the launch of its 1,056 processor supercomputer, Albert[2], one of the largest industrial computing installations in the world, aimed solely at doing CFD calculations. Indeed, the team chose the supercomputer route rather than building a second wind tunnel as their preferred way forward for aerodynamic race car design and improvement.

In the world of America's Cup Yachting, the coast of Valencia, Spain, soon will see some of the richest multinational teams vying to win one of the oldest and most prestigious sports trophies in the world. ANSYS has had the privilege of supplying two teams in the last decade who, between them, have been winners of the last three America's Cups: Team New Zealand (twice) and the Swiss team Alinghi. These teams have used ANSYS software to design their ship hulls, appendages and sails to millimeter tolerances. In 2007, nearly all of the Americas Cup competitors will have used ANSYS software in one form or another prior to the start of the competition.

In this sports and leisure industry supplement, a variety of CAE applications are illustrated that emphasize the widespread use and importance of this exciting technology. Both solid mechanics and fluid dynamics phenomena are represented in applications that range from bicycles to alpine skis, ice axes to racing oars, and surfboards to mouthguards. Ventilation schemes for sports arenas are reviewed, as are important design considerations for fitness equipment. In each case, the application has benefited in some way from ANSYS engineering simulation software. With the Summer Olympics in Beijing fast approaching in 2008 and the soccer World Cup in South Africa in 2010, computer simulation will be strongly impacting this industry in the years to come. Whether it is multinational equipment giants or niche elite sport teams, many will recognize the benefits that Simulation Driven Product Development can bring to their business or sporting goals. ■

Suggested Reading

Hanna, R.K.: Going Faster, Higher and Longer in Sport with CFD. 1st International Conference on Engineering in Sport, Sheffield, U.K., 1996.

Catching the **Simulation Wave**

Surfers are using engineering simulation to improve their gear.

Those passionate about a sport are always trying to enhance the experience for themselves and for others. For years, the search for a perfect board has been a fervent quest among surfing enthusiasts. A surfboard's performance is based on the board shape as well as the shape and size of the fins. Determining the best geometric details of the board and fins involves accounting for the surfer's weight in addition to the size and nature of the waves. Board design traditionally has been a trial-and-error process built on the knowledge and art of the shaper. However, surfers in the engineering field on both sides of the Atlantic Ocean are now using their skills in an attempt to quantify and improve surfing performance through engineering simulation.

Designing a **Better Surfboard**

By Professor Len Imas, Davidson Laboratory
Stevens Institute of Technology, New Jersey, U.S.A.

Modern surfboards normally are constructed of polyurethane foam with a wooden stringer in the center for stability. Board-shaped blanks are produced and then fashioned by a designer or shaper. At Stevens Institute of Technology, the object of the SURFace Masters thesis project, conducted by Brian Sweeney (Product Architecture and Engineering Department (PAE) Masters '06) and Dror Kodman (PAE Masters '06), was to custom fit a surfer to a specific surfboard based on a set of location-specific conditions and by using hydrodynamic analysis to quantify the board performance. Hence, this project laid the groundwork for more advanced board design by adding a level of engineering science to the existing practice.

While the hydrostatic characteristics of a surfboard are important for stability as the surfer starts and gets under way, the dynamic characteristics dominate as the surfer accelerates and maneuvers on the face of a wave at speed.

Optimized board

Ricker type: **Medium**

Rail type: **Soft rail**

Bottom surface: **Flat**

Rail type: **Hard rail**

Bottom surface: **Double concave**
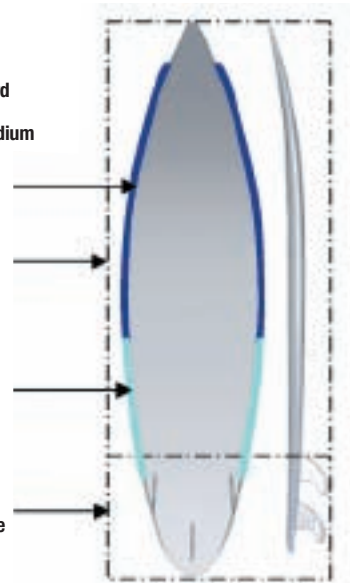
A diagram of a typical surfboard, showing parameters such as the side rails and bottom surface types
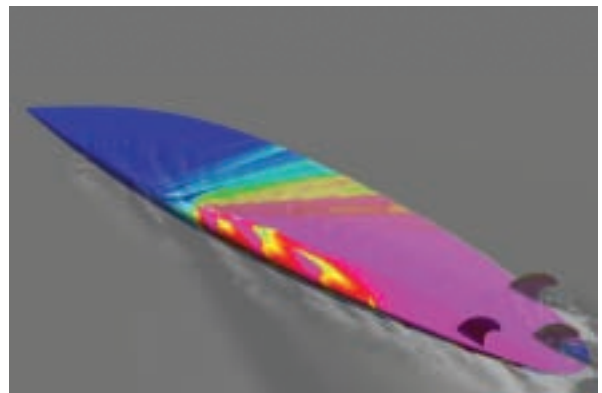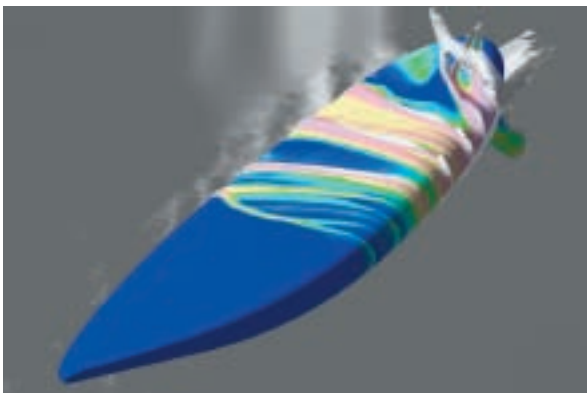
The image on the left shows water volume fraction contours on the top of the board surface with the free-surface around the board. The image on the right shows water volume fraction contours on the bottom of the board surface with the free-surface around the board.

As the board speed increases to the point where the board planes across the wave surface, the contribution of buoyancy becomes small in comparison to the dynamic lift generated.

Surfboard design is a complex association of many variables, all of which are relevant to the performance characteristics of the board. Some of the key features evaluated in this study that directly affect performance are the side rails (which define the outer shape), the rocker dimensions (which define the curvature at the front end) and the bottom surface configuration (which defines the overall contour of the underside of the board). In addition to speed, these parameters affect the board's orientation in the water and hence the resulting dynamic pressure on the bottom surface. This in turn drives the dynamic loads generated by the surfboard that allow it to plane and perform maneuvers.

A matrix of these features was created and configurations were simulated using ANSYS CFX software. The transient CFD model included turbulence, free surface prediction (between the air and water) and buoyancy.

Fixed values for the trim angle and yaw angle were used, while the roll angle was varied from a low value to the most extreme value observed. The roll angles were chosen to correspond to three standard surfing scenarios — speed generation, basic maneuvering and extreme redirection. The relevance of each of these is dependent upon wave type and user surfing abilities. The CFD simulations generated data in the form of lift force, drag force, and moment coefficients. From the data gathered it was possible to quantify the performance of each board configuration in the form of maneuverability, stability and drive. The results for each calculation then were graded and from these results it was possible to determine an optimal board based on surfer skills and wave type. The study went even further. From the results table, hybrids between the local ranges of drive and sensitivity were used to construct an optimized board for a range surfing conditions. ■

Developing **More Adaptive** Fins

By Dave Carswell, Nick Lavery and Steve Brown, Department of Materials Engineering University of Wales, U.K.

Research on the hydrodynamics of surfboard fins has been under way at the University of Wales, Swansea, for a number of years. The primary aim has been to gain a better understanding of how the moving water interacts with different shapes and sizes of fins and how the hydrofoil shape of the fin affects the drag and lift forces that are generated. The underlying philosophy is that higher lift-to-drag ratios may result in greater velocities in the water when surfing and hence better performance.

Surfboard fins are complex three-dimensional hydrofoils. Creating solid models of fins, which are essentially scaled down versions of aircraft wings, can take a long time using commercial CAD programs. As a remedy, custom software called Fin Designer™ (www.datdesigner.co.uk/index.htm) has been developed specifically to reduce fin design time to a matter of minutes, while allowing the designer to retain parametric information on components that are important in design analysis, such as the NACA foil shapes at various cross-sections.

An experimental apparatus has been built to measure lift and drag forces on objects in an existing square-section flume, and FLUENT software is successfully predicting these forces at experimentally limited Reynolds number flows, substantially lower than those found in reality. However, once the CFD models have been fully validated, the research team

A finite element analysis is used to compute the deformation on the fin, illustrated here as displacement magnitude, the major component being the displacement in the z-direction (into the page).

will use FLUENT tools to examine higher flow rates (up to 12 to 15 m/s) that are not achievable in the laboratory using the current equipment.

Of particular interest at higher flow rates are the effects of fin deformation on the fin hydrodynamics. Through user-defined functions (UDFs) in FLUENT software, a customized finite element stress analysis code has been coupled to the transient flow analysis. This allows the researchers to compute the dynamic deformations caused by the surface pressures on the fins and the subsequent effect of the deformations on the surface pressures and related hydrodynamics. The objective of this phase of the project is to help design a new generation of fins which will have intelligently adaptive sub-structures that retain hydrodynamic lift even when deforming. This new evolution of fin design would greatly enhance fin performance. ■

Contours of dynamic pressure on a plane slicing through the fin
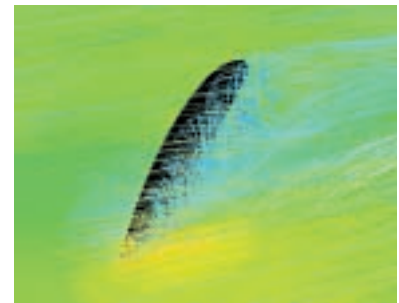
Pathlines, colored by velocity magnitude, illustrate the strong recirculation that occurs on the downstream side of the fin.

Giving
Ski Racers
an Edge

ANSYS Mechanical software is used to analyze the dynamic properties of skis.

Image courtesy Stöckli Swiss Sports AG.

Christian Fischer[1], Mathieu Fauve[2], Etienne Combaz[1,2], Dènes Szabo[2],
Pierre-Etienne Bourban[1], Vèronique Michaud[1], Christopher J.G. Plummer[1],
Hansueli Rhyner[2] and Jan-Anders E. Manson[1]

[1]Laboratoire de Technologie des Composites et Polymères (LTC)
Ecole Polytechnique Fèdèrale de Lausanne (EPFL), Switzerland
[2]WSL, Institute for Snow and Avalanche Research (SLF), Switzerland
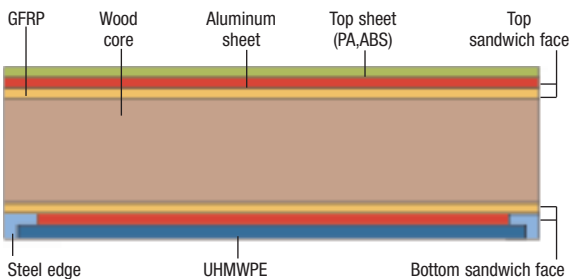
Cross-section schematic of a ski's sandwich structure. Image courtesy Stöckli Swiss Sports AG.

Images generated by FEA for (top) the second flexural mode of the front part of a ski and (bottom) the first torsional mode

As in most sports, successful performance in top-level skiing requires a combination of highly developed motor and perceptual skills from the athletes in addition to outstanding equipment. A principal aim of ski manufacturers in recent years has been to control vibration in various ways. By observing downhill racers on icy slopes in slow-motion, one concludes that low-frequency high-amplitude deformation has a detrimental effect on the ski-snow contact, decreasing control and speed. Skis completely devoid of vibration nevertheless do not provide adequate sensitivity for the athlete, making it crucial to discriminate between those frequencies that should be damped to increase performance and those that are important for the skier's "feel." Though numerical simulations have shown promise for the investigation of ski properties [1,2], little attention has been paid to the influence of the constituent materials on the dynamic response of skis.

Each constituent material of the ski has a particular purpose. The top sheet, usually polyamide (PA) or acrylonitrile butadiene styrene (ABS), is mainly a protective layer. The wood core, which has a non-uniform thickness that provides a smooth bending profile, plays an important role in damping, whereas aluminum and glass fiber reinforced polymers (GFRPs), which constitute the upper and lower faces, provide stiffness in bending and torsion. The running base is commonly made of ultra-high molecular weight polyethylene (UHMWPE) to give optimum sliding behavior. Finally, hardened steel edges are positioned on both sides of the ski to provide good control during a turn. In the present work, the influence of the topsheet on the ski's dynamic properties has been investigated using a combination of numerical simulations and

experimental measurements on skis with (referred to as ski+) and without (referred to as ski-) a topsheet.

In initial tests, the skis were clamped at their center locations lengthwise, placed in a cold room and their vibrational response characterized at temperatures between -20 and 25°C. Ski+ exhibited lower first resonant frequencies than ski- over the entire range of temperatures. This was attributed to the increased damping of the ski in the presence of the polyamide topsheet, which generally tends to reduce the resonant frequency [3]. A minimum in resonant frequency, equivalent to a maximum in damping capacity, was found to occur at 0°C for ski+, whereas the resonant frequency of ski- increased monotonically with temperature.

Dynamic mechanical analysis (DMA) was used to measure the damping capacity (loss factor) of the topsheet as a function of temperature at various frequencies. Each damping curve displayed a clear maximum value, or peak, at a temperature that increased as the frequency increased. At a frequency of 10-15 Hz, a damping peak was observed at approximately 0°C, which was consistent with the results of the vibrational response testing. To gain an idea of the dynamic properties of viscoelastic materials at frequencies beyond those that are directly accessible to DMA, time–temperature superposition is often used [4]. In the time–temperature superposition methodology, the damping capacity is plotted as a function of the logarithm of the frequency at different temperatures. These curves may then be superposed for any chosen reference temperature by shi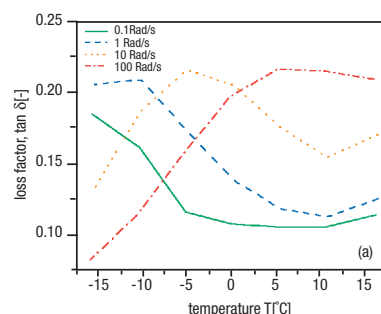fting them along the frequency axis to give a single "master curve" that covers an extended range of frequencies. Using the time–temperature superposition methodology and a reference temperature of 0°C, a clear damping peak was seen to occur for the topsheet material at approximately 13 Hz, again consistent with the results of the vibrational response testing on the skis.

Elasticity-based FEA was used to model the room temperature response

of the skis in a configuration identical to that used in the vibration response testing, i.e. clamped at their center locations lengthwise. The skis were represented by multi-layer meshes that incorporated elastic material parameters that were inferred from DMA measurements taken at room temperature and a frequency of 1.5 Hz. The agreement between the calculated and observed resonant frequencies for the first two vibration modes was good. For higher modes, however, the agreement was poorer. This was attributed to the viscoelastic nature of the polymer-based materials in the sandwich structure (glues, composite laminates and wood), which has not yet been included in the calculations but becomes increasingly important as the frequency increases.

The strong influence of the topsheet on the overall dynamic properties of the ski not only shows that thin layers of viscoelastic materials can have an important influence on the damping behavior of the structure, but also implies that certain specific resonance frequencies can be selectively damped by using polymers with damping peaks in the vicinity of the target frequency. In future work, it will be necessary to introduce more complex models that take into account the viscoelastic nature of the polymeric components. With knowledge of the dynamic response of skis to their constituent materials in hand, designers will have the opportunity to move one step closer to selectively improving ski performance for specific race conditions. In addition to purely mechanical aspects, special attention

Comparison of the first resonant frequency of *ski+* (left) and *ski-* (right) at different temperatures. These measurements were carried out using accelerometers placed on the ski. The added weight of the accelerometers reduced the resonance frequencies somewhat but had no effect on the relative influence of the topsheet.

increasingly is being paid to the athlete–equipment interaction, i.e. the influence of the skis' mechanical properties on the "feel" and the perceived performance of the skier [5]. ■

References

1. Clerc, C.; Gaertner, R.; Trompette, P.: Computer Aided Design of Skis. Finite Elements in Analysis and Design 5: 1-14, 1989.

2. Devaux, P.; Trompette, P.: Modal-analysis of a Ski by the Finite-Element Method. Journal of Sound and Vibration 73: 597-600, 1980.

3. Balachandran, B.; Magrab, E. B.: Vibrations. Pacific Grove: Brooks/Cole, 2004.

4. Ferry, J. D.: Viscoelastic Properties of Polymers. New York: Wiley, 1980.

5. Fischer, C. et al.: What Static and Dynamic Properties Should Slalom Skis Possess? Judgments by Advanced and Expert Skiers. Accepted for publication in Journal of Sports Sciences.

The evolution of the main damping peak of the polyamide topsheet as a function of applied frequency (left) and the time-temperature superposition for a reference temperature of 0°C (right)

The above two figures show the third principal stress 1.6ms after impact in a transient FEA analysis and depict the most compressive stresses at the root of the axe teeth; the stress shown is above the yield stress of the material, so any tensile stress present at a later time is a residual stress.
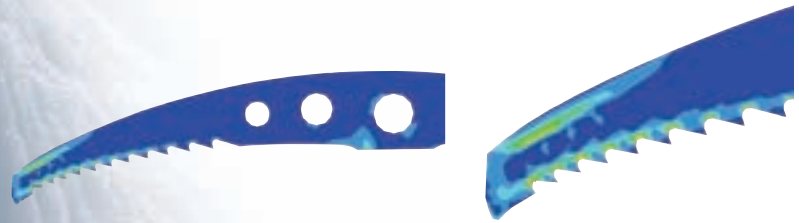
The first principal stress 4.6ms after the initial impact in a transient FEA analysis; the highest tensile stress is apparent at the root of the teeth.

Ice Axe Impacts

By Rae Gordon and Kathryn Franklin
Faculty of Advanced Technology
University of Glamorgan, Wales

Finite element analysis is used to study crack initiation on a serrated blade.

Ice climbers scaling a frozen waterfall use two ice axes, one in each hand, and crampons on their feet. The ice axe consists of a shaft with a spike at the bottom and a head at 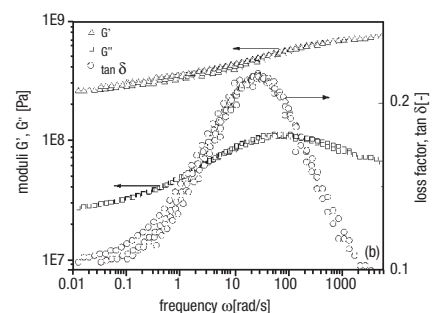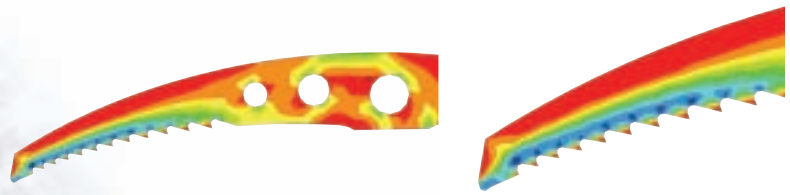the top. The head consists of an adze (or hammer) on one side and a pick on the other side. The adze is used for removing loose ice and the pick is used to help the climber advance up the face of the icefall. To improve the pick's grip in the ice, the bottom edge is serrated with a row of teeth. In operation the ice axe is subjected to various load conditions but primarily to the impact of the pick with the ice, the weight of the climber pulling up on the pick and "torquing" where the pick is placed in a fissure and twisted to achieve purchase.

There have been a number of recorded failures of ice axes, most of which involve fracture brought on by fatigue [1,2]. Typically, fatigue cracks grow from the serrated edge, particularly from the root of a tooth. An initial finite element study using

ANSYS software revealed that a steady-state load applied normal to the end of the pick results in a compressive stress along the serrated edge of the pick with the teeth acting as stress raisers. Fatigue cracks, however, are not normally initiated in areas of compressive stress, but more often are initiated if a tensile stress is present. This can occur if the compressive yield stress for a material is exceeded, resulting in a residual tensile stress. If a residual tensile stress is present at the root of the ice axe teeth, subsequent impact cycles would repetitively expose the tooth area to alternating compressive and residual tensile stresses, ideal conditions for crack initiation and growth. The majority of fatigue literature reports on cracks that are initiated at the site of impact loading, though some also focuses on the phenomena of cracks that are initiated at stress concentrations remote from the impact site.

A transient finite element study using ANSYS Mechanical software

subsequently was performed to study how fatigue cracks not normally initiated in areas of compressive stress develop at the root of the teeth. A worst-case scenario was considered in which the pick strikes against rock. The results examined the effect of cyclic impact loading at the root of the ice axe teeth. This analysis revealed that the compressive stress resulting from the impact load exceeds the yield stress and, hence, results in a residual tensile stress at the root of some of the teeth. The results indicate that the conditions required for a crack to be initiated and fatigue failure to occur on subsequent load cycles are present. Further work is required to determine if fatigue would actually occur. ■

References

1. http://www.thebmc.co.uk

2. Hellen, T.: The Engineering Stability of Metallic Climbing Equipment, in The Science of Climbing and Mountaineering: Chapter 15, Human Kinetics. Messenger, N.; Patterson W.; Brooks D.: Editors, 1997.

Tour de Force!

Aerodynamic gains can be realized by studying the interaction between a bicycle and rider.

By Keith Hanna, ANSYS, Inc.

Lisa Bentley — one of the most successful Ironman triathlon athletes — riding a Cervélo bicycle

Cervélo Cycles was formed in 1995 when two engineers, Phil White and Gérard Vroomen, decided to take their fast time trial bikes to market. Involved in bicycle and human-powered vehicle design since 1986, Vroomen realized that professional cyclists did not have the interest or expertise to develop leading-edge designs with a focus on time trialing and aerodynamics. He also realized that he could not look at the many novel designs put before him and know if they were better or just different. Hence, when an Italian pro cyclist's team approached him to evaluate bikes then available on the open market, he set a design goal for a new bike design: to be unbeatable in terms of its aerodynamics, weight and stiffness characteristics. The one-off design for this particular rider turned out to be a radical bike that pushed the boundaries of existing bikes and tested well. This customized Cervélo bike attracted attention and started to sell itself at triathlon and road racing events. The duo set up their own company, and within two years their bicycles had won numerous triathlons and time trials.

Today many professional athletes use Cervélo bicycles. With eight full-time engineers in their small company, they continue to push the bike design envelope in every way possible while keeping a focus on technology and innovation. Today the rate of development has to be fast in order to stay competitive. The company has a simple mission statement: "To help our customers win races." This mission has come true for their cycling team, Team CSC, which is currently ranked first in the world. Their P3 bike is the most successful triathlon model in Ironman history with more than 20 victories. Other Cervélo bicycles are used by racers in the Tour de France and amateurs touring along city streets and country roads.

Recently, Cervélo approached the ANSYS, Inc. office in Michigan, U.S.A., to employ their extensive aerodynamic experience to fashion a virtual design process using the computational fluid dynamics (CFD) package FLUENT. Phil White notes that in a typical one-hour time trial, perhaps 30 to 60 seconds can be taken off times through aerodynamic improvements to a given bike. Indeed, the biggest aerodynamic gain is usually in the design of the lower half of the bike, where many complex flow interactions occur. The CFD studies demonstrated the benefits to be gained by this approach. Cervélo has acquired a wealth of wind tunnel data over the last decade and is now using it for benchmarking the numerical predictions. CFD is proving to be less expensive than wind tunnel measurements for the amount of data generated and is free of probe interference errors. It can pick up many effects associated with riders interacting with bike frames and can capture rotating wheels and moving ground effects. Looking forward, Cervélo believes that CFD will provide a better understanding of critical side wind effects with yaw angles in crossflows, overall aerodynamics and cyclist packages, racing tactics, and many other riding equipment enhancements. ■
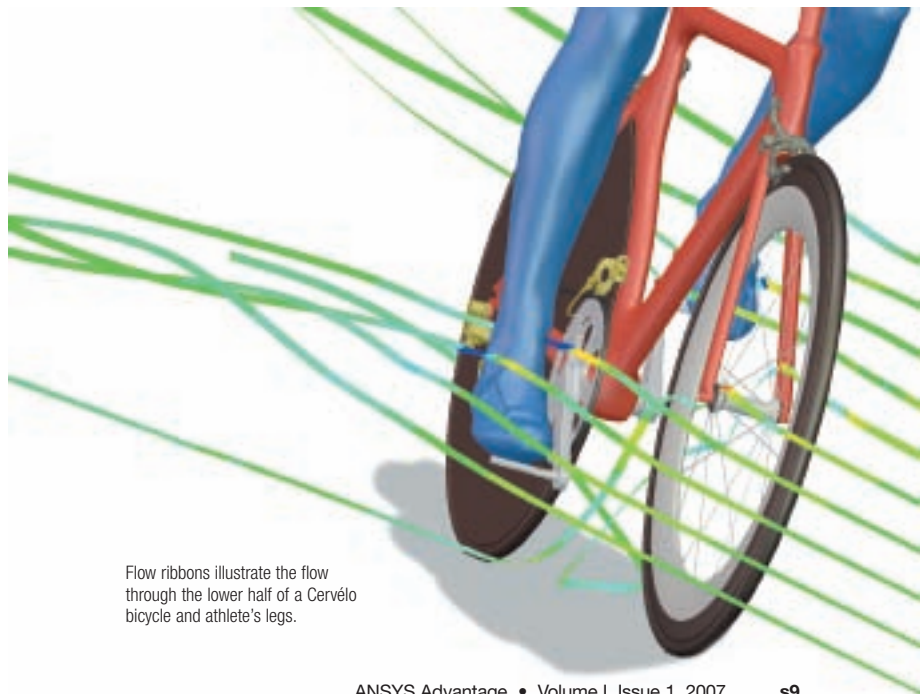
Flow ribbons illustrate the flow through the lower half of a Cervélo bicycle and athlete's legs.

Speeding Up Development Time for **Racing Cycles**

Trek Bicycle Corporation cuts product launch delays with simulation-based design using ANSYS Mechanical software.

By Brian Schumann
Trek Bicycle Corporation, Wisconsin, U.S.A.

Leading cyclists such as Lance Armstrong — six-time winner of the Tour de France — rely on Trek bicycles in the world's premier racing events.
Image courtesy Trek Bicycle Corporation.

With seven consecutive Tour de France titles, six straight 24-hour World Solo Championships and a wide range of numerous other professional wins, Trek enjoys a rich tradition of victory in the world's premier cycling events. Headquartered in Wisconsin, U.S.A., Trek Bicycle is a global leader in bicycle design, manufacturing and distribution, with a broad range of bicycles and cycling products under the Trek, Gary Fisher, LeMond, Bontrager and Klein brand names. From the first hand-built steel touring frames to the revolutionary OCLV carbon fiber molded parts first introduced in 1992, Trek's passion for innovation, quality and performance has led the field with forward thinking and next-generation technology.

Success in this highly competitive industry depends on releasing the right products at the right time. To stay at the forefront, Trek continually strives to design and build innovative products that meet the company's stringent strength and stiffness requirements.

A major challenge in one recent project was to increase the speed to market of a cycle with an assembly comprising an aluminum steer tube bonded with epoxy adhesive into a composite fork that is bolted to the wheel axle. ANSYS Mechanical software was used to accurately predict stress levels in the composite and metal fork assembly. The solution was installed by ANSYS channel partner Belcan Engineering Group, which provided training and applications support for composite materials analysis.

In analyzing the fork assembly, component geometries from SolidWorks® CAD models were imported

Component geometry from a SolidWorks CAD model (top) was imported into ANSYS Mechanical software (bottom) to determined the stresses in the fork assembly made of aluminum and composite parts. The simulation reduced the number of lengthy hardware prototype iterations and enabled the company to meet critical product launch deadlines.

into ANSYS software, in which the Virtual Topology feature was used to construct a unified mesh representing the entire assembly of composite and metal parts. Bonded contact elements automatically configured the mesh to account for epoxied parts, thus avoiding the difficult task of manually adjusting mesh densities and selecting element types. The shell lay-up capabilities of ANSYS software were of particular value in representing the complex material properties of composites in regions containing different numbers of layers aligned in various orientations.

The analysis allowed engineers to simulate changes in the lay-up of the composite parts as well as wall thickness of the aluminum parts much faster. The laboratory test setup was accurately simulated and a composite fork assembly ultimately was created that met all of the design parameters and safety standards. This saved considerable time in developing these components and is allowing critical product launch deadlines to be met. Trek engineers are leveraging this knowledge and experience so that the process can be applied to the design of future products.

Trek recognizes the value of Simulation Driven Product Development and is successfully implementing the approach with ANSYS Mechanical software. As the company moves increasingly toward more globalization in its supply chain, the value of analysis becomes even greater as a way of refining innovative designs before building hardware. ■

Scoring an HVAC Goal for **Hockey** Spectators

CFD is used to design ventilation systems for sports arenas.

By Sami Lestinen, Tuomas Laine and Tom L Sundman
Olof Granlund Oy, Finland

In today's world, the design of auditoriums, stadiums and sports arenas is not complete without one or more simulations of the interior air flow. Analyses of this type are used to determine the best heating and air conditioning systems, analyze smoke removal in the event of a fire and ensure that the occupants are exposed to a predefined thermal comfort range most or all of the time.

Several year-round indoor ice hockey arenas were recently designed in Russia. CFD simulations of the HVAC systems were carried out by Olof Granlund Oy, Finland's leading building services consulting firm. Hodynka Arena in Moscow is the main venue for the International Ice Hockey Federation (IIHF) World Championships, to be played on April 27 to May 13, 2007. This 62,000 square-meter arena has a capacity for 12,000 spectators during the championship finals. The indoor air conditions in the arena are based on a displacement ventilation system, which is well-suited to large, fully occupied stands.
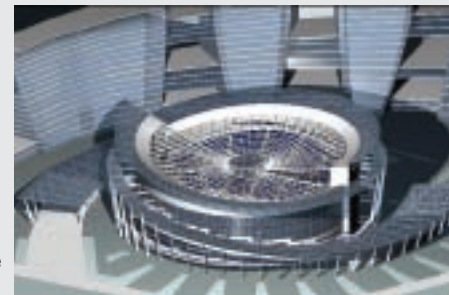
The Hodynka Arena in Moscow; CFD was used for designing the indoor air conditions.

The Tsherepovets Arena, on the other hand, was designed for 6,000 spectators and the indoor air conditions are controlled by a mixing ventilation system. Because both arenas also will be used for other events, such as concerts, CFD was used to better understand the interior conditions and flow behavior for a range of usage scenarios. The goal of these simulations was to determine how well the planned ventilation systems work to meet the desired indoor conditions.

Granlund uses CFD to research indoor air conditions in spaces where design requirements are high and detailed flow field information is important. Their typical focus is to compare a number of HVAC systems, air flow outlets, construction methods and other sources, all of which affect the indoor air quality of the finished structure.

For the ice hockey arena simulations, ANSYS ICEM CFD software was used to build the model, and ANSYS CFX tools were used to simulate and visualize the flow. The first round of simulations that is typically performed for large scale building projects such as these involves individual air supply devices to test and compare operating conditions. The results allow the design team to choose the appropriate devices for each specific location.

The device simulation results are compared to air jet theory and to the manufacturer's profile data and measurements. This step is important if the air flow behavior is to be estimated realistically in models of the building as a whole, in which the simulated supply air jet profiles can be used as boundary conditions. This technique requires fewer calculation nodes in the large model, which saves simulation time. The device models and simulation results are saved to an object library for future projects.

The goals of the arena simulations were to incorporate the parameters that most affect the flow field and ensure that target thermal conditions prevail in zones that are heavily occupied by people at any given time. Given these needs, the challenge was to design the supply air distribution so that fresh air flows to fully occupied zones and improves the thermal conditions in the arena as a whole. Draft, humidity and temperature levels during different types of events in winter and summer conditions were considered. Supply air jets, forced and natural convection, and heat sources and sinks cause very complicated 3-D flow fields, so the simulations needed to be performed with care. This need was constrained by the accuracy of the initial data, the approximations used, the level of convergence and restrictions on the allowable simulation time. The benefits of the simulations are that they provide the possibility to try out different air flow device types or supply air systems, such as mixing, displacement or a combination of both. First assumptions usually have to be corrected one or more times before the target is reached. In the end, however, a correctly performed CFD simulation is the only calculation method that can capture the indoor air flow field with the accuracy necessary for design purposes. ■
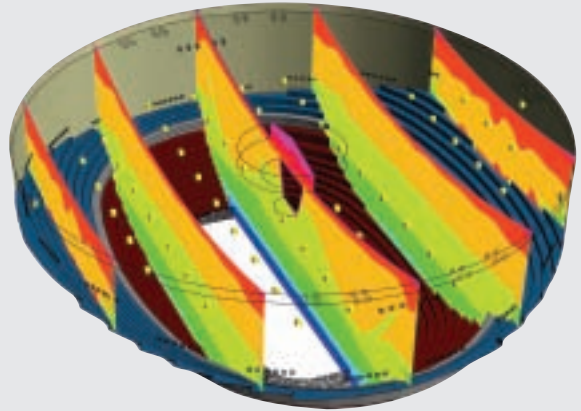
Temperature contours in the Hodynka Arena, showing stratification on several planar slices, during an ice hockey game in summer conditions with displacement ventilation
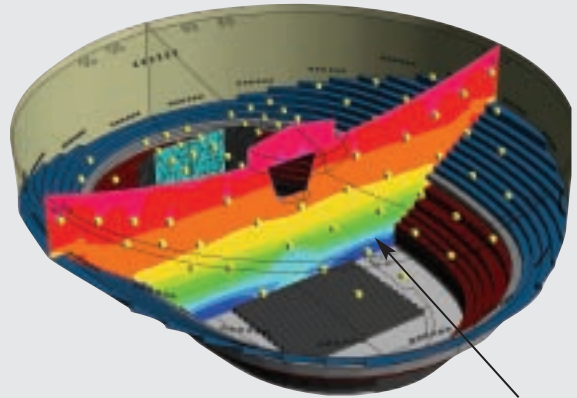
Temperature stratification during a concert event in summer conditions with displacement ventilation; the arrow points to the area where cool supply air flows to the occupied zone on the field.
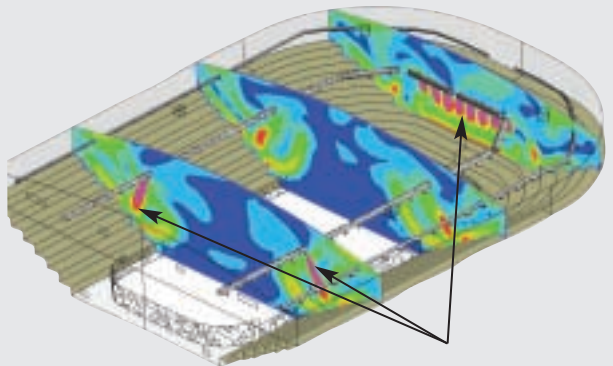
Velocity profile in the Tsherepovets Arena during an ice hockey game with mixing air flow from the roof zone; the arrows point to the areas where supply air jets flow to the occupied zones of the stands.

Taking a Bite out of Sports Injuries

Finite element analysis illustrates that both cushioning and support are needed to adequately protect teeth and surrounding tissue from impact injuries.

By Niall Paterson, Department of Materials Engineering, The Open University, U.K.

Simulation was used to rigorously study the cushioning and support provided by sports mouthguards in protecting teeth and surrounding tissue from damage.

Mouthguards have been used for dental protection in sports since the early 20th century, and a good mouthguard will significantly reduce soft and hard tissue damage. Despite the variation of available mouthguards, they are without question an effective and necessary piece of protective equipment in many sports. Their ability to protect the lips and gums from laceration by covering the incisive edges of teeth definitely warrants their use in contact sports.

The precise mechanisms by which the device provides such protection are still not well understood, however, and in particular there are 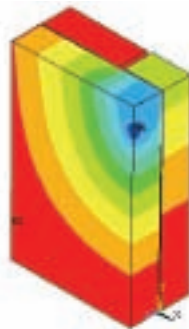no rigorous criteria by which their performance can be assessed or compared. Indeed, the degree to which they protect teeth and surrounding structure has not been thoroughly established due to a lack of meaningful data on key variables that affect their performance. To gain greater insight on the capacity of the mouthguard to absorb and spread the energy of impact, finite element analysis was used to evaluate the complex biomaterial requirements of the devices in relation to impact parameters such as peak force, loading time, and contact area.

Mouthguards provide protection by cushioning and supporting the teeth. Cushioning imposes a soft layer between the teeth and a hard colliding body, thus reducing contact stresses by spreading loads over a larger area and for a longer period of time. Lowering maximum stresses in this way reduces injuries, especially those characterized by brittle fracture and localized damage to soft tissue.

For support, the mouthguard typically is shaped to fit closely around the teeth. This design allows a concentrated load applied to the front surface to be shared by neighboring teeth.

The support depends on the rigidity of the guard and its ability to resist local deformation, which is in conflict with the requirements for good cushioning behavior.

Using a 2-D axisymmetric model, the cushioning effect of the soft layer of a mouthguard was analyzed with an explicit finite element code to predict the impact force given the geometry, material properties and impact velocity. This force was used in a 3-D simulation with ANSYS Structural software to determine tooth displacement. The results were scaled appropriately to study the supporting effect of the mouthguard.
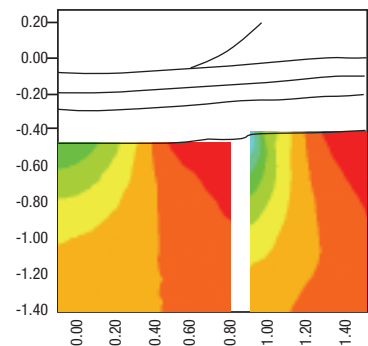
The cushioning effect of the soft layer of the mouthguard was analyzed with an explicit finite element code to predict impact force given the geometry, material properties and impact velocity.

The supporting effect of the mouthguard and biomaterials was studied with ANSYS Structural. Contact elements represented the low friction contact between the mouthguard and teeth.
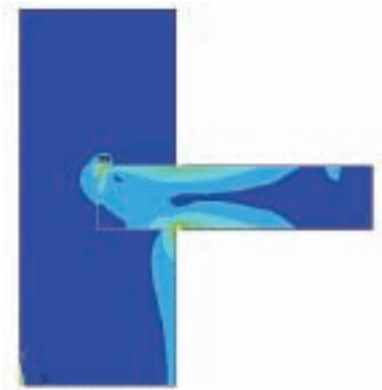
Cantilever beam models were used to determine stresses in teeth, socket and gum tissue resulting from impact.

The ANSYS model consisted of rectangular cantilevered beams representing the teeth plus a layer in front of the teeth representing the mouthguard with a static pressure distribution. Meshes were constructed with 3-D solid, tetrahedral and general hexahedral elements. Contact elements represented the low friction contact between the mouthguard and teeth. [1]

The analyses demonstrated that with a fixed load, the best overall support for the teeth is provided by highly stiff mouthguards. When the variation of impact load is taken into account, however, the cushioning effect of a soft coating outweighs the benefits of increased support provided by a stiffer layer.

The study thus provides significant insight into a modulus of elasticity and thickness for mouthguard materials that achieve optimal cushioning while not being so compliant that support is overly compromised. In this way, simulation improves the understanding of how mouthguards protect teeth and surrounding tissue so that better criteria for design, testing and standards may be developed.

Finite element analysis was indispensable in providing the level of data to rigorously study the protective performance of sports mouthguards, given the conflicting requirements of cushioning and support, the number of variables such as geometry and impact force. In the future it also may help to represent biomaterial behavior. ANSYS software was particularly well-suited to handling these complexities. Contact elements were especially valuable in representing the mouthguard, teeth and, although surrounding soft tissue behavior is too complex to model accurately at this stage, ANSYS allowed the substructures modeled to be combined into a single composite model as a flexible first approximation of the entire problem. The simulation provided insight into an important area of sports safety that heretofore was not as well understood or exhaustively assessed. ■

Reference

1. http://phoenix.open.ac.uk/~Rehana_Malik/Listing%20Folder/

Designing **Fitness** to Withstand the

Keeping bushing wear rates under control allows Life Fitness to maintain some of the highest equipment reliability standards in the fitness industry.

By Patrick Tibbits, Life Fitness, Illinois, U.S.A.

The fitness boom is evident at gyms around the world, as people of all ages take to the mats, weight machines, and cardio stations. Life Fitness is one of the world leaders in developing and manufacturing advanced fitness equipment for the home market, fitness facilities and training centers. Their commercial product lines include Life Fitness Cardio, Life Fitness Strength and Hammer Strength equipment on which professional and college athletes train. Life Fitness consumer cardio and strength equipment is aimed at home exercise programs. The extensive list of products from the company includes more than 50 U.S. patents and performance features that have led the way in the fitness industry.

Life Fitness products are intended to facilitate the muscular effort and exercise required to develop strength, speed, agility, range of motion and endurance in athletes and non-athletes everywhere. The equipment consists of electromechanical assemblies coupled with software control systems engineered for innovation, safety, reliability, quality and cost. In developing these products, engineers at Life Fitness use ANSYS Mechanical software to analyze parts and simulate the performance of fully assembled equipment to optimize designs, ensure safety and maintain some of the highest standards of quality in the fitness industry.

In cardiovascular and strength equipment, cylindrical bushings and plain bearings are used extensively to transmit high radial loads from a rotating shaft to a support structure. To meet reliability goals for the equipment, contact pressures between the bushing and shaft must be accurately determined to ensure that bushing wear rates are within acceptable limits.

Bushing catalogs often report wear rates as a function of average surface pressure, whereas Hertzian formulas generally are used to predict maximum pressure along a line of contact between the bushing and shaft. Neither of these methods accounts for axial misalignment between the shaft

Equipment
Workout

and bushing, however, which develops extremely high non-uniform pressure distributions that are difficult to determine from traditional methods.

Determining such contact pressure can be done with finite element analysis, but with many conventional codes the task is somewhat time-consuming because users must manually define which surfaces are touching and line up nodes of contacting part meshes. In contrast, ANSYS surface-to-surface contact element technology automatically detects regions where parts touch. Furthermore, higher-order elements are used that do not require nodes of contacting parts to line up.

In this way, ANSYS contact element technology readily handles solutions of such numerically difficult contact problems. TARGE170 and CONTA174 elements were used to simulate three-dimensional contact between the shaft and bushing, which get modeled with second-order solid elements with mid-side nodes. Extensive facilities for setting real constants and KEYOPTS for the contact elements greatly improve the detection of initial contact. The model of the bushing/shaft/housing assembly required no artificial constraints to prevent rigid-body motion.

Surface-to-surface contact element technology enabled Life Fitness engineers to accurately determine the contact pressure due to axial misalignment between the shaft and bushing. The maximum pressure predicted by ANSYS software exceeded the Hertzian line-contact pressure of 1,700 psi by 35 percent and was nearly 10 times greater than the average pressure of 250 psi computed by traditional formulas. Moreover, simulation showed how the distribution of non-uniform pressure varied from one point to another over the entire surface, providing valuable insight into potential bushing wear patterns and material behavior. In this respect, ANSYS technology is a microscope to examine pressure variations in the bushing/shaft contact.

Since bushing wear rate is proportional to contact pressure, integrating these ANSYS values for non-uniform pressure distribution in the shaft/bearing contact provides a more accurate calculation of wear rate. By clearly showing the effect of misalignment loading on the wear rate of the bushing, ANSYS analysis helped guide design decisions in the selection of bushing and shaft materials and surface finishes for a new model of fitness equipment. In this way, the predictive capabilities of advanced solutions such as ANSYS software play a critical role in enabling Life Fitness to develop some of the most innovative and reliable equipment in the industry. ∎

Bushings get a real workout in equipment such as this Life Fitness Cable Motion series of strength training machines. Twenty adjustments per column on the equipment's dual adjustable pulley system create a variety of exercises, and the mechanism allows for higher speed movements for sport-specific training.
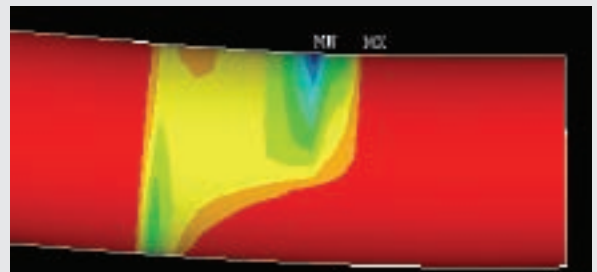
Non-uniform contact pressure distribution between the shaft and bushing enables engineers to accurately calculate bushing wear rates and thus guide decisions on the selection of component materials and surface finishes to maintain overall reliability of the fitness equipment.
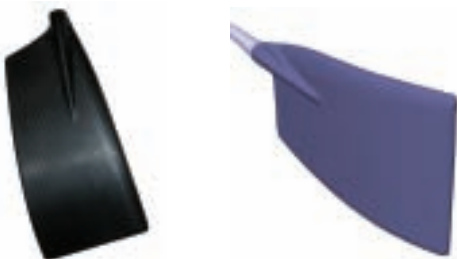
Catching a Better Oar Design

Engineers use CFD and a spreadsheet model to assess prospective oar blade designs.

By Jim Shaikh, Intelligent Fluid Solutions, U.K.

Pathlines in the wake of the oar show recirculating flow emanating from the inside edge and smoothly varying flow issuing from the outer edge.
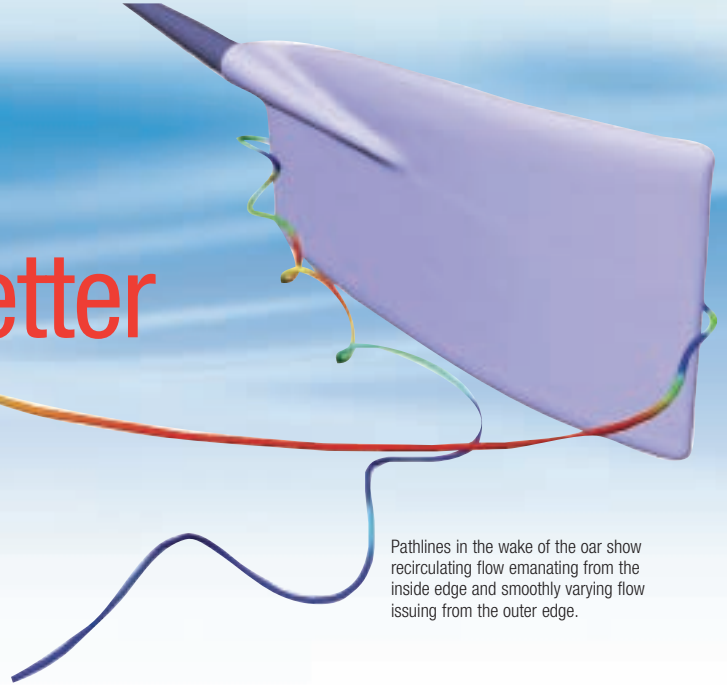
Rowing is a highly competitive international sport that is experiencing rapid growth in all parts of the world. As with most other sports, it has seen a number of changes in equipment during the past three decades, as composite materials have replaced wood and stainless steel for hulls, oars and riggers. While changes to the shape of hulls may elude the casual observer, changes to the shape of oars have not. The symmetric, tulip-shaped oar blades of the past have been replaced with a hatchet design that offers a straight edge as the oar enters the water. The increased surface area allows for a more effective transfer of power from the rower to the water.

A manufacturer of high performance rowing oars wished to develop a design tool to aid the development and evaluation of sophisticated geometries being considered for future oar designs. The primary objective was to overcome the cost and time constraints normally associated with the typical build-and-test development approach. The client approached engineers at Intelligent Fluid Solutions, a unique consulting service that specializes in fluids-based engineering design.

The design effort used a mixture of spreadsheet modeling and CFD analysis. Initially a spreadsheet model of the performance of a typical racing eight was developed. This tool was calibrated with race data and was used to link the ultimate performance of the eight to the characteristics of the oar. CFD analysis was used to examine the performance of the oar in detail. The approach used ANSYS ICEM CFD software to develop the computational model and

As expected, the pressure distribution on the front face of the oar shows the largest value at the outer edge, where the radius of rotation is maximum.

ANSYS CFX tools to obtain the flow solution. Using the k-ε turbulence model, the oar blade was assumed to be fully submersed in water, and its motion was simulated at three locations: the catch, when the oar first enters the water; the middle of the drive; and finally, the finish, just before the oar is removed from the water. The open-water boundaries were placed far from the blade to avoid edge effects. Using a rotating frame of reference, the pivot point was adjusted for each of the three positions to most accurately simulate the compound lever behavior of an oar as it propels a boat through the water.

The results of the CFD simulations were used to construct and calibrate an additional spreadsheet model of oar performance. The results also permitted visualization of the flows that develop around the oar in each position. This, along with reports for lift and drag, led to a deeper understanding of the complex oar system. The combined modeling approach greatly enhanced the client's appreciation of the mechanism of oar propulsion and offered a tool that could be used to quantify the benefits of alternative oar design concepts. ■

A photo (left) showing the geometry of an oar blade manufactured from composites. The CAD geometry of the blade (right) is used for the CFD analysis.

The Aprilia Shiver

No Shivers While Developing the Shiver

Tools within the ANSYS Workbench environment have allowed engineers to get a handle on crankshaft behavior before a motorcycle is built.

By Riccardo Testi, Piaggio & C. SpA, Italy

Motorcycle enthusiasts around the world are drawn to vehicles that deliver high style and top performance. More often than not, they are drawn to Piaggio Group, one of the biggest two-wheeler manufacturers in the world, with products ranging from mopeds to sport motorbikes and famous brands such as Vespa, Aprilia and Moto Guzzi. ANSYS products are widely used at Piaggio Group for the development of most critical vehicle components and subsystems. One of the most recent applications was part of the development of the 750 cc twin engine for the new Aprilia Shiver motorcycle, which is being brought to market in 2007.

The ANSYS Workbench environment was used to assess the fatigue resistance of the engine's crankshaft.

The component's geometry initially was imported from the CAD environment into ANSYS Workbench to generate a Modal Neutral File (MNF). The MNF was subsequently included in a multibody system (MBS), which was analyzed using Adams® motion simulation software. The MNF contains data that describe the dynamic flexibility of a component. Generation of the MNF was automated by means of an ANSYS Workbench Command Object, including all the ANSYS Parametric Design Language (APDL) commands needed to generate the interface elements and run the Adams macro itself.

Once the MNF file was generated, an Aprilia standard engine endurance bench test was simulated by assembling and solving the multibody model of the full powertrain. Dynamic effects due to the crankshaft's torsional flexibility were taken into account when importing the MNF into Adams. Other slender components' flexibility was modeled when generating the MNFs inside the ANSYS Workbench platform. The whole powertrain motion was simulated when subject to the real pressure cycles inside the cylinders' combustion chambers. The MBS provided the time histories of the forces and moments acting on the crankshaft and of the crankshaft's acceleration. The most severe dynamic equilibrium conditions were chosen and exported as environments to another ANSYS Workbench model, which was used to evaluate the stress state inside the crankshaft. The Fatigue Module of ANSYS Workbench was used to evaluate the safety factor based on infinite life, which was compared to the minimum allowable value according to Aprilia standards. The relative stress gradient effect was taken into account, by means of an ANSYS Workbench Command Object, used to evaluate the stress gradient at the most stressed point along the normal of the crankshaft's external surface. The APDL inside the Command Object singled out the most stressed point, built the normal to the surface and, finally, evaluated the relative stress gradient by means of path operations. Using these methods, the manual operations throughout the calculation process were drastically reduced so that different geometric configurations could be investigated with little effort. This allowed engineers to single out the best design while reducing the need for physical testing. ■

www.piaggio.com

The cycle of motorcycle design

The CAD model of the full engine

The ANSYS model generated by ANSYS Workbench to obtain the Modal Neutral File

The multibody model
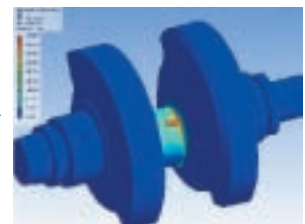
The stress field computed by ANSYS Workbench

Putting the Spin on
Air Pre-Cleaners

Dust and dirt particles are removed from the air intakes of off-highway vehicles using a novel air pre-cleaner.

By Gary Rocklitz
Donaldson Company, Inc.
Minnesota, U.S.A.

Off-highway vehicles spend much of their lives in dirty surroundings. This type of work environment poses a challenge for the engine and components, particularly the air pre-cleaner that separates dirt, sand and other large particles and debris from the air before it passes through the filter and into the engine. At Donaldson, an air pre-cleaner product line has been developed and optimized with the help of CFD. As a result of this effort, the TopSpin™ air pre-cleaner has been shown to extend air filter life and improve fuel economy for this special class of vehicles.

Air enters the TopSpin pre-cleaner chamber through stationary vanes that impart a tornado-like (free vortex) rotational velocity. This velocity generates centrifugal force that separates particle contaminants to the outside of the chamber. After the vanes, the air moving into the chamber passes through a conical section, which accelerates the flow. Accelerated air increases the particle separation efficiency and a larger chamber outlet decreases the system pressure drop. By the time the rotating air reaches the center of the pre-cleaner chamber, most of the particles from the incoming flow are spinning along the outer wall.

To remove the spinning particles, Donaldson engineers added a small number of paddles that sweep around the outer wall of the chamber. The inner vanes are designed so that the outer wall paddles rotate faster than the outer wall air flow. This causes each paddle to generate a positive pressure wave as it sweeps the outer wall of the pre-cleaner chamber. As the paddle assembly rotates, the particles are ejected by the positive pressure wave into the lower pressure atmosphere. In other words, dirty air is pushed out while the cleaned air moves inward and exits through the center of the pre-cleaner chamber.

Until very recently, the design of air pre-cleaners was much more of an art than a science. The only way to determine what dimensions were needed to optimize a design was to perform a long and expensive series of build and test cycles. Once a design was found, further testing was needed to see how the design scaled across a product line. The cost and time involved in building prototypes limited the number of prototypes that could be built and often forced engineers to settle for

The Donaldson TopSpin air pre-cleaner
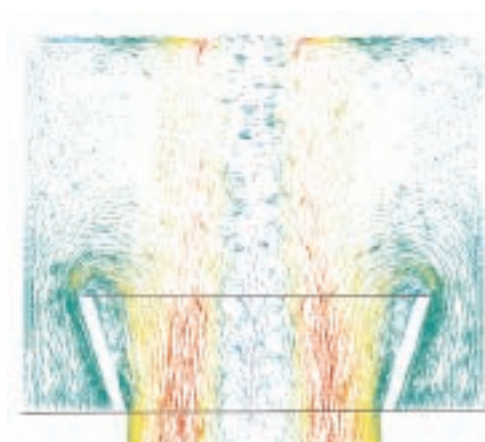
Velocity vectors on the mid-plane of the air pre-cleaner
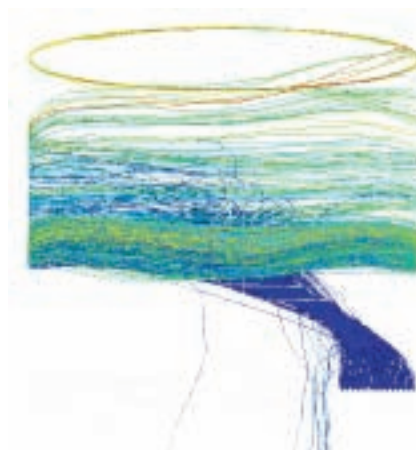
Particle trajectories in the air pre-cleaner, colored by residence time

good enough rather than optimized designs. The limitations in the physical measurements that could be obtained from an operating pre-cleaner meant that engineers gained minimal information from each build and test cycle, so many iterations usually were required to optimize the design.

Donaldson engineers have avoided these problems for the past several years by using CFD to evaluate new designs and optimize their performance prior to the prototype phase. CFD has allowed the engineers to quickly evaluate alternate configurations and get a much better understanding of why each one performs the way it does. For this project, simplified pre-cleaner designs were built using GAMBIT software. Unnecessary mechanical details were removed to save on computational time and focus on accurately depicting the passageways where fluids flow through the pre-cleaner. A rotating reference frame was used to model the rotating vanes and paddles.

Each of the 70 models was solved using four processors of a 32-processor Hewlett Packard Superdome computer. The paths of many differently sized particles were tracked and analyzed. Separation efficiencies were calculated for each particle size, and from the individual particle size efficiencies, an overall mass efficiency was calculated. The particle tracks also were used to graphically illustrate the path of each particle entering the pre-cleaner,

providing valuable information that engineers used to improve separation efficiency on future designs.

Donaldson engineers first optimized the pre-cleaner design without including the rotating elements. The goal at this stage was simply to move as many particles as possible to the walls of the pre-cleaner chamber while minimizing the pressure drop. The very first simulation provided far more information about the flow regime inside the pre-cleaner than could have been gained from a dozen physical tests. It showed the transition of the flow inside the pre-cleaner chamber from irrotational near the outside walls to rotational in the center. FLUENT software predicted the size of the "eye of the hurricane," information that was subsequently used to place the rotating vanes and paddles.

Some other key design parameters that were adjusted at this stage were the dimensions of the pre-cleaner chamber, the angle of the conic section used to accelerate the air as it enters the pre-cleaner and the dimensions of the vanes used to swirl the flow. Donaldson engineers made adjustments to the chamber geometry to provide the highest spin velocity for the lowest pressure drop.

The model dimensions of the rotating assembly and paddle angles were adjusted so that the paddles rotate faster than the swirling air inside the chamber. This created a pressure

wave high enough to remove the particles. They accomplished this by adjusting the number, size and shape of the paddles while also performing further fine tuning of the vane and conic section angles.

After each design change, they re-ran the simulation in order to measure the impact on the pre-cleaner performance and the flow regime. After completing their investigation of 70 configurations, Donaldson engineers were confident that they had optimized the design and had good knowledge of design trade-offs. Engineers also used the CFD results to calculate the forces acting on the paddles, which in turn provided loads for ANSYS finite element analysis software that was used to validate the mechanical design of the paddles.

The resulting TopSpin pre-cleaner is designed for use with many different types of equipment, ranging from crawlers and farm tractors to skid steer loaders. The TopSpin line is available in 13 models with outlets ranging from two to seven inches and air flow ranges from 100 to 1,600 cubic feet per minute. ISO 5011/SAE J726 tests showed that, as predicted by the simulation, the pre-cleaner separates 99 percent of contaminants 20 microns and larger. This greatly extends air filter service life. These performance benefits can largely be attributed to the use of CFD to optimize the design. ■

Man standing beside the pig iron channel, adjacent to where the hot metal flows

Blast Furnace Air Pre-Heater
Gets a Thermal Boost

Engineers use CFD to improve heat exchanger performance.

By Charles Assunção, V&M do Brazil; Professor Geraldo Augusto Campolina Franca, Federal University of Minas Gerais, Brazil; and Cesareo de La Rosa Siqueira, Martin Poulsen Kessler and Tales Adriano Ferreira, ESSS, Brazil

The blast furnace is perhaps the oldest, but still most widely used, technology for extracting iron from iron ore. Inside the brick-lined furnace, a combustion process takes place producing a raw form of iron (pig iron) that subsequently can be refined to more useful forms through other processes. Since the combustion of fuel can be made more efficient using pre-heated air in any combustion process, heat exchangers often are positioned before the blast furnace.

Vallourec & Mannesmann (V & M) Tubes, the largest manufacturer of seamless steel tubes in Brazil, together with engineers at Federal University of Minas Gerais (UFMG) and ESSS, a leading ANSYS software distributor in South America, used FLUENT software to analyze the air flow inside a heat exchanger used in one of V&M's blast furnaces. The main objective of the analysis was to verify the flow configuration in the current geometry and suggest possible improvements that would increase the exit temperature from the air pre-heater.

The heat exchanger consists of an array of tubes, each of which is a linear series of U-bends. The tubes are heated by the combustion gases in the furnace. In the steady-state CFD simulation, the standard k-ε model was used for the turbulent flow and the discrete ordinates (DO) model was used to account for radiation heat transfer. Combustion was not modeled. Instead, the combustion

gases were approximated by air of comparable temperature. The air inside the tubes was modeled as well.

The heat exchanger is large (15 x 4 x 2 m³), but the existing geometry is symmetric, so only half of the original geometry was analyzed using a tetrahedral mesh of about 6 million cells. To reduce the cell count, the tubes were modeled as zero-thickness walls and the shell conduction boundary condition was employed to account for the thermal mass of the tube material. Following the CFD simulation, the surface temperature predictions were used as input to ABAQUS™ to obtain the deformations and thermal stresses on the tube array that would arise from such a severe thermal load.

The analysis of the heat exchanger showed good agreement with experimental data. In particular, the results reproduced experimental observations that the tube-side air was not being heated properly. One reason for the problem was the tendency of the combustion gases to flow over the U-bends of the coils, resulting in limited surface area for heat transfer. To improve the combustion-side flow, modifications were suggested, such as the inclusion of baffles to promote cross-flow of the combustion gases through the coils. A new simulation was performed on the entire (non-symmetric) geometry of the newly proposed design using about 12.5 million cells. The results showed that the new configuration increased

the exit air temperature significantly because the baffles successfully modified the flow of the combustion gases through the coils, improving the heat transfer. The modifications have been approved by V & M Tubes and will become effective in the blast furnace design revamp in 2007. ◼
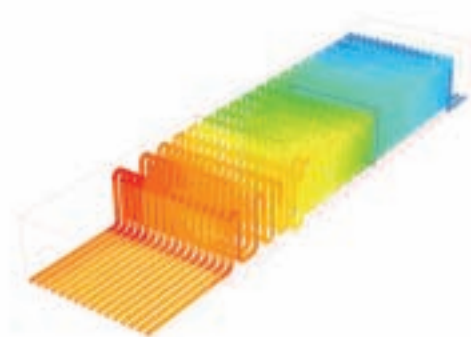
Temperature inside the heat exchanger tubes as the air flows toward the front left; the heated air is subsequently used for combustion in the blast furnace.
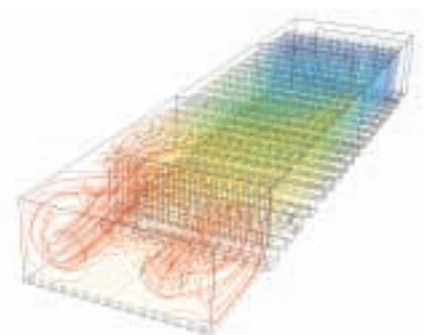
Pathlines, colored by temperature, show the improved flow of combustion gases through the heat exchanger tube array and the resultant transfer of heat to the tube array.

Fire Tests for
Molten Metal Converters

Numerical simulation helps engineers peer into a metallurgical converter in which high temperatures and adverse conditions make realistic measurements impossible to perform.

*By Hans-Jürgen Odenthal and Norbert Vogl, SMS Demag AG, Germany
and Mark Pelzer, ANSYS Fluent Germany*

A basic oxygen furnace converter in operation

In a steel mill, partially processed iron from the blast furnace is transported to a basic oxygen furnace, also known as a converter, to produce liquid steel. The converter is a refractory-lined steel vessel with a holding capacity of up to 400 tons of molten metal at temperatures above 1600 °C (2900 °F). In the converter, several jets of oxygen are blown onto the melt surface, and the subsequent oxidation process helps remove undesired secondary elements such as carbon, manganese, silicon, phosphorus and sulphur. Efficient mixing of the melts is helped by additionally introducing inert gases, such as nitrogen or argon, from the floor of the vessel, where they bubble to the surface. Optimization of the converting process depends on a number of variables, but operating trials and parameter studies on water models cannot be carried out realistically using experimental methods. Steel producers as well as steel equipment manufacturers therefore are switching to numerical simulations of the process to optimize the quality of the final product while minimizing the time and cost of steel production.

SMS Demag AG, Düsseldorf, is a leading installation builder for the steel and non-ferrous metal industries. Besides making individual components, SMS Demag plans and builds complete production lines and turnkey plants. It is a part of the SMS Group GmbH, which has interests in metallurgy, forging, casting and rolling technology, engineering services, and plastics engineering. At SMS Demag, a group of experts has been successfully working in the 100-employee Central Development area, studying the mutual relationships of individual process parameters with FLUENT software and using them in industrial applications. The bandwidth stretches from long-term development projects to project-related orders and fault analysis on operating plants. Using CFD, the flow and thermal mechanisms taking place in the melt can be analyzed in detail and displayed, thus considerably facilitating the understanding of the process. Three-dimensional projections have been used to augment spatial visualization. The Cave Automatic Virtual Environment (CAVE) at RWTH Aachen University in Germany is used for particularly important projects, such as the demonstration of complete virtual plants.
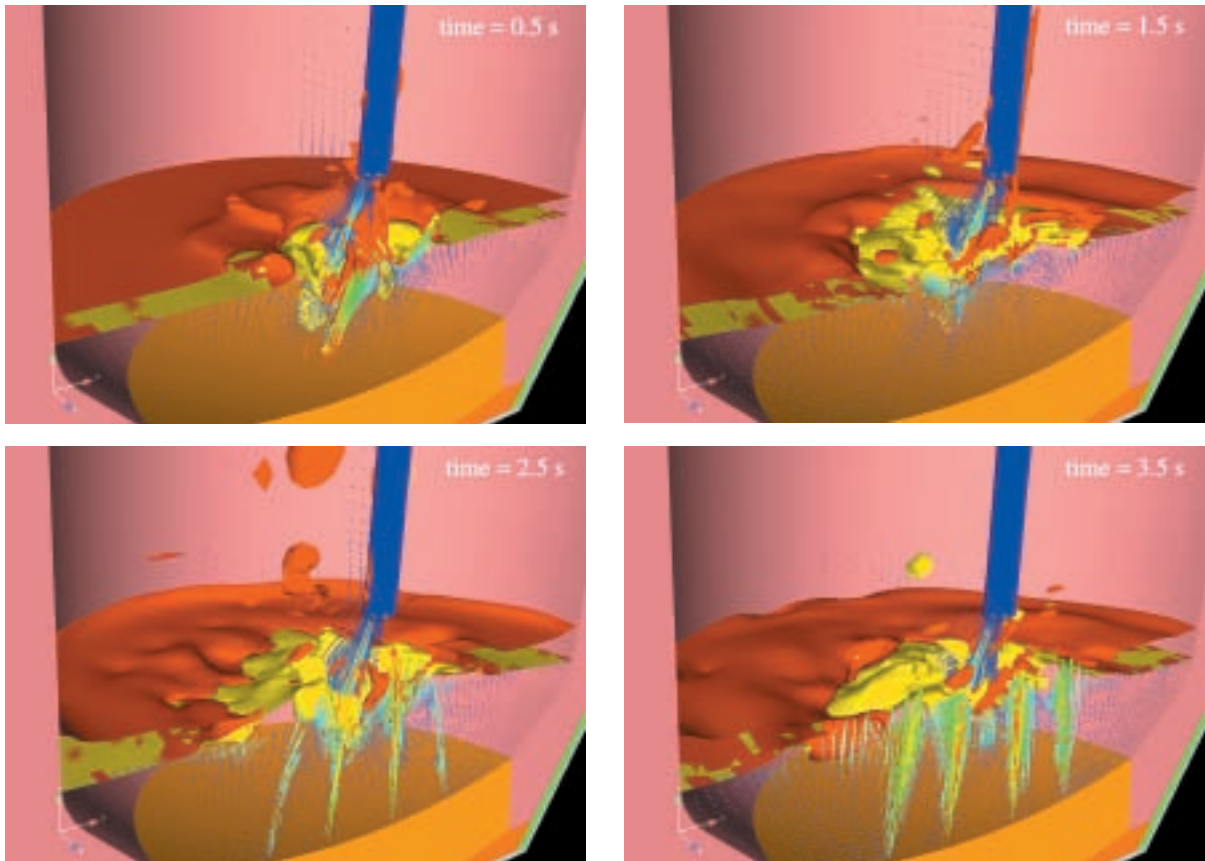
Instantaneous images of the molten metal (yellow) and slag (red) free surface after 0.5, 1.5, 2.5, and 3.5 seconds of operation, showing the penetrating oxygen jets and resulting flow field

One of the primary objectives of the CFD work has centered on the blowing of oxygen into the converter and the subsequent phenomena induced by this process. The oxygen is delivered into the converter by a top lance, which terminates in a fitting that contains several Laval nozzles. Each nozzle produces a jet at approximately twice the speed of sound. The jets penetrate deeply into the melt and create oscillation blowing cavities with large reaction surfaces. The top lance is designed to avoid undesirable effects, such as metal back-splashing, which leads to increased wear. The inert gas conduit on the floor is designed to avoid plugging so that sufficient gas can continually be introduced to the melt to achieve the desired agitation.

The process being simulated is a hot, highly turbulent, ever-changing multiphase flow. The model generation begins with ANSYS ICEM CFD Hexa for the creation of a 500,000-cell hexahedral mesh with aligned cell layers. In addition to making use of models for turbulence and heat transfer, the numerical simulation couples together two multiphase models in FLUENT software:

The geometry of a converter, showing the refractory lining and oxygen top lance, terminated with an array of six Laval nozzles. The steel melt is agitated by argon bubbles released from several locations on the converter floor. During the conversion process, a slag layer forms on the melt surface.

the volume of fluid (VOF) model for tracking the evolution of the free surface of the melt and slag, and the discrete phase model (DPM) for tracking the trajectories of the inert gas bubbles used for mixing. A Linux® cluster of up to 10 computers was deployed for two weeks to perform a 20 second-long simulation of the 20 minute-long blowing process. The high computation times are due, for the most part, to the complexity of the physical models rather than the size of the mesh. User-defined functions (UDFs) were employed to couple the models together and incorporate effects such as a varying drag coefficient for the bubbles.

The results provided information about the design of the nozzles, the penetration of the oxygen jets into the melt, the effectiveness of the agitation scheme and heat transfer into the refractory walls. Despite the comparatively short span of the simulation, the results to date have been sufficient to make clear decisions about the basic sequences in the melt and to introduce optimization measures. Thus, each converter can be adapted to the individual requirements of the customer. ■

Neutrino Detection
in Antarctica

Simulation helps speed up drilling through ice so that optic monitors can be installed.

The IceCube Laboratory at sunrise
Image courtesy Katherine Rawlins/NSF (IceCube).

By Dick Martin and Tim Thompson, HYTEC Inc., New Mexico, U.S.A. and Naseem Ansari and Chokri Guetari, ANSYS, Inc.
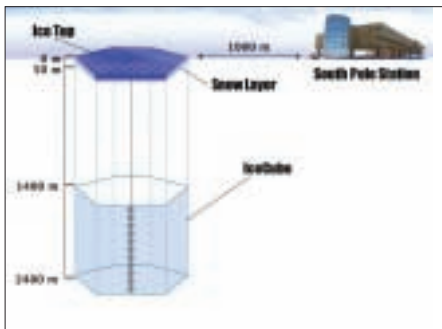
A diagram of the IceCube Neutrino Observatory

A hose going down into an IceCube hole drilled with the hot-water nozzle

Images courtesy NSF (IceCube).

Neutrinos are elusive particles. Nearly massless, they travel cosmological distances at nearly the speed of light, passing through most material without making a mark. Neutrinos are produced in nuclear reactions in the sun or in dying stars; by studying them, scientists can learn about extra-galactic events such as supernova explosions, gamma-ray bursts and black holes. Polar ice, which is exceptionally pure, transparent and free of radiation, has emerged as an ideal medium for detecting elementary particles like neutrinos. Rather than capture neutrinos directly, evidence of collisions between the particles and ice can provide important information about the particles and their history.

To take advantage of the special properties of Antarctic ice, the IceCube Neutrino Observatory is currently under construction at the South Pole. The observatory, designed by the University of Wisconsin and funded by the National Science Foundation in the United States, will gather information from cosmic neutrinos that originated in outer space above the earth's northern hemisphere. The earth itself will be used as a filter to exclude atmospheric neutrinos, and high-energy cosmic neutrinos will pass through the earth's layers, on to the ice cap of the South Pole, and upward to the IceCube detectors.

The neutrino observatory will consist of a matrix of digital optic monitors (DOMs) buried in the ice. To create this matrix, 4,800 DOMS are being suspended on vertical strings in 80 holes that have to be individually drilled into the ice. Each hole is 2,400 meters deep. The holes are created with a hot water drill consisting of a nozzle and 2,400 meters of

hose connected to a water heating plant on the surface. The descending hose melts through the ice as 90°C water is pumped through the nozzle at a given spray angle. The water cools to about 1 to 2°C as the ice melts to create the hole and this water is pumped out the top of the hole. After the string of 60 DOMs is lowered in the hole, the water freezes around them, creating a permanent installation.

The drilling phase of the process occurs as the drill melts through the ice. During the ream phase, the drill is raised with the hot water continuing to flow, enlarging the hole and providing more time for the heat to conduct into the ice. By warming the ice surrounding the hole, it takes longer for the hole to close. This is critical, as there must be adequate time after producing the hole to insert and lower the optical module string before the hole gets too small.

HYTEC Inc. took on the challenge of performing a CFD study to examine the complex drilling process. HYTEC's engineering teams conceive, design, build and test ingenious solutions to unique challenges. They meet customers' technical, cost and quality expectations in a wide range of high technology projects. For the IceCube project, they focused on the water–ice melting process at the bottom of the drill hole in hopes of better understanding the influence of nozzle size, spray angle, water flow rate and water temperature on hole shape in order to increase the drilling speed. Because
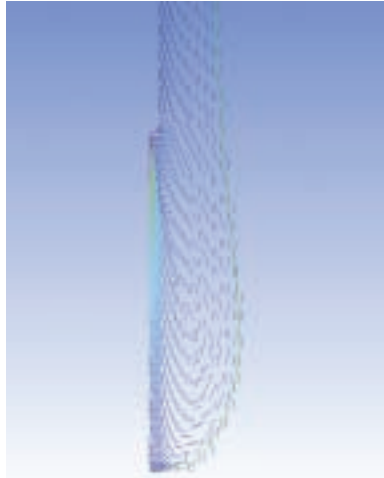
Velocity vectors showing the hot water jet and the flow of melted water surrounded by ice
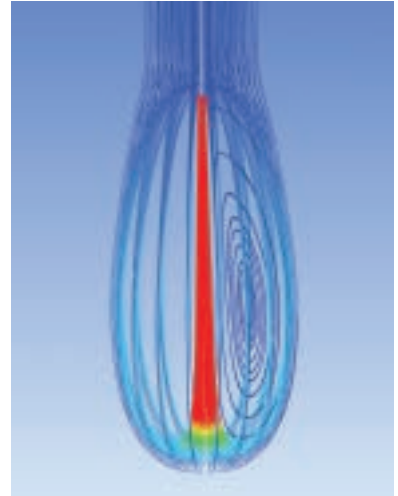
Pathlines illustrate the flow of water emanating from the nozzle, stagnating at the bottom of the hole, flowing upward, and being partially re-entrained into the jet.

the drilling season is limited (from December through February) and the cost to fuel a water-heating plant at the South Pole is extremely high, even a 10 percent increase in drilling speed could save the project large amounts of time and money.

ANSYS CFX software was applied to this problem because it has the ability to solve turbulent multiphase flow with conjugate heat transfer in an axisymmetric domain. Multiphase sensitivity studies determined that the heat transfer coefficients on either side of the water-ice interface were the most influential parameters in obtaining results that compare well with site data. A series of steady-state computations were performed for two nozzle diameters, two drill speeds and one spray angle. The runs were modeled at 1,292 meters, to allow comparison with site data.

By reducing the nozzle diameter from 1 in to 0.75 inches at the same flow rate, the hole depth was increased by 16 to 20 percent. The ANSYS CFX

results showed that using a spray angle of 25° made little difference in hole depth or diameter. Increasing the drill speed at a fixed nozzle diameter caused the drill hole depth and hole diameter to decrease. Thus, an Ice-Cube drill model was developed to evaluate the performance of nozzle designs and to specify drill speed versus depth.

During the 2004-2005 South Pole drilling season, drilling speed was between 3.5 and 4.0 ft/min. During the 2005-2006 season, a smaller diameter nozzle was used based on the ANSYS CFX analysis. The drilling speed almost doubled to 6.9 ft/min, which will save the project over 1 million dollars. ■

Suggested Reading

Martin, Richard A.; Thompson, Tim; Ansari, Naseem; Guetari, Chokri: IceCube CFD Drilling Model. Proceedings of 2006 ASME Joint U.S.-European Fluids Engineering Summer Meeting: FEDSM2006-98042. Miami, FL, July 17-20, 2006.

Hosing transports hot water to the drill tower.

Making Sure Wood Gets **Heat Treated** with Respect

The ANSYS Parametric Design Language helps establish the thermal conductivity of wood and composites to enable more effective heat treatment processes.

By John Krouse, ANSYS Advantage

Established in 1910 by the U.S. Department of Agriculture Forests Service, the Forest Products Laboratory serves the public as the nation's leading wood research institute and is internationally recognized as an unbiased technical authority on wood science and utilization. The lab is the public side of the public-private partnership needed to create technology for the long-term sustainability of forests. According to a recent study, tax dollar investment in forest products research generates as much as a 300 percent return to society.

One particular area of research at the lab is focused on more precisely establishing the thermal conductivity of wood. This property is of critical importance in the drying process since too little heat treatment results in retained moisture and long drying times while too much treatment may damage the structure of the wood. Conventional models developed more than 50 years ago serve as a rough guideline for calculating an average thermal conductivity for certain types of wood. However, because of the inherent inaccuracies of this approach, lumber mills and wood processing companies must perform trial-and-error tests to determine proper temperatures and drying times — a costly, time-consuming process that often results in high scrap rates.
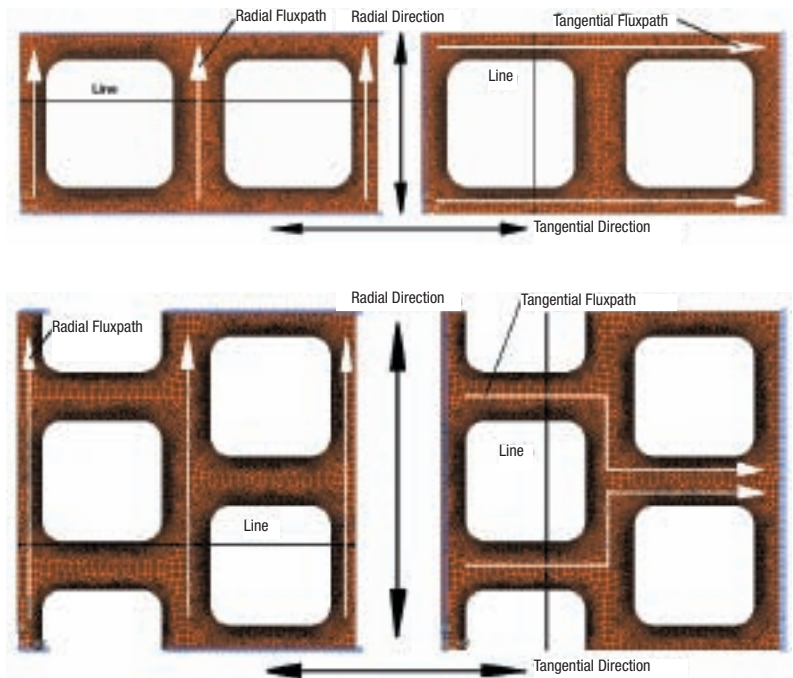
Micro-level analysis models represent different wood cell orientations, with softwood cells aligning in straight rows (top) and hardwoods having cells offset by perhaps 50 percent (bottom).

The heat transfer coefficients of wood depend on many variables, including ring density, the age of the tree, initial moisture content and orientation of the cells. Porosity can range from 70 to 90 percent in earlywood and from 10 to 30 percent in latewood growth. Moreover, in softwood, the cells tend to align in straight rows in the radial direction and are offset in the tangential direction. Compounding the difficulty, these characteristics usually are not uniform across all sections of the same tree, since wood structure
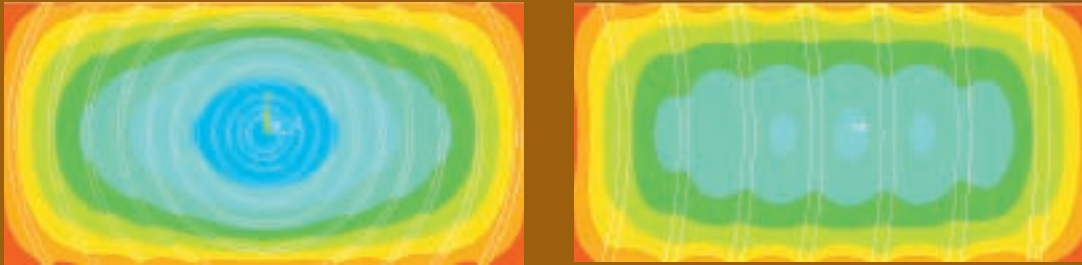
Contour plots indicate the temperature distribution at two locations with different ring structures at the same point in time in the same wood board.

can be affected by seasonal weather differences. Also, the ring density (and thus heat transfer) in small versus large-diameter trees varies widely depending on the growth rates for different conditions, which are governed by the surrounding vegetation and climate fluctuations.

On the micro (cellular) level, models programmed using ANSYS Parametric Design Language (APDL) were developed to simulate the structural variation of cell porosity and alignment in determining the effective heat transfer coefficients. For the macro level, boards cut from seven different locations in a typical log were modeled to examine the effects of wood structure on the transient heat transfer process using the thermal conductivity values obtained from the micro-level analyses. Dozens of simulations were required to determine the heat transfer rates for a wide range of wood geometries and structural

conditions. APDL performed repetitive analyses in which new values of various parameters could be entered and analyzed without manually rebuilding multiple simulation models.

The FEA simulations showed the significant effects of cell alignment, cell density, ring width and ring orientation on the thermal conductivity of wood. The study concluded that the porosity of wood as well as the growth rate of the tree play major roles in determining the effective thermal conductivities, which can be fitted to equations or stored in a look-up table for further use in macro wood models. The results also showed that heat transfer in a piece of wood is affected significantly by ring density and orientation, and quarter-sawn boards have higher heat transfer rates than flat-sawn boards due to the shorter pathway through latewood cells. This insight will enable more effective wood heat treating processes and help better utilize this

critical natural resource.

According to John Hunt from the USDA Forest Products Laboratory, "The finite element models allowed engineers to examine the fundamental thermal conductivity differences for radial and tangential heat transfer at the micro level, and also to estimate the transient heat transfer effects at the macro level in a board of wood. A parametric method enabled numerous simulations to be run efficiently, providing a broad range of data needed to determine the heat transfer coefficients. Such an approach in studying heat transfer issues in wood has many practical applications that include optimizing drying schedules for different board cuts, determining heat treatment times to kill insects and determining heat curing times for solid wood as well as composites. In this respect, FEA software is a valuable tool to explore and review the fundamental laws of science." ■
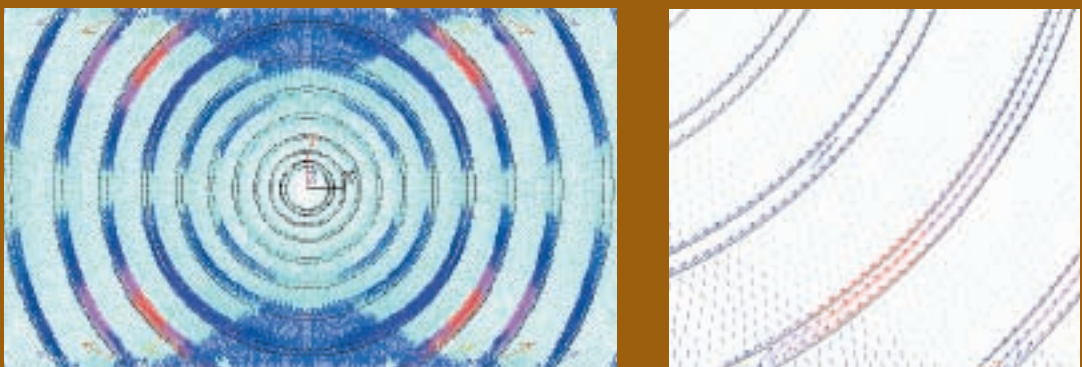
Vector plots of heat flux showed high and low thermal transfer in earlywood and latewood due to different cell densities in these regions. Heat flux vectors (left) are enlarged to show the detail near the rings (right).

Accelerating
Product Development
in a Global Enterprise

With the goal of compressing cycle times by up to 50 percent, the Velocity Product Development (VPD™) initiative at Honeywell Aerospace uses engineering simulation to eliminate delays while lowering cost and maintaining high quality standards for innovative designs.

Lee Williams

By Lee Williams, Vice President (ret.)
Defense and Space Business
Honeywell Aerospace, Arizona, U.S.A.

Honeywell Aerospace is a leading global provider of integrated avionics, engines, systems and service solutions for aircraft manufacturers, airlines, business and general aviation, military, space and airport operations. A strategic business unit of parent firm Honeywell International, the company employs 40,000 people in 97 manufacturing plants, design centers and service sites around the world.

Despite initiatives such as Design for Six Sigma (DFSS), typical problems experienced by a global organization of this type in developing complex products include lengthy design cycles, the inability to predict product performance early in design, missing product release schedules, cost overruns, slowness to adapt to market change and inefficiencies in utilizing design centers around the world. Competitive pressures demand that end-to-end cycle times be compressed while lowering costs and maintaining high quality standards for innovative product designs.

In July 2005, Honeywell Aerospace reorganized to address these challenges. At the foundation of this change was Velocity Product Development (VPD™), a process for making dramatic improvements in cycle times and success rates for new products. With a lean focus on engineering and development, VPD is based on DFSS and works synergistically with the company's advanced manufacturing and marketing groups to ensure a streamlined and quality-focused product development process.

Engineering simulation plays a critical role in enabling Honeywell Aerospace to reach the ambitious goal of VPD to reduce cycle times up to 50 percent. Through the use of analysis tools from ANSYS, engineers address several areas that often cause delays and other problems in the product introduction cycle. Simulation allows product performance and risk to be assessed and mitigated early in development so that troubleshooting and correcting problems with hardware prototypes can be avoided. Problems are fixed up front rather than later in the design cycle, so re-work is reduced. Simulation-driven development guides the design and enables engineers to create and assess innovative ideas
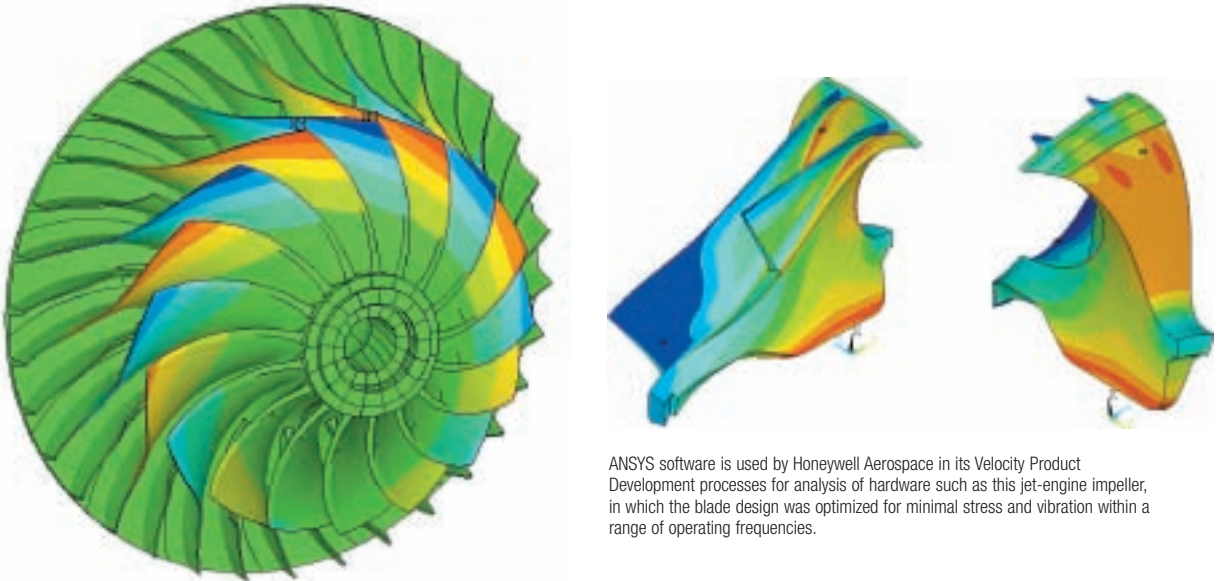
ANSYS software is used by Honeywell Aerospace in its Velocity Product Development processes for analysis of hardware such as this jet-engine impeller, in which the blade design was optimized for minimal stress and vibration within a range of operating frequencies.

that might not otherwise have emerged by studying various alternatives and optimizing the best ones.

At the inception of VPD, the Toyota Product Development System was studied to help formulate the processes to be used in Honeywell's approach. While the Toyota system is not a drop-in solution for aerospace product design, there are applicable methods and disciplines. In particular, their Set-Based Concurrent Engineering (SBCE) method and culture of disciplined Knowledge Management (KM) were identified for adoption by Honeywell.

For example, a chief engineer or systems design lead submits a target specification to subsystem design teams. The specification is highly aligned to the "Critical to Quality" customer requirements of the design offering. Initially, each subsystem team defines a design space of alternative approaches to achieve a robust system design offering. The subsystem teams then meet to present their design alternatives and collectively perform successive system-level experiments designed to find the optimal solution. These specifications are then successively cascaded down to the design of individual components. Simulation is essential in executing this process efficiently. Analysis at the system level is necessary to determine loads on subsystems and assemblies and com-

ponent parts — with all designs studied carefully with computer models.

Within Honeywell's VPD initiative, ANSYS technology is particularly valuable for stress, vibration and heat-transfer analyses of rotating hardware such as fans and compressors, as well as static equipment such as engine frames and cases. Linear and non-linear analysis is done, often taking into account creep, plasticity and contact forces. Optimization and design of experiment (DOE) studies are done to evaluate and refine designs. The models carry a fair amount of detail and make use of fine meshes to accurately represent the complex contours of aerospace components.

In one study to predict vibration in mistuned jet-engine turbines, a parametric finite element model of an impeller blade was created with a specified range of variables defining its shape. ANSYS Probabilistic Design System (PDS) then automatically went through numerous iterations in which modal deformation and stress were determined for various blade geometries. A transfer function based on the PDS database of the results helped define the regions of the airfoil in which geometric variations have the greatest impact on vibration. This information then was used to optimize the design of the blade for minimal stress and vibration within a range of operating

frequencies. In this way, the best design of the blade for a particular application could be determined. The use of simulation to achieve this goal saved months of time and tens of thousands of dollars in hardware testing.

As Honeywell Aerospace moves forward with VPD, the focus will be on the following:

- Establishing one set of development processes to deploy globally
- Using common design and simulation tools
- Assuring technology readiness before starting detailed design
- Improving the development estimating processes and disciplines
- Becoming obsessive about "Design to Unit Cost"
- Significantly increasing "Design Re-use"

At Honeywell, VPD is a key part of the "Lean Enterprise" in which value from the customer's perspective drives the process, waste is eliminated, and the workforce is involved in an atmosphere of continuous improvement. Engineering simulation is a key element enabling the company to meet these objectives and strengthen its competitive position in the aerospace market. ■
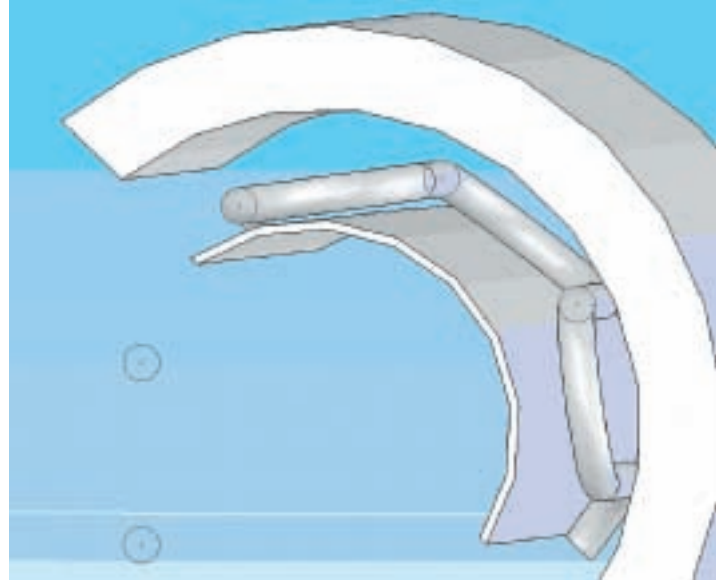
Students at the University of Pittsburgh used the ANSYS Workbench platform in modeling the complex weave geometry of an intravascular stent (left) as well as the deployment procedure (right) in which a balloon expands the device against the artery wall.

Stent Analyses
Expand Students' Exposure
to Biomedical Engineering

Engineering students gain insight into the physics of medical devices and add to the body of knowledge on stenting procedures.

By Dr. Michael Lovell, Associate Dean for Research and Director, Swanson Institute for Technical Excellence
University of Pittsburgh, Pennsylvania, U.S.A.

Professor Michael Lovell directed the student research on intravascular stents

With medical costs skyrocketing and health care systems overburdened by an aging population, experts acknowledge that the way diseases are treated must be changed radically. Undoubtedly, heavy investments must be made into addressing largely preventable diseases such as diabetes, heart disease and lung cancer. Furthermore, approaches must be established to detect diseases earlier in their course and regenerate functions as much as possible in patients rather than solely ameliorate symptoms.

One significant barrier for the medical community in this critical cycle of effective prevention, early diagnosis and functional regeneration has been a lack of sufficient analysis capability and computer processing power for real-time simulations of these procedures, which typically include complex mechanical structures and biomaterials. Recently, however, advanced analysis technologies and powerful computational resources have converged, enabling engineers and scientists to simulate device performance and regenerative medical procedures in ways previously thought unimaginable.
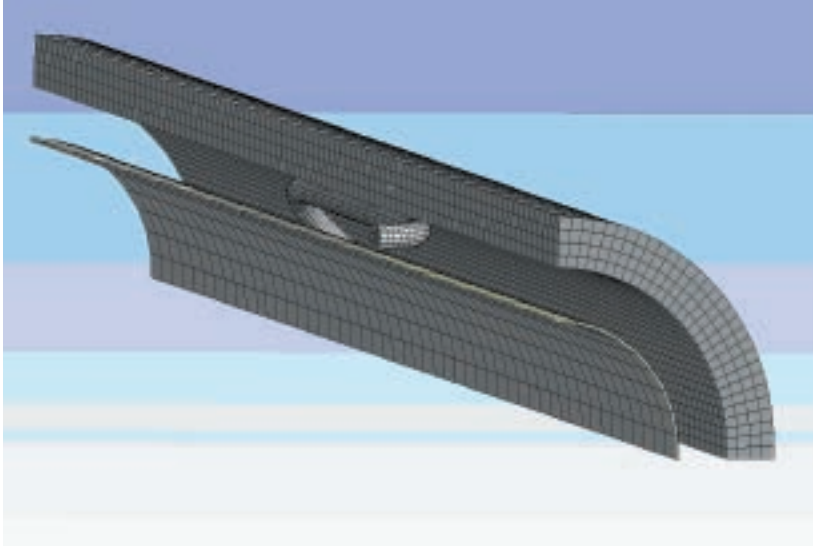
The detailed ANSYS model includes elastin and collagen fibers for the artery, a hyperelastic model for the balloon and an isotropic model for the stent.

The ability to use commercial analysis packages such as ANSYS software has allowed the scientific community to treat diseases in radical new ways using a broad range of multidisciplinary approaches. By having the capability of simulating complex medical treatments that include phenomena such as dynamic contact, nonlinear materials and fluid structure interaction, the bioengineering community has been able to bring products and technology from the bench to the bedside in a fraction of time that it did a few years ago.

At the University of Pittsburgh in the United States, these critical issues are being integrated into engineering curricula. In one class research project, students utilized the ANSYS Workbench suite of design and structural analysis software to simulate the deployment of intravascular stents: devices inserted into blocked arteries and radially expanded with a small balloon to open the artery and enhance blood flow. Through these simulations, the design of the device was optimized to maintain proper blood flow while preventing elastic recoil of the artery. ANSYS Workbench also provided a better understanding of the stresses developed between the stent and artery so that mechanical injury to the artery can be reduced. Such insight is significant in minimizing the risk of neointimal hyperplasia, an inflammation and subsequent build-up of excess cells that eventually result in restonosis, the narrowing and eventual closing of the artery.

Based on solid models of the device at various stages of the deployment, finite element models of the stent — as well as the deployment balloon and arterial wall — were effectively developed, with the quality of the mesh produced by the ANSYS Workbench platform having a dramatic impact on the solution time and accuracy of the simulations. Utilizing an advanced material model that includes elastin and collagen fibers for the artery, a hyperelastic model for the balloon and an isotropic model for the stent, student researchers were able to obtain detailed stress distributions in the artery wall when the balloon was expanded into the stent. Such information is critical to optimizing the design of the stent and the stenting procedure, which will ultimately eliminate the occurrence of restonosis.

Projects such as this add significantly to the body of information in the medical community on stenting procedures while having a tremendous education value for engineering students. Simulation provides insight into the physics of stents, better understanding of the behavior of artery walls so potential problems can be spotted quickly and alternative designs readily evaluated. Moreover, while performing this work, students learn firsthand how to use advanced analysis tools such as ANSYS software that are widely used throughout industry. ■
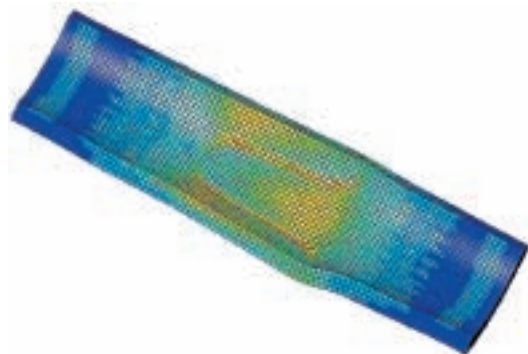
ANSYS simulation provided detailed stress distributions in the artery wall when the balloon expanded the intravascular stent.

Designing a Course for
Future Designers

Students use Volvo concept car to learn about simulation tools.

By Professor Lennart Löfdahl
Chalmers University, Sweden
and Robert Nyiredy, ANSYS, Inc.

In training the next generation of design engineers, opportunities for learning to use simulation tools are rapidly expanding. Indeed, the increasing use of commercial CFD software in industry is spilling over into undergraduate programs at universities around the world. Among the success stories beginning to emerge involves a group of students from Chalmers University of Technology in Gothenburg, Sweden and a unique course in advanced road vehicle aero-dynamics, developed during the Spring 2006 term. The course included sections on CAD cleanup, surface and volume meshing, solving, and post-processing, making use of existing tutorials on all of the software tools to be used. The students were advised not to focus on achieving perfect results but to gain an understanding of the working methods used by car designers. By extension, they needed to become familiar with the mesh generation and


The team of students at Chalmers University who worked on the Volvo concept car; Professor Lennart Löfdahl is at far right.

flow solution tools used in the automotive industry and then learn to compare their results with experimental data.

At the start of the course, the Volvo Car Corporation submitted a model of an old concept car — known as the Volvo Research into Automotive Knowledge (VRAK) — and also made engineering staff available for support during the project. Using tools such as ANSA™, TGrid and FLUENT software, the students were tasked

with examining the external air flow around the VRAK vehicle in a simulated 50 meter wind tunnel. During the 11-week course, the group considered two versions of the VRAK vehicle — one with a rear spoiler and one without. Given the short duration of the course however, there was time only to fully mesh the case without the rear spoiler, and thus a finished mesh of the spoiler case was provided by the Volvo staff.

After reading the CAD geometry from Volvo into the ANSA 12 pre-processor and exploiting a plane of symmetry, the domain was separated into the car zone and the fluid zone. The fluid zone was further divided into an inner box around the car and outer box for the far field. The students then created a coarse surface mesh to ensure that the volume was closed. Refinements were made in several locations, including around the wheels and in the area in which the wake was expected to form. The final surface mesh contained about 275,000 triangular elements, and was imported into TGrid software to create the volume mesh. Using TGrid 4, the students grew
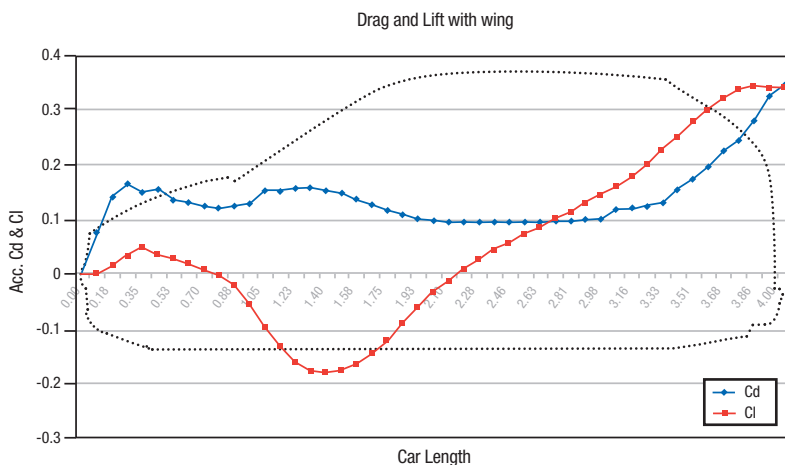
Predictions of the accumulated lift and drag coefficients for the vehicle model with a spoiler; calculations of C_L and C_D performed on a car without a spoiler were found to be higher.
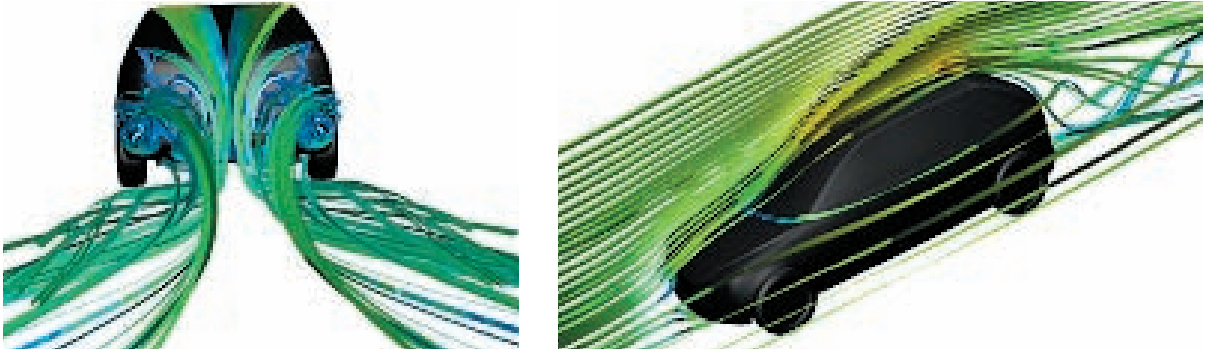
Pathlines colored by velocity magnitude were used to visualize the flow features in the wake of the car, including a pair of trailing vortices (left) and flow separation (right).

prisms from the surfaces of the vehicle to get good resolution of the boundary layer zones and then generated a tetrahedral mesh for the fluid region containing about 3.4 million cells.

The group next loaded the finished mesh into FLUENT 6.3 and simulated the viscous effects by using the realizable k-ε model. They chose this model because it limits the prediction of normal stresses, which they felt was appropriate given the heavily bent streamlines that were expected. A constant velocity inlet boundary condition of 140 km/h (87 mph) was used. To follow Volvo's wind tunnel experiments as closely as possible, both the road and the wheels were stationary in the simulation, and boundary layer suction was not considered due to time considerations.

Results of the simulations showed that the addition of a rear spoiler caused the values for the drag and lift coefficients to decrease in the wake region behind the vehicle by 4 percent and 7 percent, respectively. With or without a spoiler however, both coefficient values increased at the back end of the vehicle relative to the front. The drag increase was due to the same rear wake region. The increase in lift was due to the higher velocities on the top side of the vehicle, which had the consequence that the pressure on top was smaller than on the underside.

Also of interest to the group were oil flow visualizations, with which the students could investigate the flow patterns over the rear body in the region of the wake. These visualizations helped the group find where the flow separation occurred. With the

addition of the rear spoiler, the flow separation occurred farther behind the vehicle. Physically this meant that the drag force was reduced at that location, which agreed with the calculated values of the coefficients. Velocity pathlines served to further illustrate the point of separation.

With the analysis completed, the project results were presented to an audience that included representatives from the Chalmers faculty, Volvo and ANSYS. With little or no experience with commercial CFD tools prior to the course, the general impression was that the group had done good work. Although the students are on different career tracks, it is likely that all of them will continue to work with CFD in the future and use the software that helped them succeed in this memorable exercise. ■
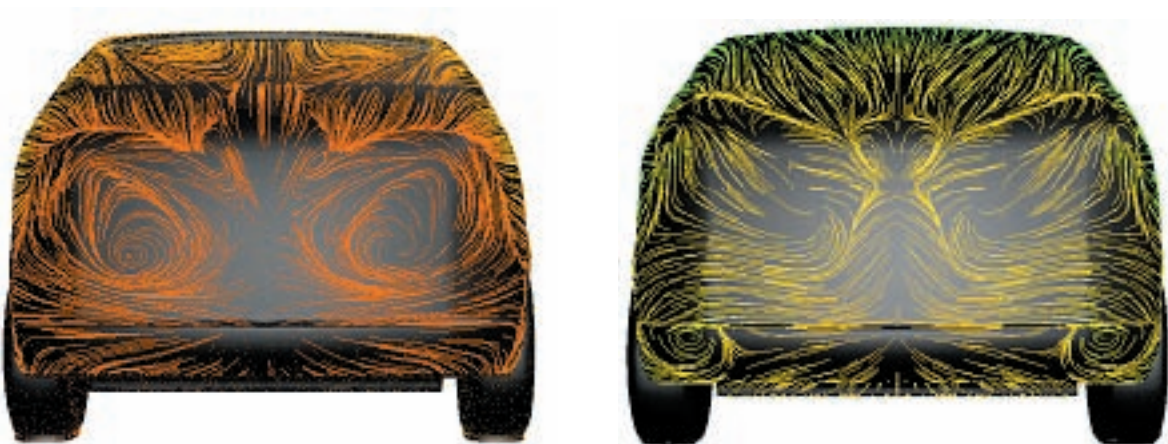
Oil film visualization was used to illustrate the different flow separation patterns.
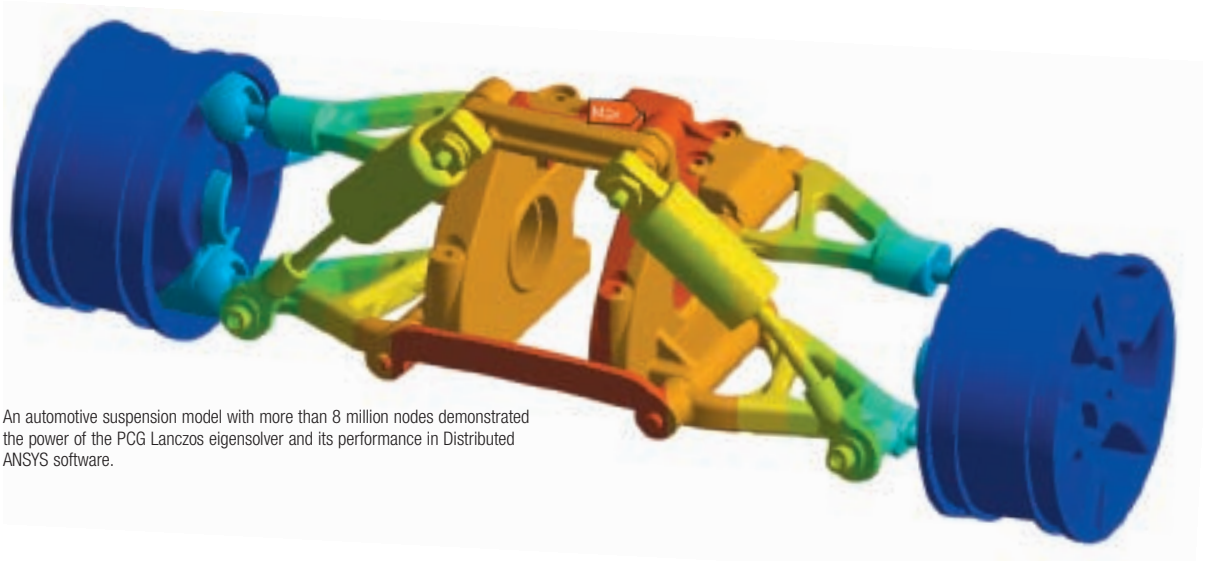
An automotive suspension model with more than 8 million nodes demonstrated the power of the PCG Lanczos eigensolver and its performance in Distributed ANSYS software.

Introducing the
PCG Lanczos Eigensolver

A new eigensolver in ANSYS 11.0 determines natural frequencies and mode shapes using less computational power, often in shorter total elapsed times than other tools on the market.

By Jeff Beisheim, Senior Development Engineer, ANSYS, Inc.

Eigensolvers determine natural frequencies and mode shapes of structures, typically some of the most computationally demanding tasks in structural analysis. ANSYS 11.0 software improves upon its extensive selection of these tools with a new eigensolver that combines the speed and memory savings of the ANSYS PCG iterative solver with the robustness of the Lanczos algorithm. This powerful combination allows users to solve for the natural frequencies and mode shapes of their model using fewer computational resources, often in shorter total elapsed times than other eigensolvers available on the CAE market.

Starting Up PCG Lanczos

The PCG Lanczos eigensolver can be selected using the LANPCG label with the MODOPT command (or in the Analysis Options dialog box). ANSYS Workbench users can choose this eigensolver simply by selecting the Iterative option in the Analysis Settings details pane.

The PCG Lanczos eigensolver works with the PCG options command (PCGOPT) as well as with the memory saving feature (MSAVE). Both shared- and distributed-memory parallel performance can benefit from this eigensolver. Note that in version 11.0, Distributed ANSYS software supports modal analyses, and the PCG Lanczos eigensolver is the only eigensolver available that runs in a distributed fashion in this software.

Controlling the Parameters

The PCG Lanczos eigensolver can be controlled using several options on the PCGOPT command. The first of these options is the Level of Difficulty value (Lev_Diff). In most cases, choosing the default value of AUTO for Lev_Diff is sufficient to obtain an efficient solution time. However, in some cases you may find that manually adjusting the Lev_Diff value further reduces the total solution time. Setting higher Lev_Diff values (e.g., 3 or 4) can help for problems that cause the PCG solver to have some difficulty in convergence. This typically occurs when elements are poorly shaped or are very elong-ated. A new Lev_Diff value of 5 is available in version 11.0 and warrants further explanation.

A Lev_Diff value of 5 causes a fundamental change to the equation solver being used by the PCG Lanczos eigensolver. It replaces the PCG iterative solver with a direct solver similar to the Sparse direct solver making the PCG Lanczos eigensolver behave more like the Block Lanczos eigensolver. Due to the amount of computer resources needed by the direct solver, choosing a Lev_Diff value of 5 essentially eliminates the reduction in computer resources obtained by using the PCG Lanczos eigensolver compared to using the Block Lanczos eigensolver. Thus, this option is generally only recommended for problems that have less than 1 million degrees of freedom (DOF), although its efficiency is highly dependent on several factors such as the number of modes requested and I/O performance, for example. A larger number of modes may increase the size of problem for which a value of 5 should be used while slower I/O performance may decrease the size of problem for which a value of 5 should be used.

While the PCG Lanczos eigensolver has many innovative algorithms to guarantee that no modes are missed, another new option on the PCGOPT command allows users to force a Sturm sequence check that helps guarantee that this is the case. It is important that users keep in mind that a Sturm sequence check works by using a direct solver to perform a numeric factorization after the Lanczos algorithm finishes. This factorization involves a large number of computations. It should also be noted that if using a Lev_Diff value of 1 through 4, executing a Sturm sequence check will eliminate the savings in computer resources obtained by using the PCG Lanczos eigensolver compared to using the Block Lanczos eigensolver.

Guidelines on Using the Tool

Typically, the following conditions must be met for the PCG Lanczos eigensolver to be most efficient:

- The model would be a good candidate for using the PCG solver in a similar static or full transient analysis.

- The number of requested modes is less than 100.

- The beginning frequency input on the MODOPT command is zero (or near zero).

Like all iterative solvers, the PCG Lanczos eigensolver is most efficient when the solution converges quickly. If a model would not converge quickly in a similar static or full transient analysis, the PCG Lanczos eigensolver would not be expected to converge quickly either and would therefore be less efficient. Other factors such as the size of the problem and the hardware being used can affect the decision of which eigensolver to use. For example, on machines with slow I/O performance or limited hard drive space, the PCG Lanczos eigensolver may be the better choice.
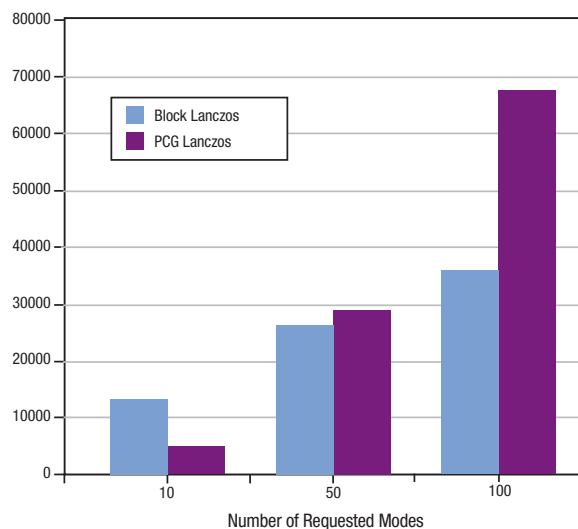
A comparison of the Block Lanczos and PCG Lanczos solvers for a problem with 1 million degrees of freedom. The simulations were run on a Dell® 670 workstation running Windows® x64 with a 3.6 GHz single-core Xeon® EM64T chip and 4 GB RAM.

Comparing Eigensolvers

The accompanying chart shows a sample comparison of solution times between the Block Lanczos and PCG Lanczos eigensolvers for a 1-million DOF model. When requesting 10 modes for this model, the PCG Lanczos eigensolver is faster. At higher numbers of modes, however, the Block Lanczos eigensolver becomes the faster option.

In other runs, as problem size increased the PCG Lanczos eigensolver was faster than the Block Lanczos eigensolver at a higher number of requested modes. For example, when solving this model with 500,000 DOF, the PCG Lanczos eigensolver was faster when computing less than 20 modes. At 1 million DOF, the PCG Lanczos eigensolver was faster when computing less than 40 modes.

Parallel Performance in Distributed ANSYS

A suspension model with more than 8 million nodes was used to demonstrate the power of the PCG Lanczos technology along with the performance of this eigensolver in Distributed ANSYS. This model was solved on an SGI® Altix® Linux server with Itanium®-II processors. Requesting 10 modes with a model of this size would likely take several days using the Block Lanczos eigensolver.

Using one processor, the PCG Lanczos eigensolver was able to find the requested modes in 37 hours. Using 10 processors dropped the elapsed time to six hours, demonstrating how the combination of the powerful PCG Lanczos algorithm along with the parallel performance of Distributed ANSYS makes the PCG Lanczos eigensolver one of the fastest eigensolvers available in the CAE market today. ■

CAE Cross Training

Engineers today need to be proficient in not one, but many analysis tools.

By Marty Mundy, ANSYS, Inc.

Historically, there have been occasions when the right combination of people and events has spawned technological advances that have effected great change in history. For numerical simulation, the mid-20th century was one such period. At that time, numerical simulation began to move to the forefront of engineering design and development. Since then, the scope of simulation tools available, computing capabilities and interest in using computer-aided engineering (CAE) to solve problems has grown. As design engineers consider how many environmental, structural, thermal and other factors to take into account in the development of a new product or the analysis of an existing one, they know that more is usually better. That is, more factors and a larger system model generally will produce more valuable information and allow for the development of a better final product. Unfortunately, this potential gain has had to be balanced by the reality that

FEA simulations, such as the spring plate shown here, take into account factors such as contact between components and loading in order to predict displacement. The initial mesh is shown on the left while the contour plot on the right depicts displacement.

more comprehensive analyses take more time to perform and ultimately cost more money. During the past 10 years, however, CAE software and computational capabilities have developed to a point where such types of analyses have become routine. As a result, designers are in a position in which they can consider performing more complex multidisciplinary analyses than ever before. To be successful in this new environment, they need to understand the variety of tools available, what those tools can accomplish and how they can work together. In short, the analyst of today is in need of CAE cross training.

For starters, all CAE tools are designed to reduce the costs of alternative efforts, like prototype testing, and to provide more detailed information than can be gathered by experimental means. At a core level, most CAE methodologies follow the same basic steps:

- **Pre-processing:** the procedures used to set up a simulation, such as defining the geometry, boundary conditions and initial conditions

- **Performing the calculation:** the meat of the analysis, in which the problem is solved to answer one or more questions

- **Post-processing:** the procedures and tools used to visualize and analyze the results of the calculation

Because the same basic steps are used in most types of CAE, if one understands the process followed for

one type of analysis, it is a fairly straightforward matter to understand the others.
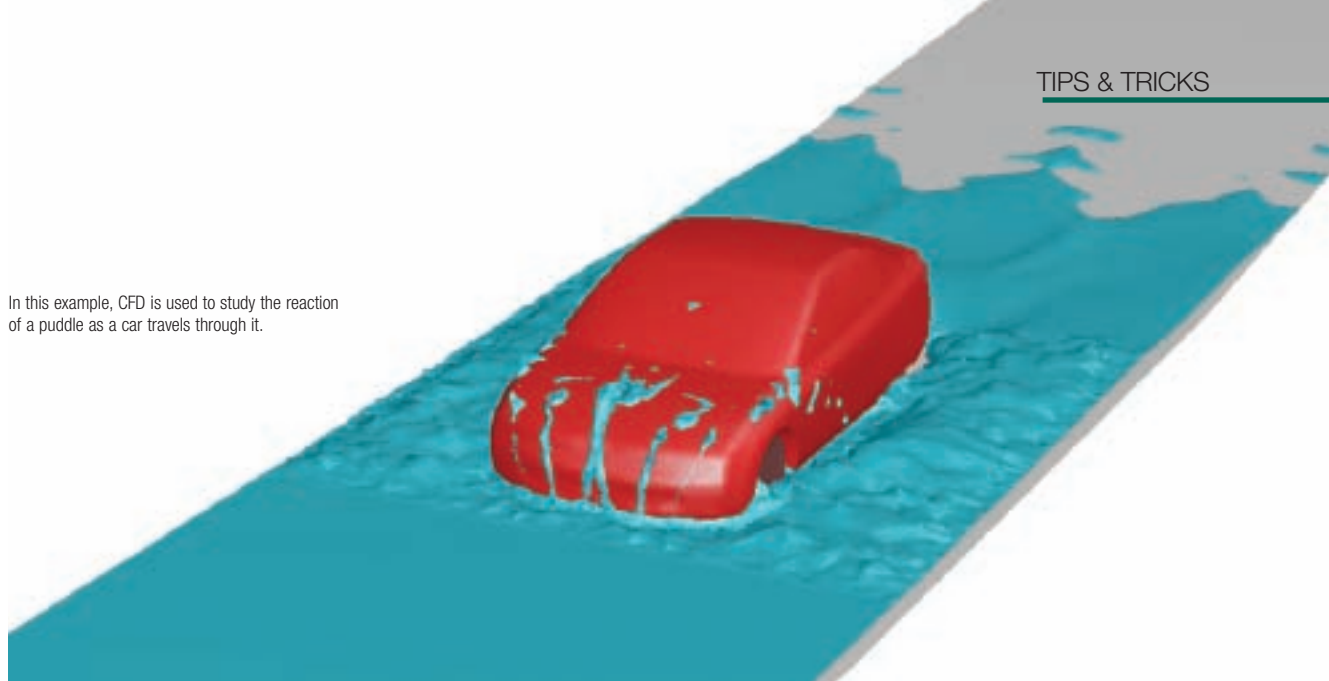
Structural Analysis

In the 1940s, structural simulation tools based on finite element analysis (FEA) initially were developed to study vibration. Further developments for applications in the automobile and aerospace industries soon followed. Over the years, FEA applications have broadened to include the study of loading, failure limits, material response, thermal applications and more. The FEA calculation uses the finite element method to calculate the stress and strain responses of a structure to initial or external conditions, analyzed as a series of smaller sub-structures or elements. These responses are analyzed as stress concentrations, displacements, vibration responses, and the prediction of resonant frequencies. During the solution, the initial loads, referred to as the structure load vector, are used to calculate a stiffness matrix for each element. The stiffness matrix consists of proportionality constants that relate applied forces to resultant displacements. These displacements are used, in turn, to determine the behavior of the entire structure under analysis. The calculation is performed either through a direct or iterative process. For a thermal analysis, load vectors in the form of heat fluxes or nodal temperature constraints are used and an overall energy balance throughout the domain is sought. The finite element method, while initially

In this example, CFD is used to study the reaction of a puddle as a car travels through it.

developed for structural analyses, also is used in fluid mechanics and for other numerical analyses.

Computational Fluid Dynamics

The development of computational fluid dynamics (CFD) began in the 1950s and was introduced into the aerospace industry in the 1960s. CFD solvers implement one or more of three primary numerical techniques: finite volume, finite element or spectral element, in order to solve a series of coupled, non-linear differential equations. The differential equations are used to describe the conservation of mass and momentum, for example, or the transport of quantities such as heat and turbulence. In general, the equations include terms that represent the change of a quantity though mechanisms like convection, diffusion and source or sink terms. CFD simulations can be performed for equilibrium (steady-state) or transient (time-marching) conditions. Turbulence models have played an important role in the development of CFD. The robust k-ε model was introduced by the Fluid Dynamics Group at Los Alamos National Laboratory in the United States in the 1960s and is still widely used today. It was not until the 1980s that CFD entered the mainstream industrial market. Its presence in the industrial community has lagged behind that of FEA, primarily due to the

computational demands of the underlying physics and chemistry, which can be difficult to solve simultaneously.

Fluid Structure Interaction

Coupled CAE simulations combine two or more different types of simulation and allow the engineer to better understand certain scenarios. The ability to combine tools together can enable design optimization to its fullest extent, giving a competitive edge to those who pursue this route. For example, fluid structure interaction (FSI) tools have enjoyed a rapid gain in popularity for applications ranging from sail design to artificial heart valve analysis.

As the name suggests, FSI is a coupled fluid–structural analysis that allows a structure to have a physical response to fluid behavior. Often, the problems faced by engineers are not independent of the environmental conditions, but rather factors of them. In the FSI methodology, the mechanical load that a fluid imparts on a solid body is used to predict the deformation, motion or vibrational response of that body. The deformation of the solid simultaneously changes the boundary conditions that affect the fluid behavior and are thus taken into account in a subsequent iteration of the fluid analysis. Additional applications of FSI include the study of airplane wing fatigue, the effect of detonation

on a structure, airbag or parachute deployment, and the effect of wind on urban structures.

The CAE spectrum of tools is used every day in the design, guidance and optimization of applications that are used in the manufacturing, environmental and nuclear industries, to name a few. With the goals of reducing prototype costs, reducing product time to market, and providing insight into otherwise less understood phenomena, CAE developments are ever advancing and broadening the scope of capabilities of the engineer. ∎
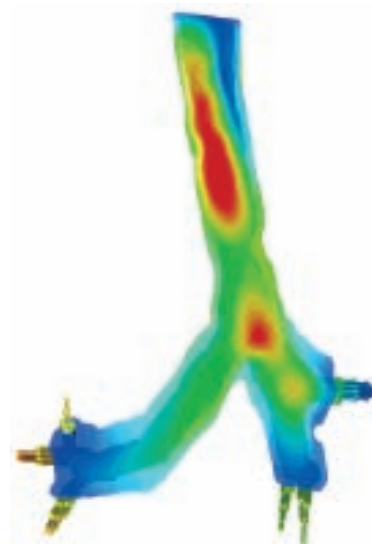
An FSI simulation was done using FIDAP to predict the deformation of the outer trachea wall, with velocity vectors shown at the bronchi exits. The shadow of the inner trachea wall is also shown. (CT data courtesy Dr. Thomas Frauenfelder, University Hospital of Zurich, Switzerland.)

View-Factoring Radiation
into ANSYS Workbench Simulation

The ANSYS Radiosity Solution Method accounts for heat exchange between surfaces using Named Selections and a Command object.

By Sheldon Imaoka, ANSYS, Inc.

Radiation can play an important role in heat transfer analyses. In ANSYS Workbench Simulation, a "Radiation" load is available that allows users to account for losses to the surroundings, although this does not include radiation exchange between surfaces. To utilize the ANSYS surface-to-surface radiation capabilities, an easy method is available to include these effects within ANSYS Workbench Simulation via Named Selections and a Command object.

How Heat Radiates

Radiation is a highly nonlinear mode of heat transfer. A simplified form of the equation describing radiation from one surface to another is:

$$Q_{ij} = A_i \varepsilon_i F_{ij} \sigma \left(T_i^4 - T_j^4 \right)$$

in which A is the surface area, ε is the emissivity of the surface, F is the view factor between surfaces, σ is the Stefan–Boltzmann constant, and T_i and T_j are the two surface temperatures in absolute temperature units.

When using the Radiation load in ANSYS Workbench Simulation, the user enters the emissivity and ambient temperature, while the form factor F_{ij} is assumed to be 1.0. In this situation, T_i reflects the absolute temperature of a node on the surface while T_j represents the ambient temperature. Temperature values are specified in °C or °F, and ANSYS Workbench Simulation automatically converts these values to Kelvin or Rankine, respectively.

This method is suitable when radiation only occurs to space and no surfaces are assumed to radiate between one another. In cases in which radiation occurs between surfaces of the model, however, this built-in ANSYS Workbench Simulation Radiation load is not sufficient, and surface-to-surface radiation utilizing the *ANSYS Radiosity Solution Method* is needed instead.

Performing Radiosity Solutions

The Radiosity Solution Method differentiates each independent region of radiation as an *enclosure*. Within each enclosure, view factors are determined, and the radiated heat flux is computed between each element face as well as to space, if the enclosure is open.

It is worthwhile to note that conduction and radiation calculations are performed in a segregated fashion. Depending on the degree of radiation heat flow in the model, the solution may require more iterations than regular ANSYS conduction-based solutions.

To use the Radiosity Solution Method, the following two steps are performed:

1. Designate surfaces that will radiate to each other via *Named Selections*.

2. Insert necessary ANSYS Parametric Design Language (APDL) commands using a *Command* object.

To create Named Selections, surfaces (or edges in 2-D) are selected and the "Create Selection Group" icon on the Named Selection toolbar is chosen. Surfaces that radiate between each other in an enclosure can be defined in one or several Named Selections. The only point the user needs to keep in mind is that all surfaces in a given Named Selection will be assumed to have the same emissivity values. With multiple Named Selections, however, each set of surfaces can have different values of emissivity.

As an example, consider one hollow block positioned inside another. Two Named Selections (REGION_A and REGION_B — one for each block) are needed to represent all surfaces that can radiate between each other. By making use of two Named Selections, the emissivity values of the REGION_A surfaces can be specified independent of those of the REGION_B surfaces.

In addition to specifying the surfaces of the blocks that can radiate between each other, a Command object also needs to be inserted under the "Environment" branch in ANSYS Workbench Simulation. Only a few APDL commands are required to specify the parameters needed for the Radiosity Solution Method:

```
sf,REGION_A,rdsf,0.9,1
sf,REGION_B,rdsf,0.8,1

stef,5.67e-8
toffst,273.15

hemiopt,10

tunif,20
```
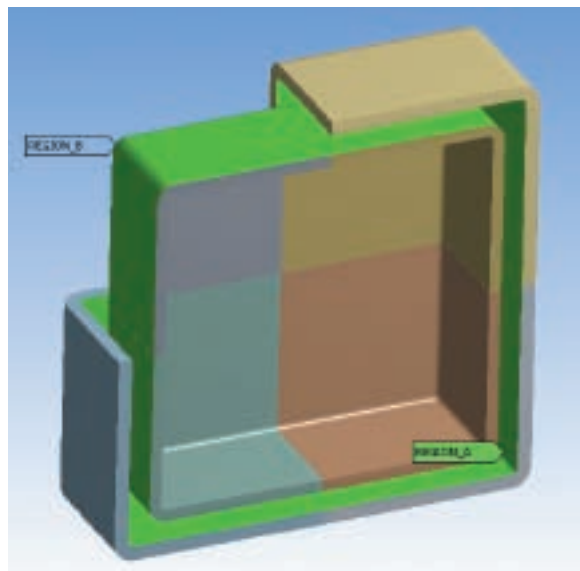
Two Named Selections (REGION_A and REGION_B) represent surfaces that can radiate heat between each other.
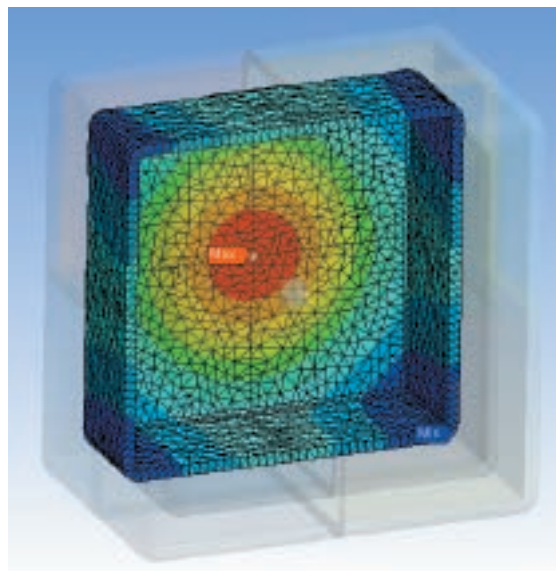
The temperature distribution of three blocks radiating between each other is shown in the figure above. In this case, the center small block (transparent) has a fixed temperature and is radiating to the middle block (displayed). This block, in turn, radiates to another block (transparent) that encloses it, and the third block radiates to space. This model illustrates the use of three radiation enclosures — two are closed while one is open. Radiation is the only mode of heat transfer between the three blocks, yet the Radiosity Solution Method can be incorporated easily inside ANSYS Workbench Simulation to solve these types of problems.

The meaning of these commands is explained below:

- The first two commands (SF) group Named Selections (in this case "REGION_A" and "REGION_B") to a given enclosure (1) with emissivity values of "0.9" and "0.8," respectively. If the enclosure is open, the SPCTEMP command also should be used to designate the space (ambient) temperature.

- The next two commands, STEF and TOFFST, specify the Stefan-Boltzmann constant and the temperature offset to absolute zero. These are unit-dependent, so users should make sure that the active unit system is set accordingly from the "Units" menu.

- The HEMIOPT command is optional, but it is used to set the resolution of the viewfactor calculations F_{ij}. The default value

is 10, but it may be increased for more accurate viewfactor calculations at the expense of additional CPU time. (For 2-D analyses, the analogous command is V2DOPT instead of HEMIOPT.)

- The last command, TUNIF, specifies the initial temperature in °C or °F. For a nonlinear steady-state thermal analysis, specifying a reasonable initial temperature will help convergence.

Handling Multiple Enclosures

If multiple enclosures (i.e., independent radiating regions) are present, SF commands may be repeated for the other Named Selections but referencing a different enclosure ID.

Users also are advised to turn on "Auto Time Stepping" and to specify the number of initial, minimum and maximum substeps under the

"Solution" branch since radiation problems can be highly nonlinear.

A number of advanced Radiosity Solution Method options also are available:

- RADOPT to specify solver controls
- VFOPT to write or read the viewfactor file
- RSYMM and RSURF to take advantage of planar or cyclic symmetry
- RDEC and RSURF to coarsen the element faces for radiation calculations only
- Temperature-dependent emissivity specification is also possible

All of the APDL commands are documented in the *ANSYS Commands Reference*, and additional details of Radiosity Solution Method options can be found in Sections 4.6 and 4.7 of the *ANSYS Thermal Analysis Guide*. ■
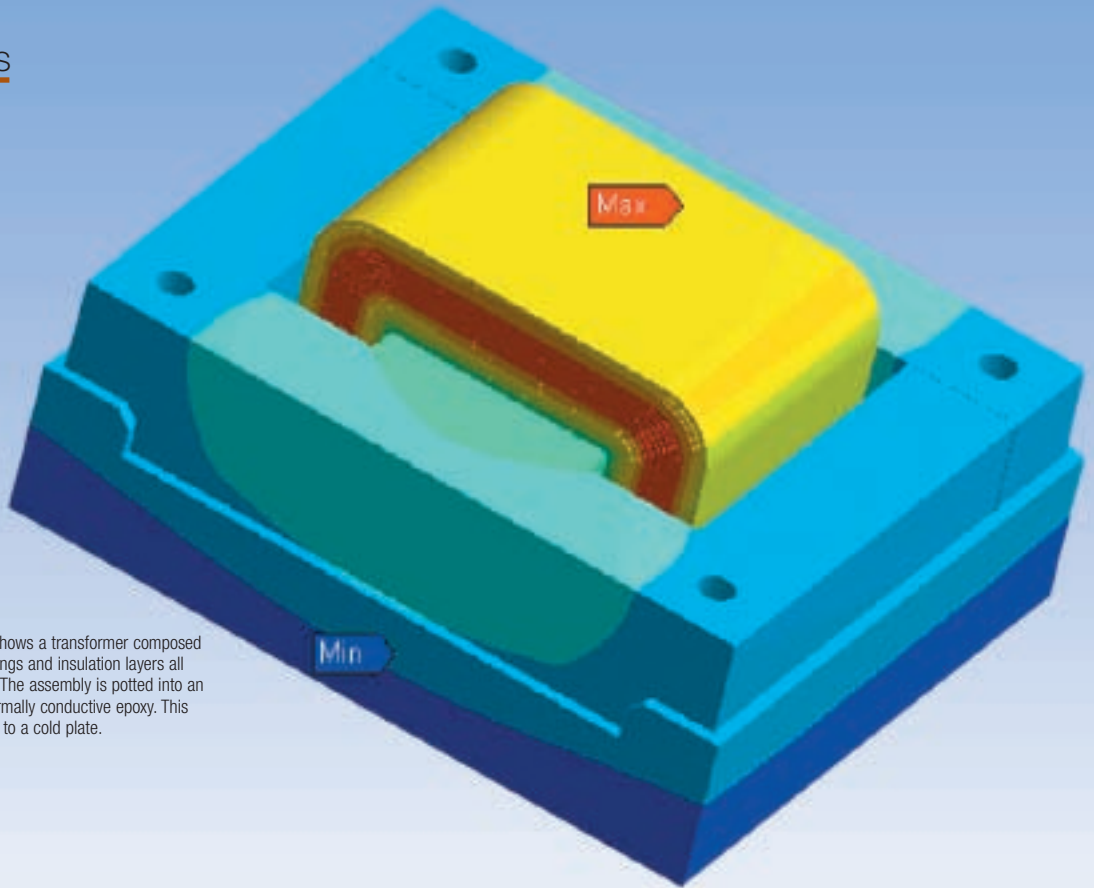
This MatWeb application shows a transformer composed of steel core, copper windings and insulation layers all spiraled into an assembly. The assembly is potted into an aluminum case with a thermally conductive epoxy. This assembly is then mounted to a cold plate.

Going to the **Source**

MatWeb material property data is seamlessly available to ANSYS Workbench users.

Because of the ANSYS partnership with MatWeb® announced in May 2006, ANSYS Workbench users can easily import material property data directly from MatWeb (www.matweb.com), which has an on-line library of data sheets on more than 62,000 metal, plastic and ceramic materials — plus online tools including a unit converter, weight and inertia calculator and hardness converter. On each data sheet, MatWeb lists the source of the materials data. About 90 percent of the data originates from testing by material suppliers. Other data is collected from similar materials, professional societies, handbooks and compilations as well as the MatWeb staff.

One of the companies benefiting from this MatWeb library within ANSYS is TDI Power, which develops power systems for computer/networking, telecommunications, military/aerospace, medical and industrial markets. Sebastian Messina, principle

mechanical engineer at TDI, explains that in building uninterrupted power supplies, for example, TDI uses a variety of materials including steel, aluminum, epoxies, RTV (room temperature vulcanization) materials, glass, plastics, copper and Teflon. TDI engineers can export 20 material libraries at a time in ANSYS library format (XML), with material property values in appropriate units automatically added to the XML file.

The idea for MatWeb came from engineers frustrated by having to interrupt the work on their software application and take valuable time telephoning manufacturers for material property data or hunting through handbooks. "When the Internet exploded, it made perfect sense for us to plug our knowledge into it," says Dale Kipp, who heads up day-to-day operations at MatWeb. "Now engineers have convenient access to property data for the exact material they need.

With just a few clicks, they can import the information right into their ANSYS software application."

Sebastian Messina at TDI agrees: "If there was no integration between ANSYS and MatWeb, I would be spending extra time creating material property tables. That's hours per analysis." He adds that the tools are user-friendly. "I don't have the time to work in code and become a software guru," says Messina. "I want to plug and play, and the ANSYS MatWeb tools allow me to do so."

"The collaboration with MatWeb reflects the ANSYS strategy to team with expert partners to provide integrated solutions in areas outside our core competencies," says Mike Wheeler, vice president at ANSYS, Inc. "It is an extremely useful extension of the flexible framework of ANSYS Workbench." ■

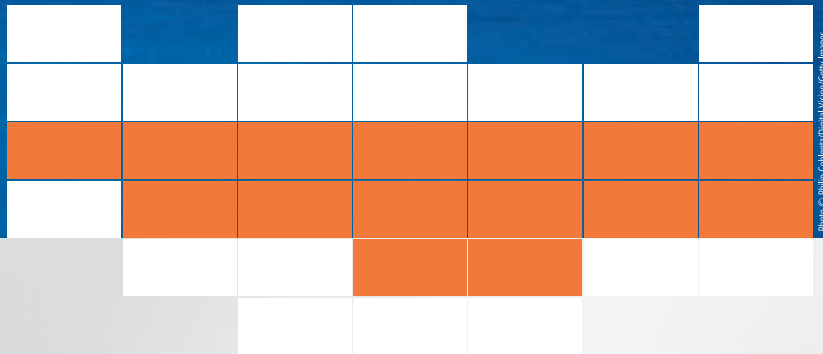*Microsoft*®

Harness the Power of **Two**

ANSYS®

Microsoft® and ANSYS, Inc. are leading the evolution of CAE tools and technologies. By leveraging the combined strength of ANSYS® and FLUENT® applications with the power of Windows® Compute Cluster Server, you can develop solutions and access data quicker and more efficiently, all while working in a familiar environment backed by Microsoft's secure infrastructure.

For more information on how you can harness the power of two, log on to www.microsoft.com/hpc

Windows®
Compute Cluster Server 2003

Photo © Philip Coblentz/Digital Vision/Getty Images