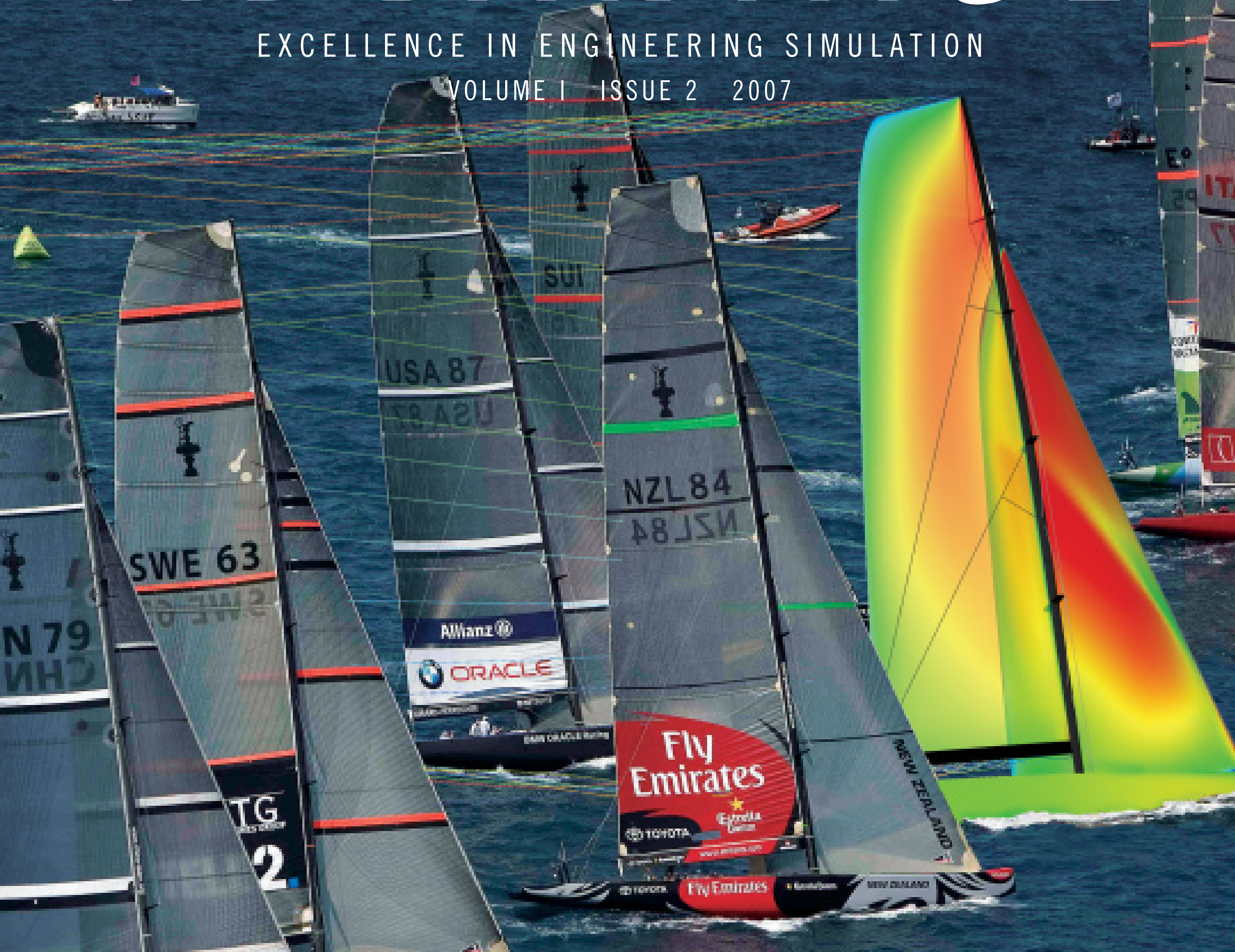
ANSYS

ADVANTAGE

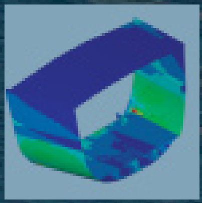
EXCELLENCE IN ENGINEERING SIMULATION

VOLUME I ISSUE 2 2007



THE PROVEN WINNER IN CAE

PAGE 3

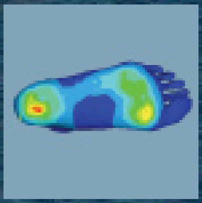


COMPOSITES

PAGE 7

HAIR CARE

PAGE 10

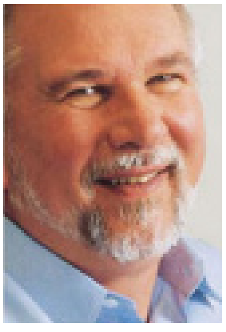


INSOLE DESIGN

PAGE S7

The **Big** Question

With engineering simulation becoming so widespread, the big question now for manufacturers is not *if* to use the technology but *how*. And their answers will determine who gains the competitive edge.



Companies are investing in engineering simulation at unprecedented levels. According to the latest statistics from market research and technology assessment firm Daratech, sales revenue for computer-aided engineering (CAE) software and services grew from $2.31 billion in 2005 to more than $2.43 billion in 2006. Daratech forecasts a compound annual growth rate of 13 percent through 2010, when figures are expected to top $3.7 billion. Driving this expansion is the tremendous need for companies to shorten time to market, lower costs, improve performance and develop steady streams of knock-out, innovative products. With survival on the line in many cases, manufacturers use simulation as a proven way of addressing these issues.

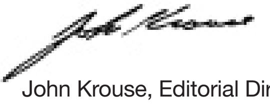
According to Daratech statistics, the number-one player in CAE is ANSYS, Inc. So a substantial portion of the broad range of simulation applications worldwide is based on the company's suite of solutions. The breadth of applications is evidenced by the articles in this current issue of *ANSYS Advantage* on simulation projects involving the design of products ranging from trucks and turbojet engines to consumer goods and healthcare equipment. The content also demonstrates the vast range of company sizes, from the one-man design firm Stein Design in the story "No-Hassle Kitchen Appliance" to the $70 billion global consumer product giant Procter & Gamble Company in the article "The Democratization of Engineering Analysis."

As these and other successful simulation users know, gaining market advantage now takes more than just utilizing analysis tools. Because of the ubiquitous use of CAE technology, the competitive edge isn't necessarily determined by which companies use simulation — most manufacturers now implement it in one way or another — but rather how they uniquely apply the technology in their organizations and integrate it into their product development processes.

To fully leverage a solution, many successful firms have initiatives for performing more upfront simulation to refine designs early instead of trying to hurriedly fix problems near the end of development. In most cases, this means deploying appropriate tools beyond the ranks of dedicated analysts to more rank-and-file engineers and designers for routine use throughout development.

There is no cookie-cutter way to best implement such an approach. Rather, companies have found that they must carefully evaluate their existing processes, skill sets, organizational structures, product strategies and business priorities to leverage simulation most effectively. Scheduling, funding and performance reviews generally are adjusted to allow for training; the approach also gives engineers and designers the time they need to perform analysis, what-if simulations and optimizations in the early stages of development rather than the usual rush to finalize computer-aided design (CAD) models.

These and other necessary organizational changes require a significant investment in time and effort, of course, but the level of commitment defines how companies uniquely leverage simulation; it also determines which firms will most likely lag behind while others reap the greatest business value from Simulation Driven Product Development. ■



John Krouse, Editorial Director

For ANSYS, Inc. sales information, call **1.866.267.9724**, or visit **www.ansys.com**.
To subscribe to *ANSYS Advantage*, go to **www.ansys.com/subscribe**.

Editorial Director
John Krouse

Production Manager
Chris Reeves

Art Director
Susan Wheeler

Editors
Marty Mundy
Fran Hensler
Erik Ferguson
Richard LaRoche
Thierry Marchal

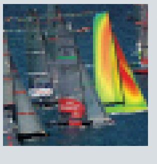
Ad Sales Manager
Beth Mazurak

Editorial Advisor
Kelly Wall

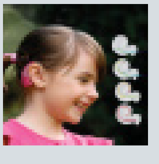
Circulation Managers
Elaine Travers
Sharon Everts

Designers
Miller Creative Group

Email: **ansys-advantage@ansys.com**

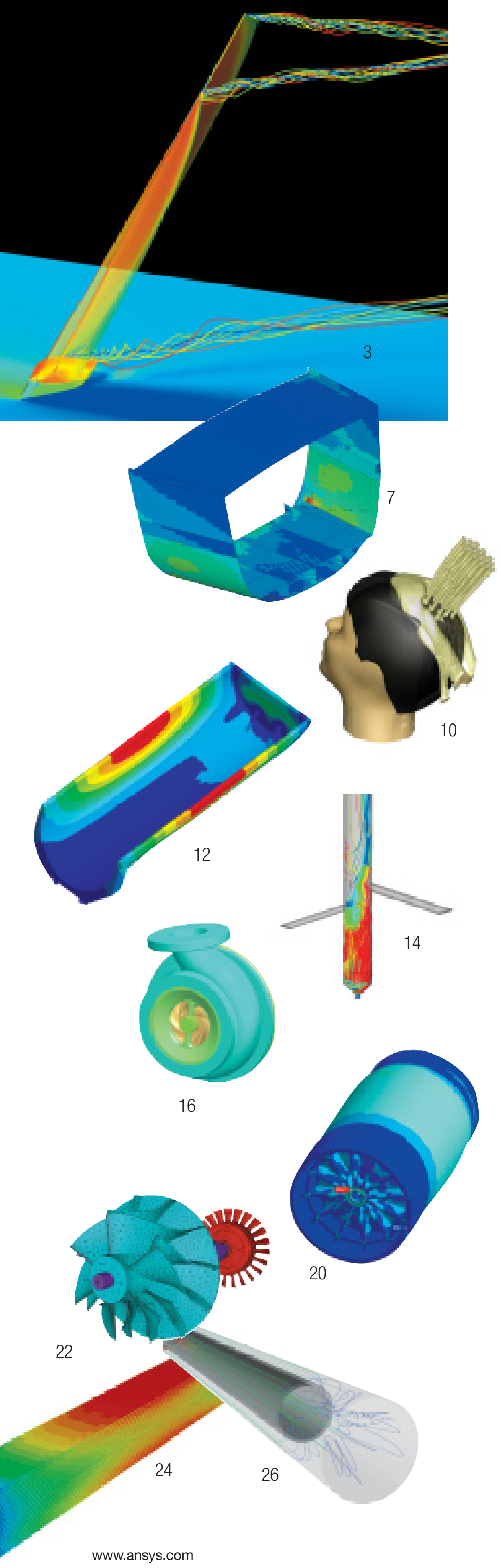


About the Cover
Racing yachts competed this summer in the America's Cup competition in which all the leading teams employed computer simulation to gain an edge. See page 3. Cover photograph ©ACM 2007/Photo: Carlo Borlenghi. Simulation courtesy Christos Pashias, Team Shosholoza.



About the Biomedical Spotlight
The biomedical industry is emerging as a strategic user of engineering simulation. One research team has found that improvements in cochlear implants might be possible using shape memory alloys. See page s4.

ANSYS Advantage is published for ANSYS, Inc. customers, partners and others interested in the field of design and analysis applications.



Contents

Feature

3 SPORTS
The Simulation Race for America's Cup
Yacht designers used engineering simulation in a variety of applications to edge out the competition.

Applications

7 MATERIALS/PARTNERSHIPS
Plying the Composite Trade
ESAComp software overcomes challenges in designing with composites.

10 CONSUMER PRODUCTS
Hair Today
Product developers in the cosmetics industry can put simulation to use in performing hierarchical analyses of hair care product performance.

12 AUTOMOTIVE
Heavy-Duty Lightweight
An innovative aluminum design gives a truck-body manufacturer the competitive edge in the worldwide construction industry.

14 POWER GENERATION
Gassing Up with Coal
A two-fluid multiphase model allows for more accurate simulation of coal gasification.

16 PROCESS EQUIPMENT
Chopping Away at Solids
CFD simulation provides a pump company with a virtual test facility.

20 CONSUMER PRODUCTS
No-Hassle Kitchen Appliance
Finite element analysis helps redesign a countertop water filter.

22 AEROSPACE
Overcoming Big Challenges for Small Turbojet Engines
Engineers used FEA to develop an impeller for a microjet turbine engine for unmanned drone aircraft.

24 POWER GENERATION
Keeping It Cool
Modeling fluid flow and heat transfer throughout a nuclear fuel assembly helps prevent reactor burnout.

26 PROCESS EQUIPMENT
The Greening of Gas Burner Design
Simulation assists in developing efficient and environmentally friendly recuperative burners used in heat-treating applications.

(Continued on next page)

Departments

19 TRENDS & PRACTICES

Managing Engineering Knowledge

Web-based solution is aimed at hosting and integrating simulation data, processes and tools for more effective Simulation Driven Product Development.

28 THOUGHT LEADERS

The Democratization of Engineering Analysis

To compete successfully in today's business climate, Procter & Gamble makes analysis tools available to rank-and-file engineers as well as to analysts and advanced simulation experts.

31 ANALYSIS TOOLS

Rotordynamic Capabilities in ANSYS Mechanical

Useful features are available to study vibration behavior in rotating parts.

34 TIPS & TRICKS

Submodeling in ANSYS Workbench

To obtain accurate stress in a local region, submodeling separates local analysis from the global model.

Spotlight on Engineering Simulation in the Biomedical Industry

s2 **Making Life Longer and Better**

The biomedical industry is emerging as a strategic user of engineering simulation.

s4 **Turning Up the Volume**

The use of shape memory alloys offers the promise of better functioning in cochlear implants.

s6 **Hip to Simulation**

Evaluation of designs for a hip replacement prosthesis overcomes physical and scientific limitations.

s7 **Walking Pain Free**

New insoles designed with the ANSYS mechanical suite relieve pain from foot disease.

s8 **Engineering Solutions for Infection Control**

Simulation assists in designing a hospital ward to reduce the airborne transmission of disease.

s10 **Standing Up Right**

ANSYS Multiphysics sheds light on the wonders of the human spine and how to fix it.

s12 **Designing with Heart**

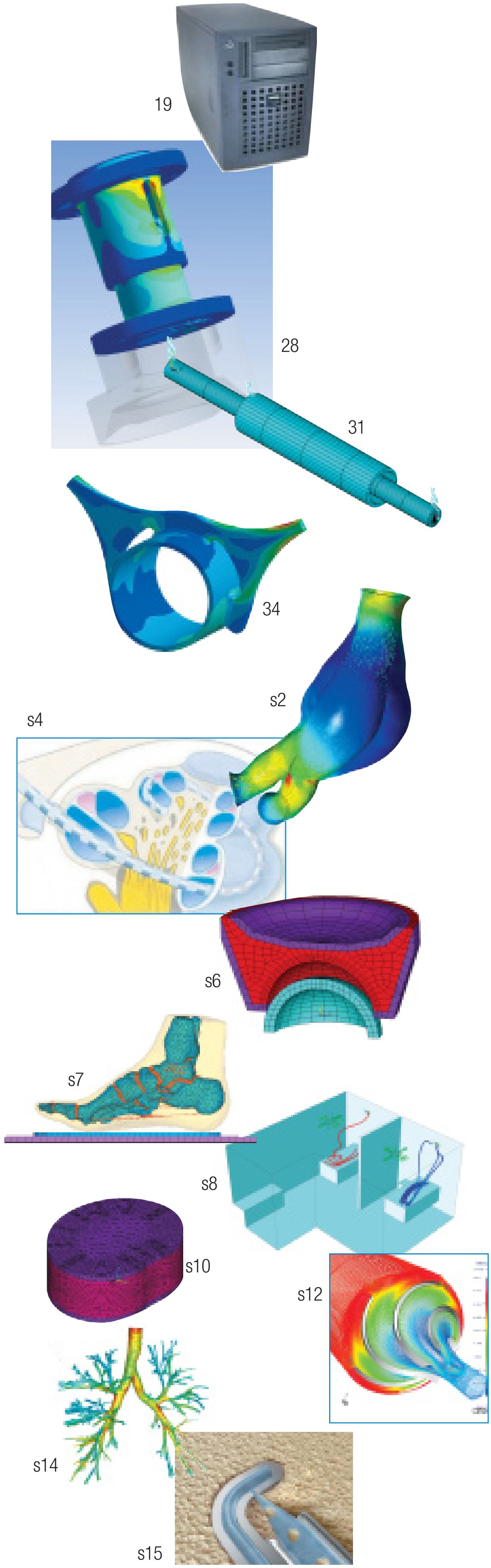
CFD-based design optimization for a pediatric implant can shave years off the development cycle.

s14 **Going with the Flow**

Functional biomedical imaging through CFD provides a new way of looking at pathological lungs.

s15 **Battle of the Bulge**

Rapid prototyping results in a new surgical tool to treat back pain.



Emirates Team New Zealand used CFD to predict the effect of design alternatives on yacht performance.

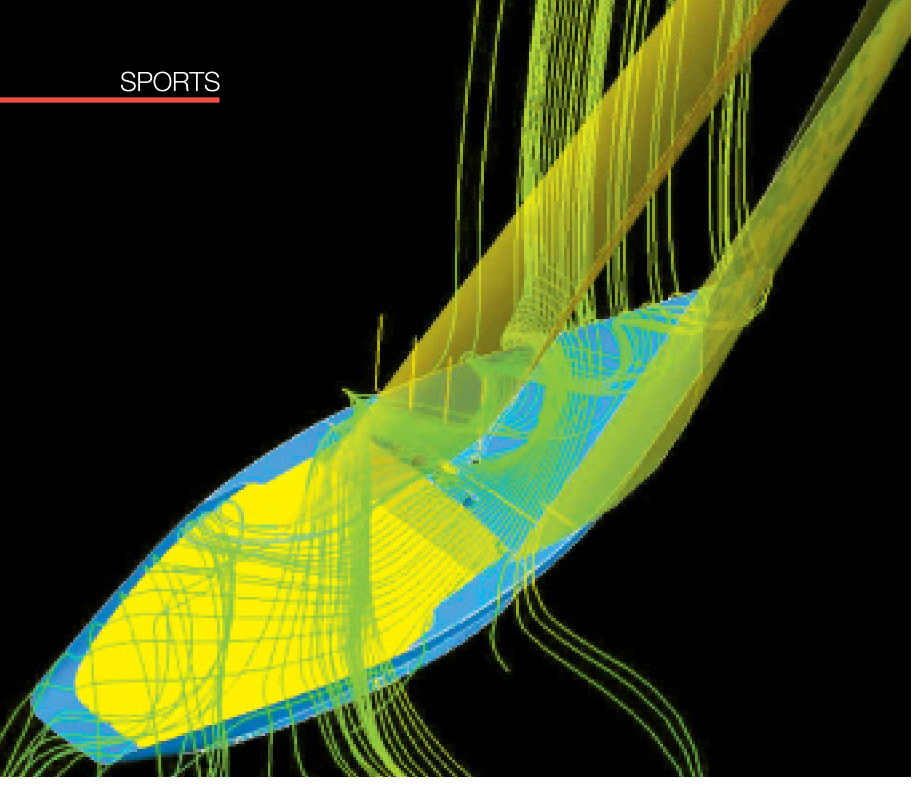
The Simulation Race for America's Cup

Yacht designers used engineering simulation in a variety of applications to edge out the competition.

The America's Cup is the most famous sailing regatta in the world and also the oldest active trophy in international sport. The trophy, originally known as the Royal Yacht Squadron Cup, was first awarded in 1851 when the New York Yacht Club schooner *America* defeated 15 Royal Yacht Squadron challengers in a race around the Isle of Wight in England. In honor of *America*'s victory in the first competition, and the subsequent dominance of American boats for over a century, the trophy officially became known as the America's Cup.

Despite its name, it is truly an international competition. In 2003, the Swiss challenger Alinghi defeated Team New Zealand to win sailing's grand prize; Alinghi successfully defended this summer at the 32nd America's Cup in Valencia, Spain. The boat sizes and designs have varied through the years, ranging from the 130-foot J-class yachts of the 1930s to a 60-foot catamaran in 1988. Since 1992 though, the teams have sailed an International America's Cup Class (IACC) sloop, a monohull boat that has an average length of about 75 feet. To determine which team would challenge Alinghi for the trophy in 2007, an ambitious schedule of regattas was held, commencing in 2004 and culminating with the Louis Vuitton Cup this past spring. The America's Cup match series was held in late June and early July, with Alinghi the winner in the closest Cup in recent history.

The racing syndicates that compete for the cup are composed of the best sailors, designers, sailmakers, nautical engineers and boat builders in the world. The top teams expend more than 150,000 labor hours to optimize the designs of their boats. All of the leading teams employ computer simulation to determine the power generated by the sails, the drag produced by the boat's hull and the air resistance of the deck. Four of the top teams, including BMW ORACLE Racing from the United States, South Africa's Team Shosholoza, Emirates Team New Zealand (ETNZ) and defending champion Alinghi from Switzerland, use computational fluid dynamics (CFD) software from ANSYS, Inc. to predict the effect of design alternatives on yacht performance down to the smallest details.



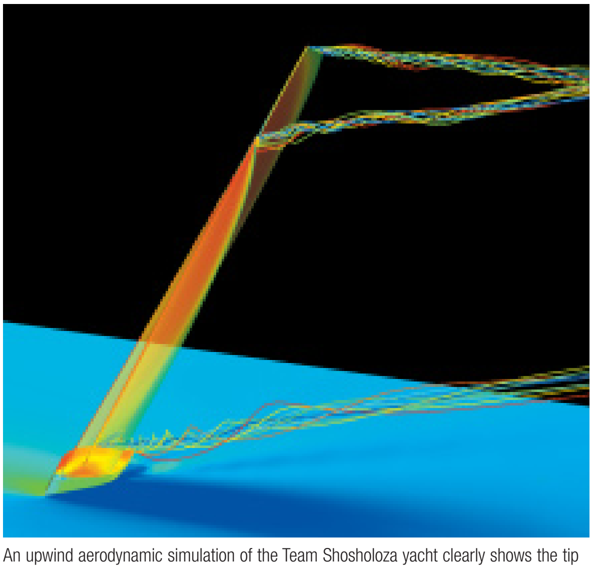
CFD simulates the wind flowing over the deck and cockpit of the Alinghi boat. Note the vortex that formed in the bow where the wind wraps around on the deck.

The two most critical aspects of yacht performance are the sail aerodynamics and the hydrodynamics of the hull and appendages. Picture this analogy: A racing yacht is like a plane floating on its side with one wing sticking up in the air and the other down into the water. The art of yacht design is to extract drive force because the two fluids (air and water) have different speeds and directions. The curvature of the sails generates lift in a manner similar to an airplane wing, while the keel of the boat generates lift in the opposite direction — like the opposite wing of the airplane — to prevent the boat from moving sideways. The keel can be proportionately much smaller than the sails because it operates in a fluid 800 times denser than air. As in aircraft design, improving performance of a racing yacht is basically a question of maximizing lift and minimizing drag. Small changes in geometry often make the difference between a competitive boat and an also-ran.

BMW ORACLE Racing: It's In the Details

In the 2003 competition, BMW ORACLE Racing used a public-domain CFD code to simulate the performance of their boat. However, they found that meshing and solution times were so long that they were forced to simplify their models to the extent that they could not distinguish between small design changes. For the 2007 race, the team used ANSYS CFX software. BMW ORACLE Racing ran models with 10 to 15 million cells on large computer clusters that can resolve the performance impact of the smallest design changes. The team's designers simulated the performance of large numbers of different sail shapes and trims to understand performance under a variety of conditions. They evaluated the aerodynamic effects of the deck, such as the shape of edges and corners and the position of the winches, and they also looked at the shape of underwater components, such as the ballast bulb.

"Our new simulation methods make it possible to model the most complex problems down to the finest details in a day or two," said Ian Burns, design team coordinator for BMW ORACLE Racing. "We now can determine the effect of the smallest changes, such as the shape of the deck or small hardware components on the mast. Some of these changes can have a significant impact on performance and are helping us make significant performance improvements. We have analyzed and improved nearly every detail of the boat with ANSYS CFX software."



An upwind aerodynamic simulation of the Team Shosholoza yacht clearly shows the tip vortices. Induced drag reduction is important for sails operating near their maximum lift.

Team Shosholoza: Big Things from Small Packages

Team Shosholoza, South Africa's first America's Cup entrant, was one of the smaller teams in this year's competition. Unlike some of the larger teams, Shosholoza has only one boat, so it can't rely on running two boats against each other to evaluate design changes. Therefore, CFD simulation is critical to the team, which has built a 42-node cluster that places it near the top in terms of computing capabilities among the smaller entrants. Shosholoza used computer-aided design (CAD) tools to develop a parametric model of the boat and then read the model into the ANSYS ICEM CFD Hexa meshing tool, which quickly generates a series of models by varying a key design variable over a defined range. Shosholoza then solved the models with ANSYS CFX software, and designers used the results for force and drag to predict the velocity.

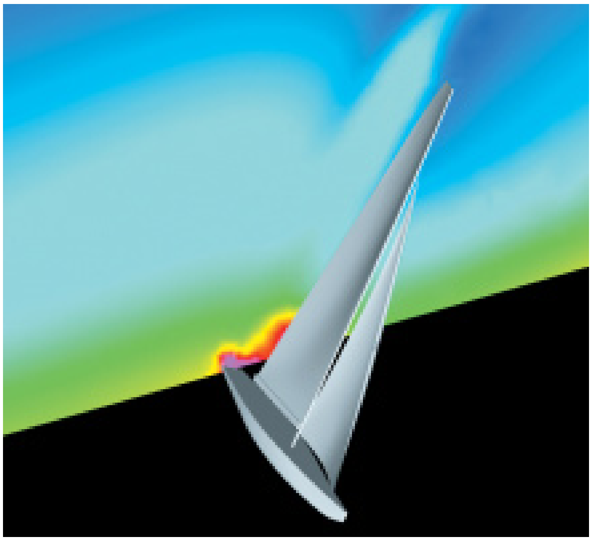
"To date, the area where we have made the greatest improvements is in the shape of the sails," said Christos Pashias, fluid dynamicist for Shosholoza. "We are trying to get as much power out of the sails as possible because the winds in Valencia are so light. We set up a parametric model to automatically generate sail models. This enabled us to have a quick turnaround and study more shapes. Being a new team, initially we made improvements of between 5 and 10 percent in driving force. A 1 percent improvement in driving force typically increases the speed of the boat by about 0.1 percent. We have tested boats with the new designs and discovered that they actually do provide the performance improvements that ANSYS CFX predicts. Since we made those initial big gains, we have made many other improvements that have provided smaller gains, typically in the area of 1 percent, which is what most teams are after. Testing already has shown that these predictions are accurate, so we trust them to make more improvements."

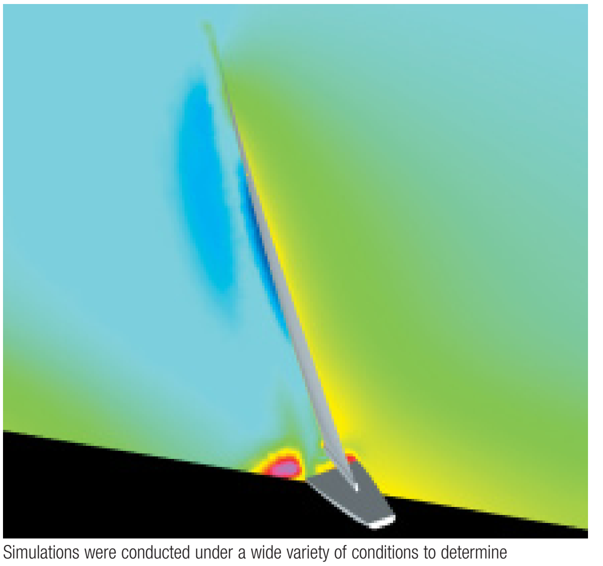
Shosholoza also used FLUENT CFD software to better understand the flow of water around the yacht. The ranking of candidate hull shapes by FLUENT software agreed well with experimental results.

Emirates Team New Zealand: Location, Location, Location

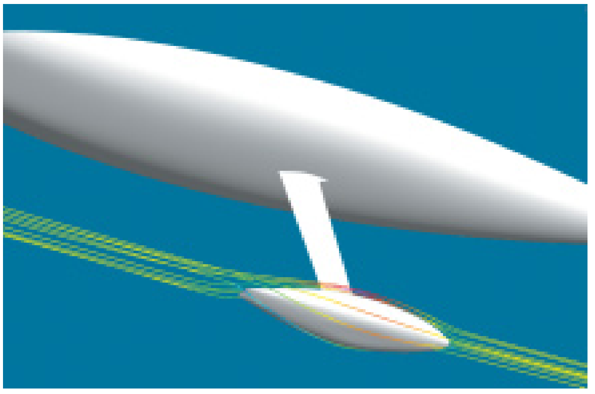
ETNZ has been focused on improving the ballast bulb at the bottom of the boat. At about 21 tons, this torpedo-shaped lead component makes up nearly 80 percent of the boat's mass and provides the craft with the stability to balance a very large sail area. Choosing a bulb shape with a lower center of gravity increases the boat's righting moment and enables the sail to provide a larger driving force. On the other hand, moving to a lower drag force wastes less of the available driving force and increases the speed of the yacht. In preparing for the 2003 race, the New Zealand designers were able to lower the center of gravity substantially without any increase in drag. With these major improvements under its belt, the team's goal for 2007 was to make more subtle and site-specific changes, such as optimizing the bulb design for the expected conditions off Valencia.

"We developed a genetic algorithm that works by defining the geometry of the bulb with control points whose coordinates and weighting are considered to be genes," said Nick Holroyd, designer for ETNZ. "Then the population was seeded with a range of candidates, and mutations were introduced into each generation to adequately spread the population across the design space. Each candidate was simulated with ANSYS CFX software using the laminar-to-turbulent transition model to provide a drag value. This value is factored against the stability contribution of the shape to provide a fitness score for the design. We developed a family of new bulb shapes with a better





Simulations were conducted under a wide variety of conditions to determine performance. Velocity magnitude contours around the hull and sails of the BMW ORACLE Racing boat are shown (windward above and leeward below) with plane cuts that are perpendicular to the boat track.



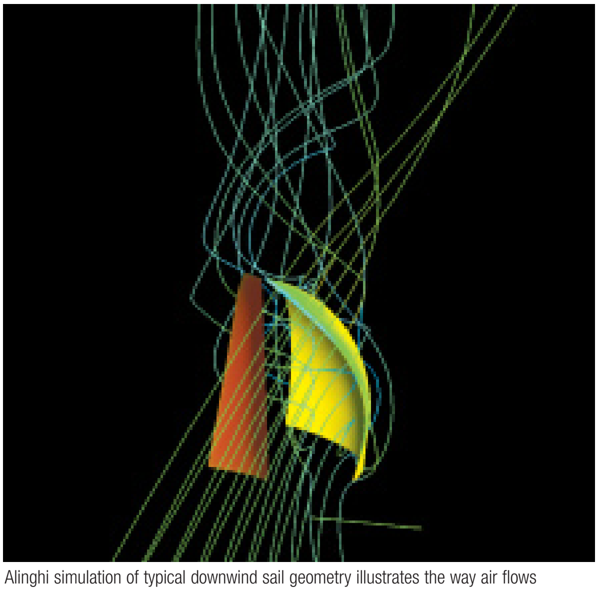
BMW ORACLE Racing has analyzed and improved nearly every detail of the boat, including the keel–ballast bulb juncture.

Team Shosholoza

drag/stability trade-off for the racing conditions expected at Valencia. This approach made it possible to evaluate the design space with much less time than would have been required manually."

Alinghi: Defending Its Honor

Winner and defending champion Alinghi used CFD to evaluate every portion of the boat, including the sails, the underwater portion of the hull and deck details. Alinghi designers spent more than a year evaluating CFD results compared to wind tunnel testing and scientific papers. "We gained confidence in the ANSYS CFX software and calibrated its results," said Jim Bungener, CFD engineer for Alinghi. "The main areas where we have made performance improvements have been in the winglets on the ballast bulbs and the downwind sails or spinnakers. We also have made smaller gains in areas such as winch placements and pillar shapes. These improvements have significantly increased the speed of the boat. When considered as a whole, the results that we have achieved with CFD aided us considerably in defending the America's Cup." Bungener also used ANSYS Structural software to identify the composite laminar structure that withstands the loads on the hull while minimizing weight.

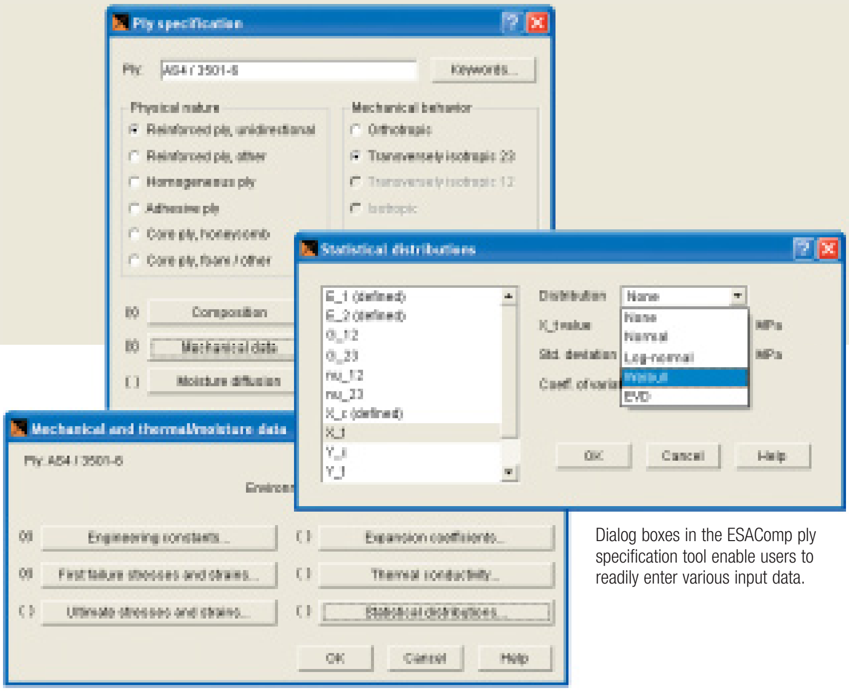


Alinghi simulation of typical downwind sail geometry illustrates the way air flows over the sails. A large vortex is created behind the spinnaker, a billowing sail used when the wind is behind the boat.

Steady Wins the Race

Computer simulation has played a crucial role in the boat design process for many of the top racing syndicates. With all entrants now using CFD to optimize the performance of their boats, different design groups have arrived at generally the same conclusions and made substantial performance improvements. As a result, the boats are closer together in terms of performance, making tiny improvements that much more important. The teams now are all creating finer and finer meshes using larger clusters of computers so they can evaluate the effects of smaller design changes on yacht performance. The America's Cup is thus becoming a showcase, not only for the world's fastest yachts but also for its most powerful simulation tools. ■

This article was written through contributions from Alinghi, BMW ORACLE Racing, Emirates Team New Zealand and Team Shosholoza.



Dialog boxes in the ESAComp ply specification tool enable users to readily enter various input data.

Plying the **Composite Trade**

Coupled with technology from ANSYS, Inc., ESAComp software overcomes challenges in designing with composites, enabling engineers to evaluate part designs and better use these versatile materials to their full advantage.

By Harri Katajisto
Componeering Inc.
Helsinki, Finland

Carbon-fiber reinforced plastics and other composite materials are used in a wide range of applications because of their high strength-to-weight ratios. High-performance composites made of continuous fibers bound with thermoset resins can be used in making extremely efficient structures, and laminated composites are well suited for lightweight parts with complex surface contours.

Composites present some complex challenges in utilizing these materials to their best advantage, however. Material properties are anisotropic — that is, they are directionally dependent on the orientation of the reinforcing fibers. Differences in thermal expansion of the matrix and reinforcing materials cause residual stresses, and asymmetric structures especially can yield unexpected responses to temperature variations. Moreover, sandwich structures exhibit complex behavior because of large differences in strength and stiffness between layers.

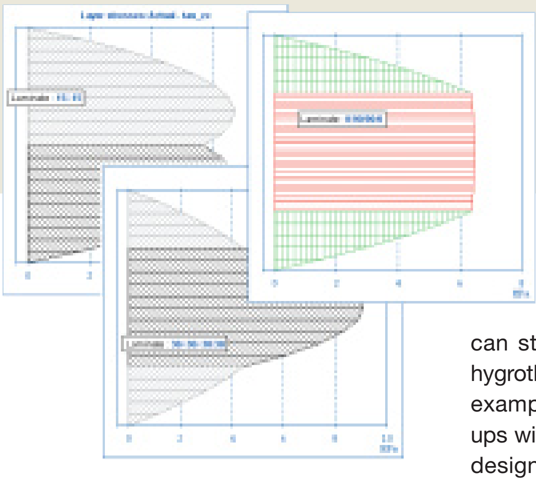
When designing with laminated composites, engineers must take into account these and many other considerations in establishing important design variables, including selection of material types, layer orientation and thickness, number of layers, and stacking sequence. Compounding the difficulty, complete material property data cannot always be found from the supplier data sheets.

Composite Analysis and Design

High-performance composites are used extensively in the aerospace industry, where engineers rely on in-house tools developed specifically for composite analysis. These programs require considerable resources to develop and maintain, however. Engineers need extensive training to understand the specialized command-based interfaces and numerical outputs. In addition, users often have to transfer data manually between multiple programs for modeling and analyzing components.

Concerned about the inefficiency and lack of consistency between the wide range of in-house codes used in the aerospace industry, the European Space Agency (ESA) initiated a project in the early 1990s to standardize the analysis approach with a single software platform combining various tools under a unified user interface. ESA, with headquarters in France and consisting of 17 member states, is in charge of shaping the development of Europe's space capability and ensuring that investment in space continues to deliver benefits to the citizens of Europe. By coordinating member resources, the agency can undertake programs and activities far beyond the scope of any single European country. ESA also works closely with space organizations outside Europe.

Development work for the composite project was conducted by Helsinki University of Technology in Finland, and the first version of ESAComp software was released in 1998. Development responsibility later



Layer charts indicate the effect of layer orientations on interlaminar shear stress distribution in a short beam test sample.

was transferred to the spin-off Finland-based company Componeering Inc., which now distributes and supports the software. Although the software originated in the aerospace industry, it has been developed as a general tool for engineers in other applications designing with high-performance composites, including automotive, marine, construction, machinery, rail transportation, sports and wind energy.

The software has analysis and design capabilities for solid–sandwich laminates and micromechanical analyses. It further includes analysis tools for structural elements: plates, stiffened panels, beams and columns, and bonded and mechanical joints. ESAComp focuses on the conceptual and preliminary design of composite structures as well as detailed product evaluation using ANSYS Mechanical and other analysis software. Engineers can study constitutive relations and hygrothermal behavior of laminates, for example, and compare laminate lay-ups with respect to strength and other design requirements. Input checks help guarantee that analyses are not performed with inadequate data.

Users can run preliminary design checks to ensure that columns do not buckle, plates withstand applied loads without deflecting excessively, pressure vessels carry specified internal pressures, joint configurations are efficient for load transfer, holes in plates do not cause severe stress concentrations, and scatter in material properties does not cause unexpected problems. The initial solution obtained gives a starting point and benchmark when going to finite element analysis (FEA) of the full structure, after which post-processing of the results helps survey the numerous failure mechanism possibilities.

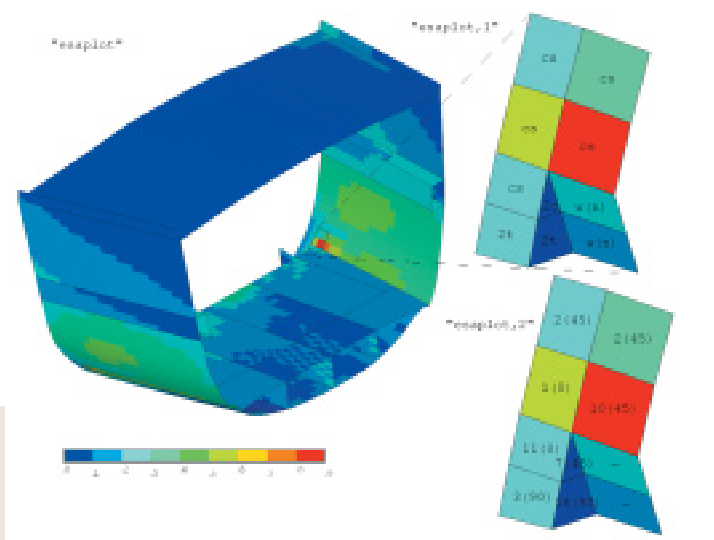
The software includes a material database of fibers and matrix materials, adhesives, sandwich core materials, and reinforced material systems from commercial suppliers.

A ply specification tool in ESA-Comp is particularly valuable in setting up the input data for various material configurations, such as a cured fiber-matrix system or a honeycomb core material. Since ply behavior is typically between isotropic and fully anisotropic, the ply specification tool utilizes material symmetry rules to help in defining the data. Ply data also can be derived from fiber and matrix data with micromechanics analyses.

With the laminate lay-up tool, laminates can be created and edited efficiently. The user has a wide range of options for performing analyses as well as selecting and combining result data. For example, several laminates, laminate orientations or failure criteria

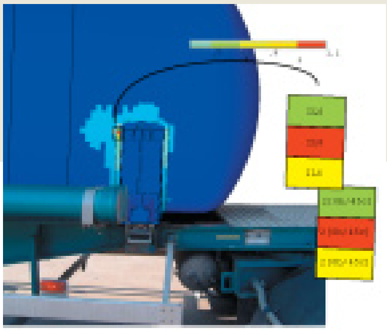
ANSYS, Inc. software and ESAComp were instrumental in the design of the 75-foot Wally-class racing yacht.

Image courtesy Johannes Schlieben, University of Applied Sciences of Northwestern Switzerland.



The esaplot viewing utility gives quick insight into the overall safety performance of the yacht design. For each composite element, a safety factor determined by the most critical layer of the section is given. Options provide detailed data on the laminate failure mode and the most critical layer, with this information overlaid on each element. Here, for example, cs indicates core shear failure, and w(n) denotes wrinkling of the face sheet. The second option characterizes the most critical layer: the stacking number and the orientation.

A liquid-hauling tank and associated structures of the truck were analyzed. The truck's tank was made of filament-wound composites and sandwich structures.



Analysis determined criticality of the interlaminar shear strength for the trailer tank structure during deceleration. Margin to safety is indicated with contours, and results of the failure analysis are overlaid on the elements.

can be selected for different types of analyses. The results display options include numeric tables, line and bar charts, failure envelopes, and contour plots.

Integration with Software from ANSYS, Inc.

ESAComp is fully integrated with ANSYS Mechanical software. ESAComp FE export supports ANSYS pre-processing. Also, the program can be launched from the ANSYS interface to perform detailed stress analysis and post-processing. ANSYS Mechanical software allows defining FE model input files in text format using specific commands, which is, in many cases, the best way to set up models; the ESAComp FE export capability fits in this scheme. The ANSYS Workbench platform supports these text format laminate definitions as well. For each part in the model tree, the user can give ANSYS commands through ANSYS Workbench command objects. Laminates can be defined with ESAComp FE export, and the definitions override the default material definitions.

Currently, the best way to simulate complex composites structures is to import computer-aided design (CAD) geometry in ANSYS Workbench as surface bodies and use enhanced contact features, automated meshing and environment commands. Then, open the simulation model in ANSYS Mechanical software and read in all laminate definitions from an ESAComp FE export file. When the geometry is imported as surfaces, ANSYS Workbench automatically uses shell 181 elements. After elements have been updated to correspond to the correct laminate definitions, the model is solved and post-processed.

Integration of ESAComp post-processing with ANSYS has been realized with the versatile ANSYS Parametric Design Language (APDL) and is used through two commands: esapost and esaplot. The most relevant data can be combined in a single ANSYS contour plot showing safety margins for the most critical failure mode, including layer failure, interlaminar shear, or sandwich core shear and wrinkling. Text labels on elements provide additional information on the failure modes or critical layers. Through this procedure, the user quickly identifies design-driving areas, since all relevant failure modes are considered automatically and clearly displayed.

ANSYS, Inc. technology and ESAComp are complementary tools used routinely in developing products made of high-performance composites. The technologies were instrumental in the design of the 75-foot Wally-class racing yacht, for example, which features a unique canting keel for balancing the moments produced by the sails. In this application, the weight of the boat was a dominant design driver; simulation tools were critical in optimizing the lay-ups and certifying the laminates of heavily loaded components, such as the chain plates and the junction between the keel box floor and the hull.

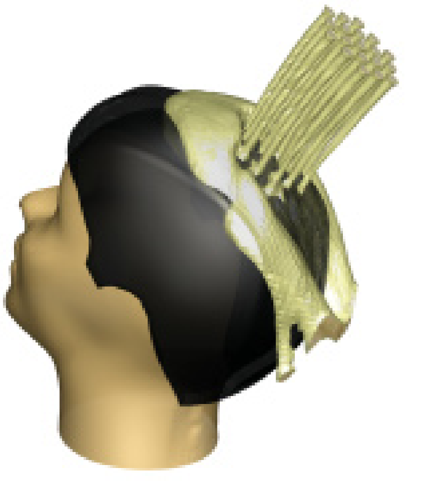
Another application involved a design project for a truck with a liquid-hauling tank made of filament-wound composites and sandwich structures. Advanced contact features and automatic meshing capabilities in the ANSYS Workbench environment were used to transform the CAD geometry of the tank support structure to the FE model. ANSYS parametric modeling features and interfacing capabilities with ESAComp were further used to optimize the design. Finally, ANSYS Mechanical software was used for validating the design against the certification authority's requirements. Processing indicated how interlaminar shear (ILS) strength of the laminate structure is a dominant design factor in the discontinuity location while the truck is decelerating.

The ANSYS ESAComp post-processing utility indicates to designers the weakest point of the structure, the weakest ply in that location and the most likely mechanism of ply failure. This information gives users valuable insight for making informed decisions on refining the design of the structure. ■

Hair Today

Product developers in the cosmetics industry can put simulation to use in performing hierarchical analyses of hair care product performance.

By Aniruddha Mukhopadhyay, ANSYS, Inc.



Simulation of water flow rinsing process using head-scale modeling

In the consumer-driven world of cosmetics, consumer experience and expectations are anything but an exact science. Qualitative performance testing, to gather information such as “Does this product increase hair’s shine?” or “Does this product spread through the hair well?” usually is achieved through subjective testing. As an alternative to such testing, product developers and researchers can use computational fluid dynamics (CFD) coupled with appropriate surface science and emulsion decomposition mechanisms for virtual testing of hair care products.

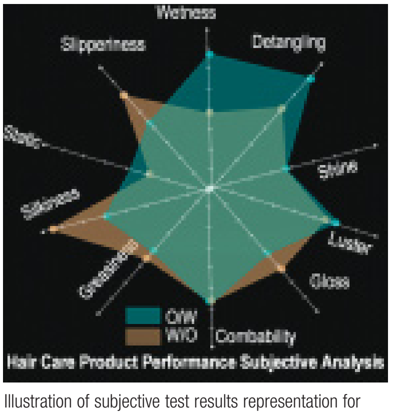
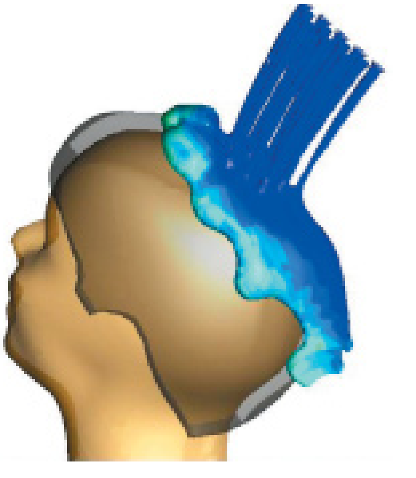


Illustration of subjective test results representation for two hair care systems

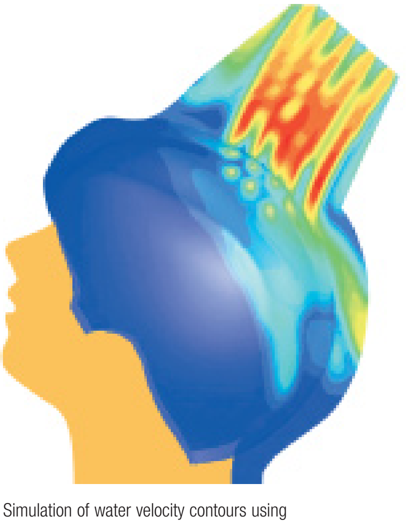
In order to mimic the subjective test procedure, a standard (baseline) hair with the standard (baseline) product can be simulated at the outset to establish quantitative correlations between subjective characteristics and chemical or fluid properties. Examples include the measurement of tackiness associated with surface tension, greasiness with viscosity and resulting glossiness with optical reflectance. As simulation progresses and correlations are developed, product developers also need to understand how and what to model on various scales.

Consistency in simulation is only as reliable as the details of physics and chemistry in the models. Within a predefined scope, simulation provides controlled test conditions. For example, a simulation-driven test procedure could be set up to begin with a known test subject, possibly developed within a “hair library” in the simulation software, of specified morphology, age, pore size, moisture absorption properties, temperature, and grease in and on the hair. The researcher then could define the environment around the test material (a sample hair assembly or tress) and apply the product making various assumptions, such as the choice to define application such that it yields approximately a uniform layer on the head. More detailed options include an applicator or a fingertip for studying the spreading and coating.

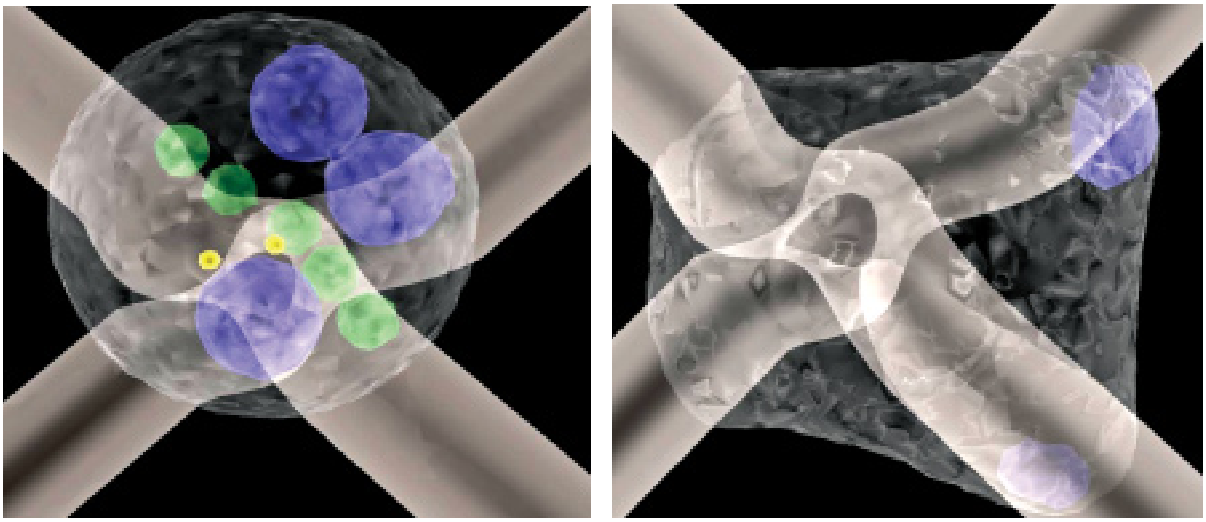
Varying size and scope of the CFD model can provide insight for behaviors that are best observed on various scales. Product application and spreading can be accurately modeled on a relatively large “head-scale,” while functions such as glosser binding, which actually occur at the hair surfaces, are best modeled at a much smaller “hair-scale.” An effective



Simulation of shampoo concentration contours using head-scale modeling



Simulation of water velocity contours using head-scale modeling



Spread of a complex oil droplet over a pair of cross-hairs: on the left, initial state in which ingredients suspended in the product's emulsion are represented as sub-droplets in the larger drop; on the right, the state after spreading has occurred

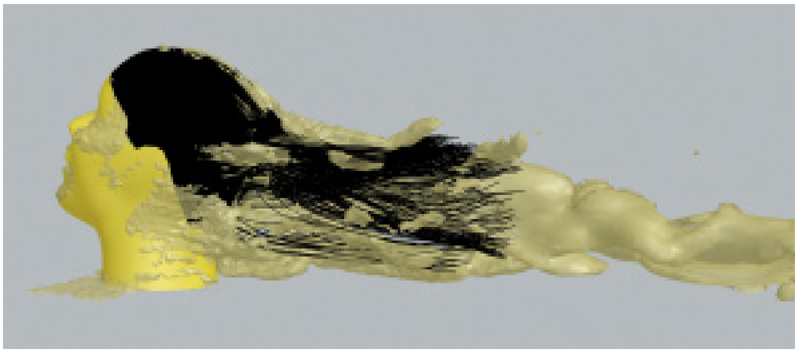
overall modeling approach involves coupling external flow with micro-phenomena near the hair surface. With this method, based on the large-scale flow conditions, the model is used to extract useful hydrodynamics data down to microscopic fluid volumes near a single hair and locally evaluate performance of various agents. This would enable gathering detailed information about the effectiveness of factors such as grease removal rate or product decomposition, which is relatively difficult, if not impossible, to consistently observe through tress-based tests.

Hair care products usually are packaged as emulsions, multi-liquid dispersions with suspended ingredients that don't segregate while stored. They are designed to dilute and break down when applied to the head with either fingers or a stream of shower. Variations in properties such as density, rheology, surface energy, chemical potential, temperature and phase-equilibrium of different immiscible and dissolving ingredients pose the design challenges to product developers. To understand the product breakdown process that occurs during application, a hair-scale simulation is required.

To examine the emulsion decomposition process, a complex, multiphase simulation is performed. A drop of the specified product is deposited at a location at which two hairs cross. The drop being modeled is about three times the hair diameter and includes suspended sub-droplets intended to represent the elemental ingredients in the emulsion. The simulation demonstrates the capillary effects of the cross-hair assembly and provides the product designer with information on the state of decomposition and spread of the product that will occur on such cross-hair configurations. Due to a variety of governing physics and the dissolution kinetics, pretreated hairs as well as conditioner ingredients greatly affect surface forces on the product drop that is being decomposed and spread.

The embedded constituents can be further defined to have their own specific material properties. For example, they could be defined as wettable, which means they stick to the hair, thereby serving as active deposition sites for various ingredients. In a case involving ingredients that are responsible for "hold" qualities, the sub-droplets could be defined as polymers that will undergo glass transition, leading to a firmer film at room temperature. This film structure will provide added elastic strength for the hair strand and evolve as a hold quality. One complexity for these films is that they will neither be exactly homogeneous in content nor have isotropic properties for factors such as elasticity, smoothness or thickness.

It is possible to set up a range of simulations for different starting compositions, sizes, temperatures and environmental dilutions and then to observe the final state for each distinct model. Although each simulation will predict a single resulting state, as though the product is in fact homogeneous and isotropic, a heuristic compilation of multiple simulations can together provide a more realistic statistical representation and characterization of the relative performances of various formulations. ■



Dynamic simulation of long hairs in a liquid stream

This Volvo hauler truck is equipped with a lightweight aluminum Alutip tipper bed that tilts back to unload vehicle contents. The unique semicircular bed is designed by Axis Developments Ltd.

Heavy-Duty Lightweight

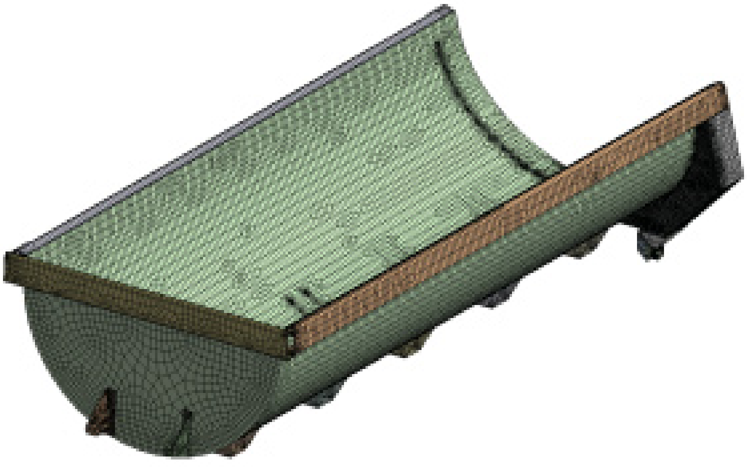
An innovative aluminum design gives a truck-body manufacturer the competitive edge in the worldwide construction industry.

By Mauritz Coetzee
Axis Developments Ltd.
Pretoria, South Africa

In developing hauler trucks, every extra pound of vehicle weight increases manufacturing costs, lowers fuel efficiency and reduces vehicle payload capacity. So Axis Developments Ltd. had an idea for making one of the largest parts of the vehicle out of lightweight aluminum: the tipper body bed that tilts back to unload soil, rock, debris or other contents.

A designer and manufacturer of trailers and truck bodies for the worldwide highway transportation and construction industries, South Africa–based Axis is known for its Alutip series of aluminum tipper beds, which weigh considerably less than comparable steel bodies. Developing these structures is an engineering challenge, however, since body strength must be maintained with aluminum material, which has different properties than steel; the amount of material must be minimized as much as possible for further reduction in weight and cost; and there is the additional desire to get new designs released quickly without numerous physical prototype testing cycles.

In redesigning an existing tipper body having a capacity of 15 cubic meters, Axis addressed these issues upfront in the design cycle with software from ANSYS, Inc. by readily evaluating stress levels for different configurations. Geometry of the existing design was imported from Autodesk Inventor, a computer-aided design (CAD) package, into ANSYS DesignModeler software, which has functions for preparing design models specifically for simulation. The engineering team used a mid-surface extraction tool in ANSYS DesignModeler to convert the solid model of the tipper body's 10-mm-thick plates to a simpler surface representation. This simplification enabled the software to model the structure with a minimal number of shell elements for greater solution speed while still retaining information on plate



Using aluminum materials saves weight but presents a new set of challenges. The first step in redesigning the tipper body was importing existing geometry into ANSYS DesignModeler software. Axis Developments' engineers modeled the truck body with shell elements and parameterized so models could be readily modified by changing a few key parameters, instead of rebuilding the entire model from scratch.

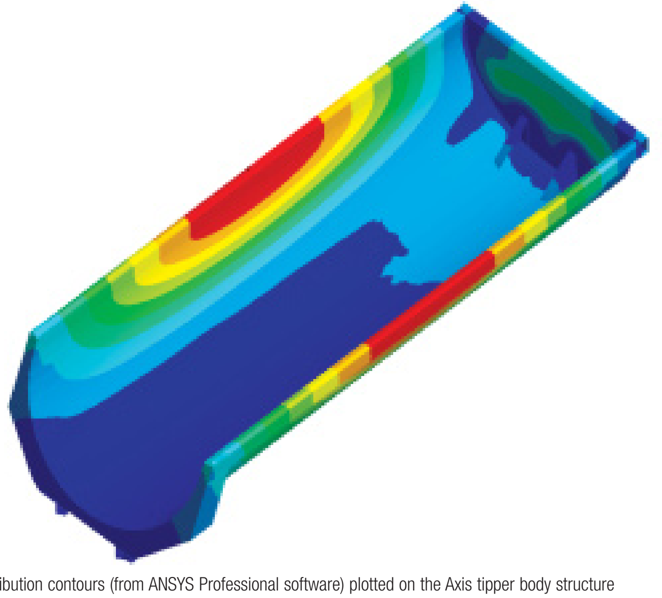
thickness throughout the simulation. Additionally, the model was parameterized so engineers could quickly modify the geometry of the model by changing only a few key parameters, instead of having to rebuild the entire model from scratch.

Next, design geometry passed from ANSYS DesignModeler to ANSYS Professional software for structural analysis. Since the two modules both operate on the ANSYS Workbench platform, transfer of data occurred with a menu pick, allowing switching between design and analysis without having to open and close different applications. In this way, Axis Developments' engineers quickly developed a mesh and performed stress analysis in ANSYS Professional software; they then were able to appropriately modify the geometry in ANSYS DesignModeler and immediately perform another analysis to ensure that stress concentrations were eliminated.

Using this approach, engineers quickly arrived at an optimal design by performing three iterations, with a total solution time of only five minutes per iteration. By experimenting with different types of designs, Axis Developments determined that the traditional support beam configuration could be replaced with a more effective semicircular design having a reinforced rib structure and end plate for additional stiffness. As the only manufacturer employing this unique design shape, the Axis bodies are easily recognizable on the road and quickly are becoming the company's trademark. Body weight was reduced 25 percent yet provided the additional strength needed for higher payload capacities — a benefit for customers and, thus, a definite competitive advantage for Alutip in the hauler truck market. The material cost savings paid for the company's software investment within only 10 truck bodies.

Axis Developments' engineers had no previous experience with finite element analysis, yet they were productive after only two hours of training. The tipper body design was completed in less than two days, which would have been unfeasible using conventional hand calculations. Moreover, the design was refined with fewer hardware prototypes. Reduction in prototype testing was a huge benefit, since these large structures are extremely time-consuming and expensive to build and test. With the success of this tipper body redesign, Axis Developments now uses a simulation-based product development approach in which all new design concepts are evaluated, "what-if" scenarios are studied, problems are fixed and designs are refined before detailed CAD work is started. ■

The authors would like to acknowledge the efforts of SolidCad (www.solidcad.co.za), a South Africa reseller of ANSYS, Inc. products that provides software and training.

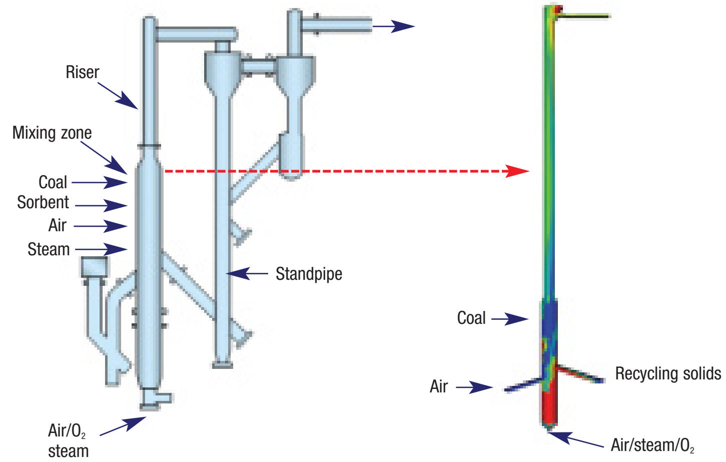


Stress distribution contours (from ANSYS Professional software) plotted on the Axis tipper body structure

Gassing Up with Coal

A two-fluid multiphase model allows for more accurate simulation of coal gasification.

By Christopher Guenther, U.S. Department of Energy, National Energy Technology Laboratory West Virginia, U.S.A., and Shaoping Shi and Stefano Orsino, ANSYS, Inc.



PSDF gasifier schematics (left) and an exploded view of the mixing zone (right) colored by contours of CO fraction

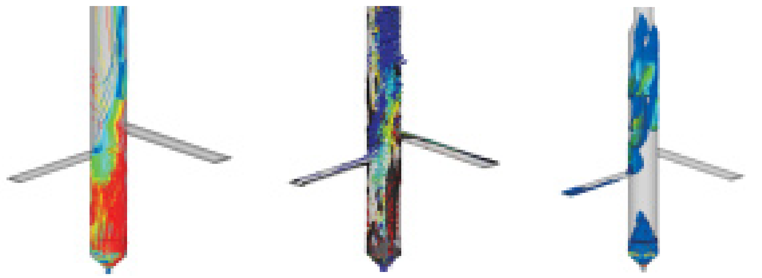
The technology of coal gasification has existed since the early 19th century. Prior to the discovery of natural gas, coal was used to produce so-called "town gas" for lighting and heat in cities across the United States and Europe. Specifically, the gasification process is used to convert any carbon-containing material into a synthesis gas, or syngas. Syngas contains mostly carbon monoxide (CO), carbon dioxide (CO_2) and hydrogen (H_2) and can be used as a fuel to generate electricity or as a basic chemical building block for a large number of applications in the petrochemical and refining industries. Gasification thus adds value to low-rank coal feedstocks by converting them into marketable fuels and products. Due to more recent technological advances, gasification offers one of the most efficient and cleanest ways to convert the energy content of coal into electricity, hydrogen, methanol and other usable forms.

Based on the mode of conveyance of the coal and the gasifying medium, gasifiers can be classified into fixed- or moving-bed, fluidized-bed, and entrained-flow reactors. Entrained-flow gasifiers are normally dilute-flow with small particle sizes and have been successfully modeled with computational fluid dynamics (CFD) using the Euler–Lagrange, or discrete phase, model approach [1]. For fluidized-bed gasifiers however, Eulerian–Eulerian (E-E), or two-fluid multiphase, model is the most appropriate approach. The E-E model treats the solid phase as a distinct interpenetrating granular "fluid" and is the most general-purpose multi-fluid model.

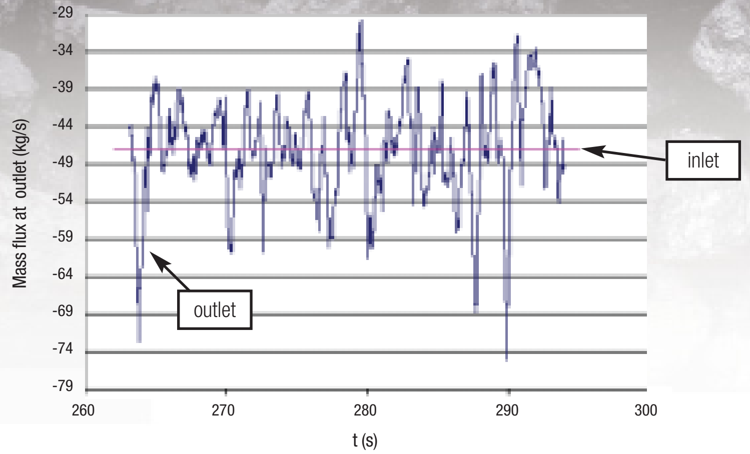
Transport gasifiers are based on circulating fluidized–bed (CFB) reactor technology and have the ability to achieve higher throughput, better mixing, and increased heat and mass transfer rates compared to other conventional technologies. CFB reactors have been an established technology in the chemical and power generation industries for years. However, new reactor designs to improve performance, reliability and safety have been slow to emerge due primarily to the lack of understanding of the complex hydrodynamics of the gas and solid phases.

The idea of describing fluidized beds and CFBs with two-fluid hydrodynamic models has existed since the early 1960s [2]. Even with today's powerful computers, numerical solutions of large-scale CFBs are rarely found in the literature, and even fewer that consider 3-D solutions [3]. Fortunately, the E-E modeling approach is one that can help researchers understand the complex interactions between the gas and solid phases and aid engineers in the design of new reactors. This approach can provide detailed 3-D transient information inside the reactor that otherwise could not be obtained through experiments due to the large scale, high pressures and high temperatures involved.

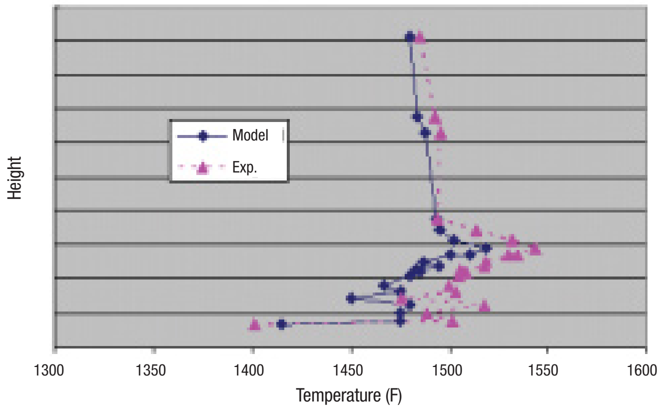
To gain more insight into the process phenomena, ANSYS teamed with the U.S. Department of Energy's National Energy Technology Laboratory (NETL) to develop different CFD models for simulating coal gasification applications.



Visualizations of the flow in the mixing zone of the PSDF gasifier for a case with air-blown and steam-enhanced lignite fuel. Included are flow pathlines colored by CO fraction (left); velocity vectors on isosurfaces of solid fraction of 0.2 and 0.3, in which the formation of particle clusters can be seen (center); and contours of carbon reaction rate (right).

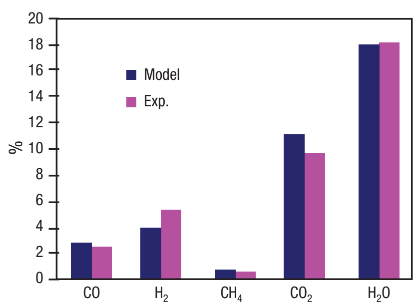


Fluctuations of the mass flux (including both solid and gas) at the gasifier outlet. The negative value represents the outgoing flow at the outlet. The magnitude of these fluctuations can deviate by as much as 70 percent around the mean of –47.12 kg/s.



Time-averaged temperature distribution along the PSDF center line as compared to experiment

Their objective was to illustrate how CFD can be used for complex large-scale geometry with detailed physics and chemistry. Using FLUENT software, the team developed a 3-D transient model of KBR, Inc.'s Power Systems Development Facility (PSDF) transport gasifier. KBR is a global engineering, construction and services company that has partnered with other companies to build a commercial transport gasification unit, based on the technology developed from the PSDF, at a 285-MW power generation facility in Florida that promises to be the cleanest coal-fueled plant in the world.



Outlet gas composition for the PSDF transport gasifier as compared to experiment

In the FLUENT simulation of the PSDF, 11 species were included in the gas phase while four species were assumed to be in the solid phase. A total of 16 reactions, both homogeneous (involving only gas phase species) and heterogeneous (involving species in both gas and solid phases), were used to model the coal gasification chemistry. The gas combustion reactions were simulated with a finite-rate combustion model. The coal reactions, including moisture releasing, devolatilization, char combustion, char gasification, tar cracking and water–gas shift reactions, were modeled with a heterogeneous reaction scheme and a set of user-defined functions. The geometry was meshed with 70,000 cells, and each simulation case was run in parallel on an eight-processor machine. Post-processing the data was done once the solution reached a pseudo-steady state, which required running the simulation until it generated physical data representing about 40 seconds of time.

The basic design of the PSDF transport gasifier included a mixing zone, which kept the recycling solids present long enough for the carbon left in the particles to react with the incoming gas (O_2, steam or CO_2). Visualizations of the system interior showed that the flow was recirculating and mixing in the mixing zone before it moved up into the riser section, and also that local conditions were very chaotic and turbulent. At the bottom of the mixing zone, combustion of the carbon present in the recycle material depleted the available O_2. Further combustion occurred as the solids moved up higher into the mixing zone. At the same time, other reactions such as CO and H_2 combustion were competing for the O_2. These exothermic reactions generated the necessary heat for the endothermic reactions, including steam gasification and CO_2 gasification of carbon.

The research team validated the overall computational results against PSDF experimental data for both bituminous and sub-bituminous coals under both air-blown and oxygen-blown conditions. The computational difference between the mass flux at the inlet and average mass flux at the outlet was only 0.1 percent, which meant that the mass was balanced well from the simulation standpoint. The team drew the same conclusion for the heat balance. For the temperature profile, the difference between the simulation and measurement was due mainly to the location of the probes relative to the center line. Despite the finding of very uneven temperature distributions at any given cross section, the overall trends of the temperature profiles were in good agreement with the measured data. ■

References

[1] Shi, S.; Zitney, S.; Shahnam, M.; Syamlal, M.; Rogers, W., Modeling Coal Gasification with CFD and the Discrete Phase Method, 4th International Conference on Computational Heat and Mass Transfer, May 2005, Paris.

[2] Davidson, J., Symposium on Fluidization — Discussion, *Trans. Inst. Chem. Eng.*, 1961, 39, pp. 230-232.

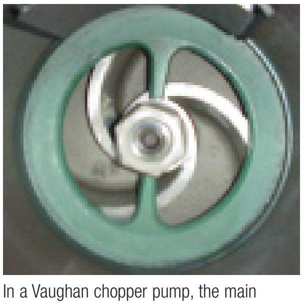
[3] Guenther, C.; Syamlal, M.; Shadle, L.; Ludlow, C., A Numerical Investigation of an Industrial Scale Gas–Solids CFB, *Circulating Fluidized Bed Technology VII*; Grace, J.; Zhu, J.; de Lasa, H., Eds.; CSCHE, Ottawa, 2002, pp. 483-488.

Chopping Away at **Solids**

Geometry of a typical casing, impeller and cutter bar assembly

CFD simulation provides a pump company with a virtual test facility.

By Glenn Dorsch and Kent Keeran
Vaughan Company Inc., Washington, U.S.A.

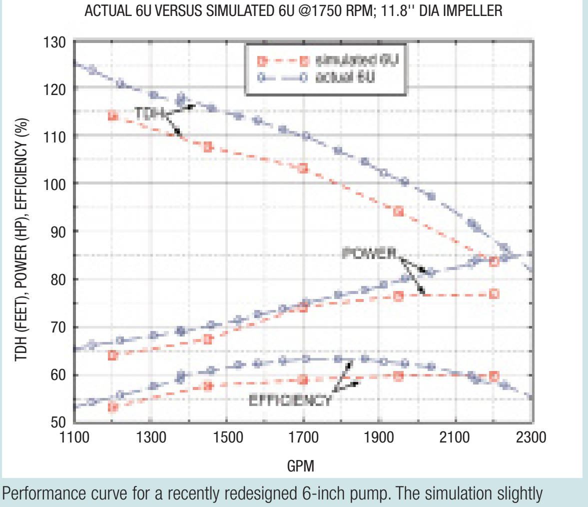


In a Vaughan chopper pump, the main impeller vanes extend all the way to the center hub of the impeller, and the suction plate includes two stationary fingers that protrude to the center of the suction opening. As the main vanes pass by the stationary fingers, a chopping action results, which macerates any solids entering the pump.

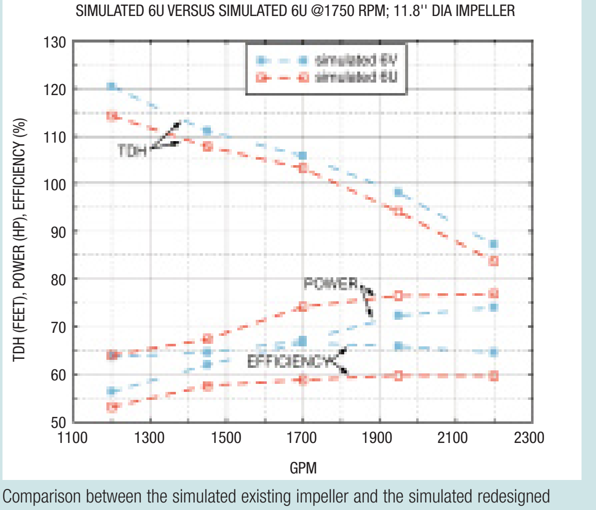
Chopper pumps utilize a chopping action between the impeller and the suction plate to break down solids that pass through the pump into smaller pieces. Vaughan Company, an established pump manufacturer in Washington, U.S.A., designs and manufactures a line of centrifugal chopper pumps. These pumps originally were designed in the 1960s for use in the local dairy industry to transport manure to and from storage tanks. Since then, Vaughan chopper pumps have been refined continually and awarded a number of patents; the company has earned wide acceptance for many applications that require solids handling. Today, Vaughan chopper pumps are used in various phases of municipal and industrial sewage treatment, food processing, and pulp and paper industries, in which the pumped liquid contains solids that need to pass through the pump without clogging or plugging.

The benefit of a Vaughan chopper pump over a typical non-clog or slurry pump is that it reduces the solids size of material passing through the pump. The unique chopping requirements and suction arrangement of these pumps make it difficult to apply standard impeller design practices in order to evaluate hydraulic performance. As energy costs continue to rise, developing more efficient pumps becomes increasingly critical for all pump manufacturers. Vaughan Company found that simulation was an effective and efficient way to approach the optimization of pump design.

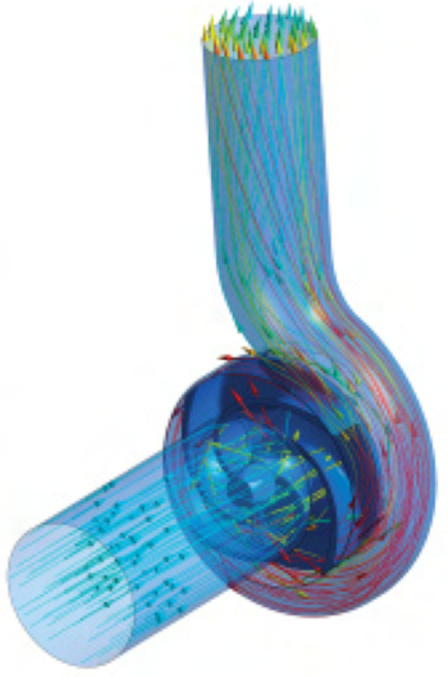
Vaughan Company's simulation process begins by importing computer-aided design (CAD) models from Pro/ENGINEER® into ANSYS DesignModeler software. The impeller domain and casing domain are meshed separately and assembled within the CFX pre-processor in which boundary conditions are applied. The ANSYS CFX solver performs the required calculations; then, results are viewed and pump performance is calculated in the computational fluid dynamics (CFD) post-processor. The ANSYS Workbench platform facilitates the entire simulation process, from geometry import through visualization.



Performance curve for a recently redesigned 6-inch pump. The simulation slightly underpredicts TDH because the geometry for the impeller and casing had to be reverse-engineered, and there were likely some differences between the model and the actual parts.



Comparison between the simulated existing impeller and the simulated redesigned impeller ensured that the redesigned impeller had TDH characteristics that were as good as the original impeller. The new design achieved an approximately 8-point increase in efficiency over most of the flow range.



Results of the simulation were visualized using ANSYS CFX-Post software.

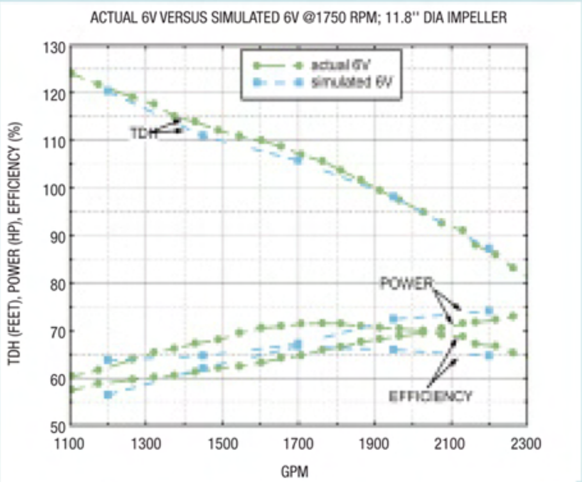
In order to optimize designs for increases in impeller performance, new impeller designs are modeled, and then their performance is simulated and compared to the simulation for the existing impeller. Originally, the impeller blade shapes were generated in Pro/ENGINEER. But the process was cumbersome, and ANSYS BladeModeler blade design software is now used. This software allows for easy generation of blade shapes to meet specifications and for the export of control curves to Pro/ENGINEER, in which the solid model is constructed.

Vaughan Company's primary interest has been in improving the efficiency of existing designs, as opposed to generating new models. A simulation is run for a given pump, and these results are compared to real performance data. In testing the real pump, a valve on the discharge side of the pump is progressively opened or closed. At each different performance point (that is, valve position), pressure and flow data are collected. These data points then can be connected to show total dynamic head (TDH, measured in feet), power (measured in horsepower) and efficiency (percent) with respect to flow (gallons per minute, or GPM). The simulation is run in a similar manner. Several simulations at various flow rates are performed on a given model. The performance data then can be extracted via the CFD post-processor, and a virtual performance curve can be constructed for that model.

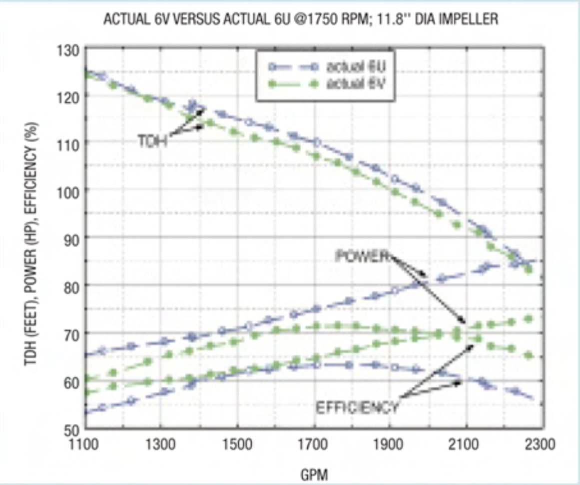
The simulated pump performance accurately predicted the actual pump performance in all seven models completed to date. Similar results are expected for the two models currently being redesigned.

The results of simulation have been very rewarding for Vaughan Company, especially when compared with fabricating and testing prototypes, which are very expensive and time-consuming. Such good performance testing correlation has been achieved between the simulation and cast impellers created from the same design that physical prototypes are no longer pursued by the company. Instead, all research and development (R&D) prototypes are modeled, simulated and optimized, then go straight to production castings.

Vaughan Company has been able to utilize ANSYS CFX simulation software, in combination with Pro/ENGINEER® Wildfire™, to effectively build and test prototype pump models at an engineer's desk. The relatively low cost of this type of R&D program allows testing of a large number of impeller blade shape variations, an approach that enables better optimization of any given design. In addition, it is a simple matter to extract a wide variety of information, including not only pressures and flows but also component forces, to better optimize the complete pump design. This optimization affects hydraulic design as well as mechanical design, such as the bearing selection via accurate impeller loads. ■



Comparison of the simulated redesigned impeller with the actual redesigned impeller test results show the TDH curves matched very well. Efficiency exceeded expectations, probably because very conservative simulations were run that slightly overpredicted the power required.



When comparing performance of the original pump and the new design, the TDH is a very close match and an 8- to 9-point improvement in efficiency was achieved across the entire flow range.

ANSYS

ADVANTAGE

Spotlight on Engineering Simulation in the

Biomedical Industry

s2 Making Life Longer and Better

s4 Turning Up the Volume

s6 Hip to Simulation

s7 Walking Pain Free

s8 Engineering Solutions for Infection Control

s10 Standing Up Right

s12 Designing with Heart

s14 Going with the Flow

s15 Battle of the Bulge

SWEET SOUNDS FROM SIMULATION

COCHLEAR IMPLANTS

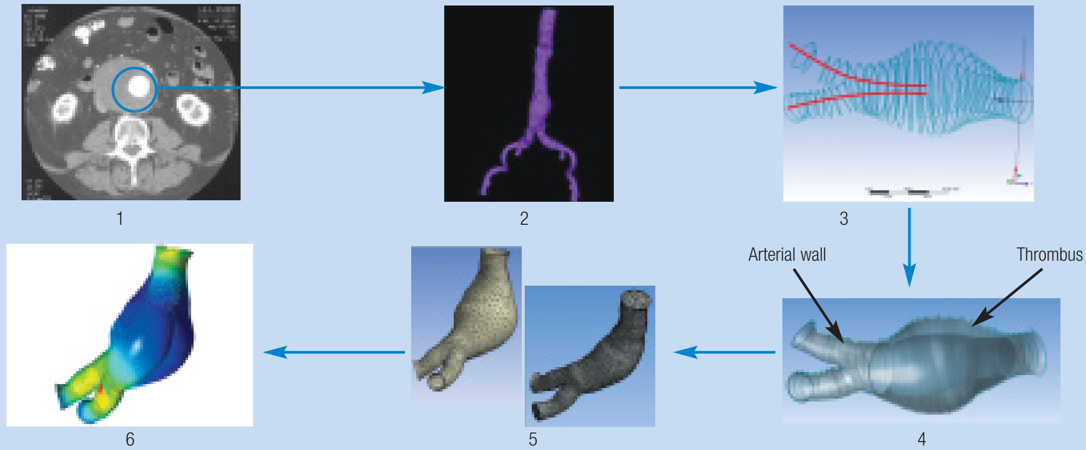
Simulation Driven Product Development: Making Life Longer and Better

The biomedical industry is emerging as a strategic user of engineering simulation.

By Thierry Marchal and Kumar Dhanasekharan, ANSYS, Inc.

Recent analyses show that leading biomedical companies around the world are continuously growing their investment into research and development (R&D), with an increase of 12.5 percent in 2006 that reached total R&D expenses exceeding $9 billion [1]. This is no surprise, given the need for advanced medical treatments and care due to a large and growing population of aging individuals, the need to find minimally invasive treatments for conditions such as diabetes and heart disease, and the increasing demand for artificial organs. As medical product innovation continues to become more complex, there is a strong emerging need for Simulation Driven Product Development, which has been seen and is broadly accepted in the semiconductor, aerospace and automotive industries.

Simulation is becoming an integral part of the product design cycle in biomedical applications ranging from prosthetics and artificial organs to endovascular techniques to surgical devices, medical equipment and diagnostic products. There are a number of reasons for such simulation to continue its entrenchment in biomedical product development. First, the advancement in technologies such as high-performance computing (HPC) is able to meet the demands of biomedical product development, allowing healthcare institutions, life science researchers and the industry to conduct large-scale simulation studies. The increasing ability to import computed tomography (CT) scans and magnetic resonance imaging (MRI) into simulation software — a process now becoming routine — makes it feasible to address in vivo device design needs (such as with respiratory drug delivery and endovascular devices), essentially enabling virtual prototyping. In addition, the integration of simulation techniques across multiphysics, from structural analysis to flow modeling to thermal analysis, is enhancing the virtual prototyping needs of the biomedical industry. For example, in studying aneurysms, ANSYS simulation tools have been used to import CT scans into the simulation



Simulation Driven Product Development is being applied regularly in the biomedical industry. This aneurysm study was performed within an integrated environment to analyze coupled fluid flow and structural simulation. The steps are: 1) CT scan; 2) segmentation from scans to extract branches; 3) cuts are written in form of splines; 4) creation of solid geometry composed of arterial wall/thrombus and automatic creation of fluid volume from the solid geometry; 5) independent mesh for each simulation technique (flow modeling and structural modeling); and 6) coupled fluid and structural model with model setup, analysis and post-processing in a single environment.

environment, allowing researchers to study a structural analysis of the weakened arteries along with the flow patterns in a single virtual environment, truly creating a virtual prototype model with multiphysics, all in an integrated manner.

Another growing area is drug delivery, particularly with medicines that are released into the bloodstream or respiratory system. There is a need to better understand the process and how adjustments can be made to accelerate drug delivery to the point of highest efficacy, which then will allow healthcare companies to design better devices that administer appropriate dosages.

Similarly, orthopedic departments are paying more attention to the virtual prototyping approach brought by computed-aided engineering (CAE). Bones are critical pieces of the body, having complex, specific geometries; they are made of different materials exhibiting strongly nonlinear behavior. Until now, scientists have lacked proper, robust models that can be used to bring together, into a single simulation, characteristics as complex as poroelasticity, nonlinear viscoelasticity and linear elasticity, which are needed for an accurate description of an intervertebral disc (ID), for example. The improved robustness of existing models together with the availability of reliable material properties now provides evidence that these numerical results can bring new, invaluable information to doctors. As a result, healthcare institutions now are studying how a hip prosthesis will perform related to a comfortable walk over a long period of time as well as investigating — prior to planning spinal surgery or even designing an ID implant — whether the remodeling procedure leading to the unification of the pedicle screw and the vertebra is likely to progress smoothly. [See Standing Up Right on page s10.]

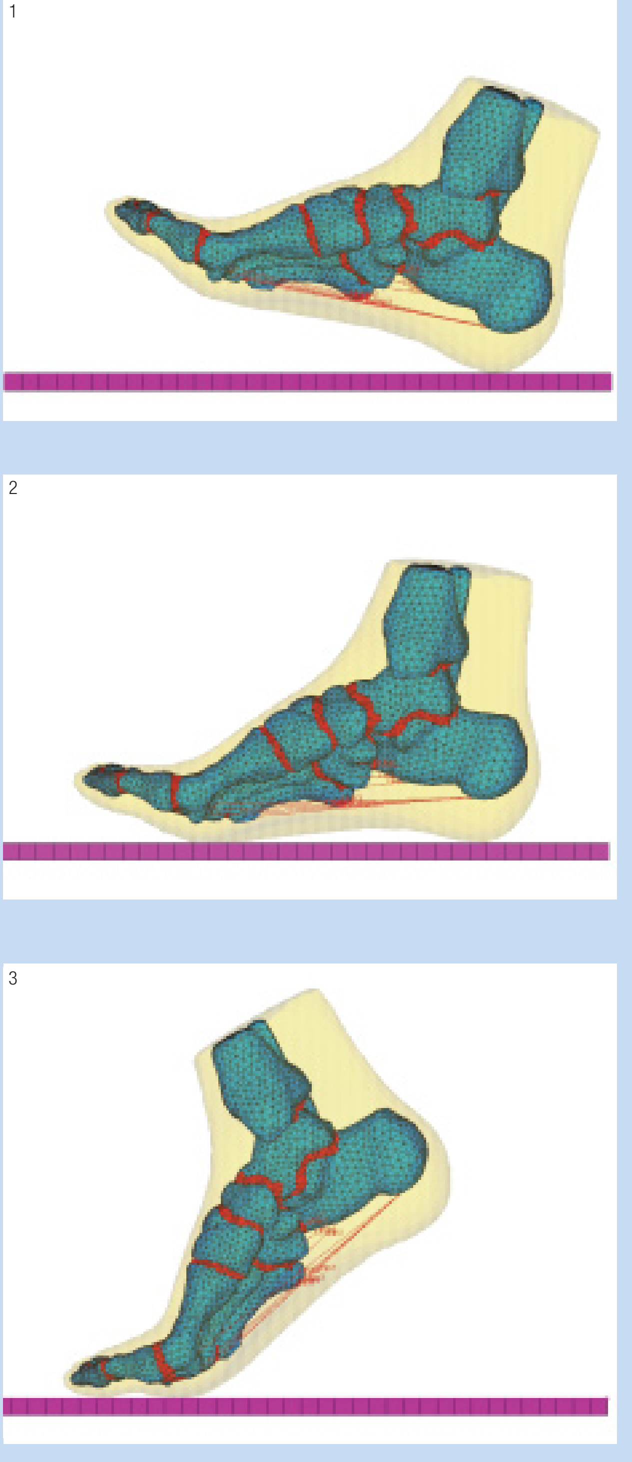
To illustrate recent concrete progress in addressing real-life problems and pain relief via CAE, this biomedical spotlight describes applications in which simulation technology has made a major difference. Both fluid flows and solid mechanics, or the combination of the two, appear in surprising applications. Some are critical to patient life or function, such as lung air flow and spine implant; others simply make life more comfortable through better ear implants and insole design.

For the future, imagine the impact of simulation to drive the development of patient-specific medicine and medical care. For example, tomorrow's surgeons may be able to take CT scans of patient physiology and use simulation to conduct virtual surgery as well as study the procedure's effectiveness as part of the overall process. This is enabled through automation of simulation along with rapid design comparisons through automated parametric studies — and it is rapidly becoming reality. The era of simulation in the biomedical world is rising. ■

1

2

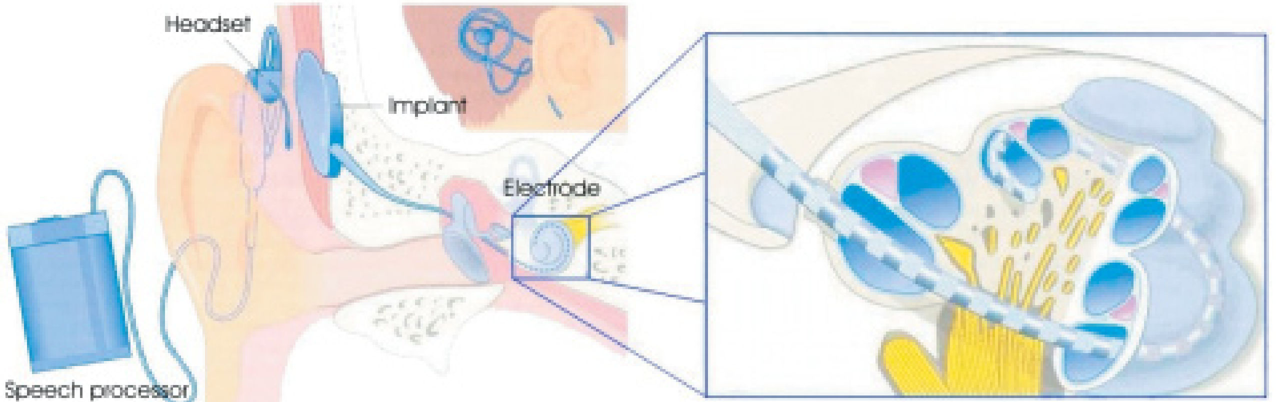
3



Proper design of a medical insole required to develop an accurate modeling of the foot at different stance phases during required ambulation: 1) the initial contact state; 2) the mid-stance state; and 3) the toe-off state. The resulting data was used to calculate the pressure and stress induced on the plantar surface as well as inside deep tissues.

References

[1] *The R&D Scoreboard 2006*, Volume 2, Department of Trade and Industry (DTI), U.K.



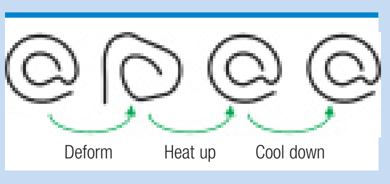
Cochlear implant diagram: implant components (left) and insertion in the cochlea (right)

Turning Up the Volume

The use of shape memory alloys offers the promise of better functioning in cochlear implants.

By Dieter Kardas, Institut für Baumechanik und Numerische Mechanik (IBNM), Leibniz Universität Hannover, Germany
Wilhelm Rust, Fachhochschule Hannover, Germany
Ansgar Polley, CADFEM GmbH, Burgdorf, Germany
Tilman Fabian, Hannover Medical School, Germany

Cochlear implants (CIs) are electronic hearing devices designed to restore partial hearing to those who are deaf or severely hearing-impaired. The devices consist of three external and two internal components. The external device comprises a microphone that picks up sounds from the environment, a speech processor and a transmitter. The internal components include two surgically implanted devices: a receiver that works with the transmitter to convert speech processor signals into electronic impulses and an electrode array that uses those signals to stimulate the auditory nerves within the ear. One of the traditional limitations of the electrode array is the inability to achieve optimal depth of insertion into the cochlea, the auditory portion of the inner ear. A German team including CADFEM GmbH, the Hannover University of Applied Sciences and Arts, and the Leibniz University of Hannover has found that an improvement might be possible using shape memory alloys (SMA).



Demonstration of one-way shape memory effect, from left to right: initial shape of a component, deformed shape, shape on warming, shape on cooling after warming

Shape memory materials display distinct thermo-mechanical behavior. In the case of shape memory effect (SME), a body that has undergone plastic deformation will return to the original shape or form that it had prior to deformation by heating it above a critical temperature. After being heated and returning to its original form, a shape memory material will not change back to its deformed shape if cooled. This phenomenon can be observed in many shape memory alloys, specifically nickel-titanium (Nitinol), which has a wide range of applications in the automotive and aerospace industries. In addition, due to its high biocompatibility, high resistance to corrosion and, above all, the thermal-induced SME, Nitinol is very useful in the field of medical engineering.

In the case of the CI, the research team thought that by taking advantage of the thermally induced shape memory behavior of Nitinol, greater implantation depth for the electrode array could be achieved. The concept was to design an SMA component whose shape matched that of the cochlea. Prior to the insertion process, the component would be deformed pseudo-plastically, and then, relying on heating from the body itself, it would return to its original form during implantation. To pursue this idea, implant simulations that accounted for the pseudo-plastic deformation and shape memory behavior were carried out using ANSYS Multiphysics tools.

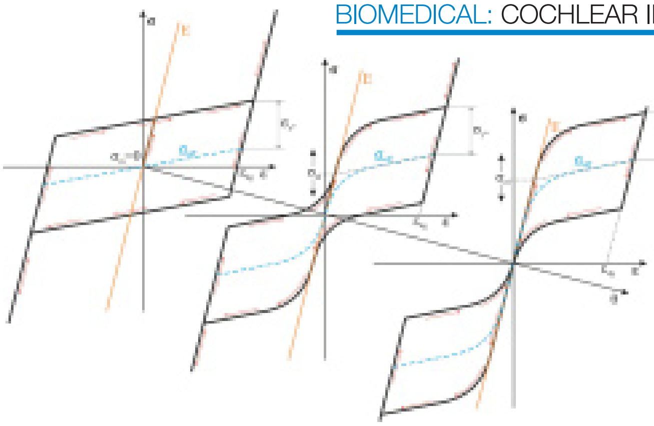
For these simulations, the team created a material model for SMA and implemented it in ANSYS Multiphysics via user-interface USERMAT for three-dimensional finite elements. The phenomenological material model was developed using stress–strain–temperature data for SMA and was based on a linear kinematic hardening model. The stress–strain behavior of shape memory materials, which is highly nonlinear in nature and varies with temperature, was incorporated into the simulation with the addition of a temperature-dependent scalar parameter: the middle stress σ_m.

The shape memory stress–strain curve differs from the standard linear kinematic model in that the shape of the stress–strain hysteresis — which one gets by periodically changing force direction — is ripped in a manner that varies with temperature. Shape memory alloys exhibit pseudo-plasticity at a low temperature range and pseudo-elasticity at a high temperature range. These temperature ranges depend on the percentage composition of nickel and titanium; generally both are equiatomic, which means that the rip of the curves increases with increasing temperature.

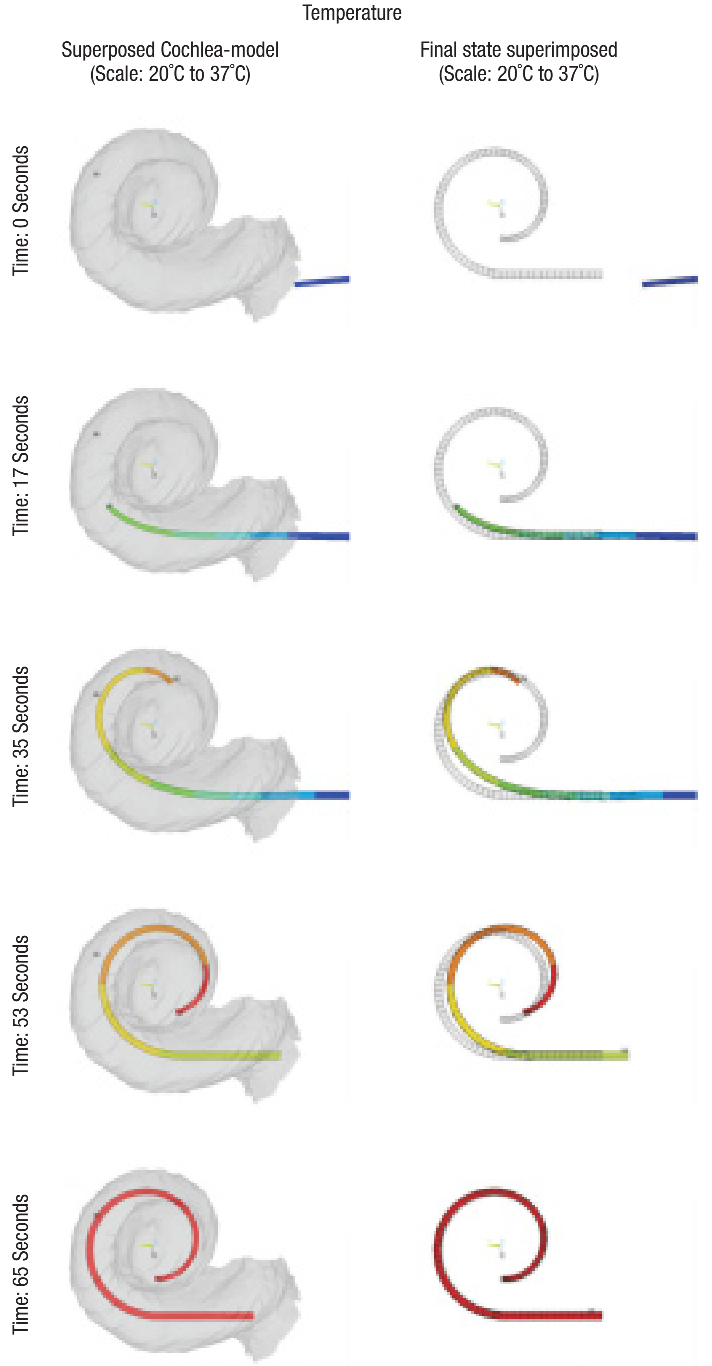
The degree to which the curve is ripped is determined by the mentioned middle stress, σ_m. If σ_m is set to zero, then the hysteresis experiences no rip, and pseudo-plasticity can be represented. If σ_m is set to a higher value than the so-called amplitude stress σ_{y^*} (half value of the distance from upper flow curve to lower flow curve), pseudo-elasticity can be represented. The actual value of the middle stress was determined using experimental data taken at various temperatures. In order to obtain a smooth, nondiscontinuous representation of the flow curve, a tanh-function was included in the equations that describe the offset/rip behavior as a function of σ_m.

By incorporating this offset-function σ_{off} (tensor-function of order two) into the material model, the shape memory behavior was effectively captured with only two sets of material constants: one set for pseudo-plasticity and another for pseudo-elasticity. ANSYS Multiphysics software itself interpolates between these parameter sets to provide the material constants for the actual temperature. With this technique, it was possible to reproduce any intermediary state between pseudo-plastic and pseudo-elastic stress–strain behavior.

By including this shape memory behavior, the CI development team was able to simulate implantation of a shape memory cochlear implant (SM-CI) into the cochlea. The results of a 65-second simulation of the implantation process supported the idea that the temperature of the human body could have enough of a thermal effect on the array that, when implanted, it could return to the original shape: that of the cochlea. These findings support the possibility of a solution that can provide deeper implantation and, thus, better functionality for the CI. ■



Pseudo-plasticity $\sigma_m = 0$ (left) and pseudo-elasticity $\sigma_m > \sigma_{y^*}$ (right). The middle stress (σ_m) rips the shape memory alloy stress–strain hysteresis as temperature increases.



Time-spaced results of the implant simulation for a shape memory cochlear implant. The red color indicates that body temperature has been reached by the implant.
Cochlear geometry data courtesy Hannover Medical School, Dr. Omid Majdani.

Hip to Simulation

Evaluation of designs for a hip replacement prosthesis overcomes physical and scientific limitations.

By Joel Thakker, Integrated Design and Analysis Consultants, U.K.

X-ray of a hip showing a prosthesis, including the socket, ball and stem. Image courtesy DePuy Orthopaedics, Inc.

Hip replacement surgery involves replacing the damaged or diseased ball-and-socket joint configuration with artificial parts. During surgery, a cup or hip socket — a dome-shaped shell/liner — is implanted into the acetabulum portion of the pelvic girdle after the bone has been hollowed out using a grater. The thigh, or femoral, portion of the hip replacement prosthesis is composed of a ball, which acts like a bearing where it fits into the cup and is attached to a stem that further attaches to the femur. The Duraloc® uncemented acetabular hip socket, a replacement cup developed by DePuy Orthopaedics, Inc., in the U.K., uses an interference fit to hold the socket in place in the hip bone. To assist DePuy in the design of the Duraloc product, Integrated Design and Analysis Consultants (IDAC) used ANSYS Mechanical software to develop parametric models that are used to establish both the necessary implantation and disassembly forces for variations of the replacement joint.

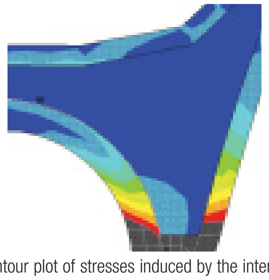
The Duraloc® uncemented acetabular hip socket is made from titanium and has a porous coated shell.

IDAC performed a two-dimensional analysis on the cup assembly in order to model the force required to remove the socket axially. A three-dimensional model was used to analyze rotational removal of the joint, since a two-dimensional case would not represent the behavior fully. The ANSYS Mechanical simulation used nonlinear contact elements in the prosthetic hip socket and accounted for friction between the cup and bone. In all analyses, the implant cup was modeled in titanium while the bone was treated as an anisotropic material.

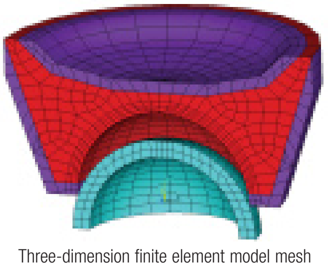
For both analyses, IDAC created parametric models in order to evaluate different bone and implant cup geometries, material properties and boundary conditions. The assembly conditions involved inserting the cup into the bone to overcome interference, allowing the frictional effects to hold the cup in place, and subsequently removing, either axially or rotationally, the cup from the bone to establish disassembly loads.

This form of modeling allows DePuy to evaluate different configurations of implant design numerically rather than by physical testing, which is time-consuming and expensive in comparison. Physical testing is limited as real bone materials are not highly available. Some synthetic and naturally occurring materials can be used, but their material properties do not precisely match that of human bone materials. Numerical modeling allows DePuy to view detailed stress and deflection distribution plots and load versus time history plots that cannot be created easily from physical tests. Comparisons between the results obtained through simulation and those obtained from previous testing reveal a close correlation.

As a result of this study, DePuy has used this type of design evaluation in other orthopedic implant products, including artificial knee joints. ■



Contour plot of stresses induced by the interference fit between the prosthesis and the bone; the areas colored in grey illustrate the region of the bone that could be expected to yield during the assembly process.



Three-dimension finite element model mesh of bone and prosthesis

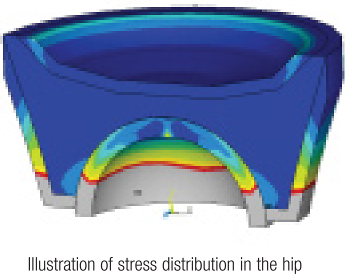


Illustration of stress distribution in the hip joint assembly after the prosthesis has been pressed into place

Walking **Pain Free**

New insoles designed with the ANSYS mechanical suite relieve pain from foot disease.

By Bum Seok Namgung, Dohyung Lim, Chang Soo Chon and Han Sung Kim
Yonsei University, Seoul, Korea

The human foot does more than simply enable mobility. Feet are an important part of the body because they bear weight, absorb shock and stabilize body structure, but they usually get little of our attention. When foot disease appears and pressure and stress exceed a given limit, pain occurs — making a person suddenly aware of just how critical a function the feet provide. For people with diabetes, subject to poor circulation and neuropathy, even ordinary foot problems can get worse and lead to serious complications.

One research project designed to benefit such patients involves developing insoles that will prevent pressure sores on the deep tissues inside the plantar surface of the foot. A team at the Institute of Medical Engineering at Yonsei University in Korea is finding new ways to gather information on the mechanical response of the foot to various insole designs. They are utilizing finite element analysis (FEA) software from ANSYS, Inc. to design new patient-specific insoles that reduce both pressure during ambulation and stress within the feet, ultimately relieving pain. The team selected the ANSYS mechanical suite because of its reliability and flexibility for handling complex and irregular geometries. Furthermore, its nonlinear, hyper-elastic models and advanced contact conditions provide a realistic alternative to experimental approaches for gait analysis.

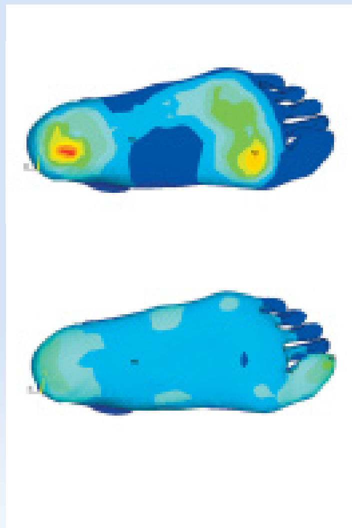
Using the ANSYS technology, the researchers first created a three-dimensional model using computerized tomography (CT) images obtained from the right foot of a subject with hallus valgus, commonly called a bunion. Commercial software, CANTIBio™ (CANTIBio, Inc., Korea) and meshing software were used to fine tune the contours of the foot.

Three geometries representing three primary states (initial contact, mid-stance and toe-off) during ambulation then were created. The simulation models incorporated two insole designs: one flat and one contoured to contact the entire bottom of the foot. Each design was analyzed at various values of elastic modulus (0.3 MPa, 1.0 MPa and 1 GPa) in order to represent a variation in insole firmness and identify which more effectively redistributed von Mises stresses on the plantar, or bottom, surface of the foot during standing.

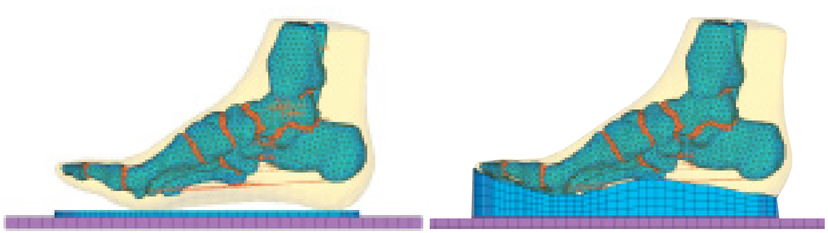
During ambulation, ANSYS software showed that high pressures first appear on the plantar surface region overlying the heel bone for the initial contact state, progresses through the middle of the foot for the mid-stance state, and finally, for the final toe-off state, is concentrated in the vicinity of the metatarsal head bone at the front of the foot. These results are in agreement with those obtained from a foot scan system used in experimental gait analysis.

The results found that stresses on the plantar surface are significantly lower with the total contact insole compared with those of the flat insole; stresses also are dependent on the insole elastic modulus. This confirms that customized design of an insole for patients with foot disease may be necessary, and the solution should include biomechanical and clinical points of view. ■

During ambulation (top to bottom), the highest pressure progressively shifts from the plantar region under the heel bone forward to the metatarsal head bone.



Von Mises stress distributions on the plantar surface of the foot using the flat (top) and total contact insoles (bottom)



Two insoles, one flat (left) and one shaped to contact the entire sole of the foot (right), were compared in this analysis to understand the impact of the geometry on foot pain.

Engineering Solutions for **Infection Control**

Simulation assists in designing a hospital ward to reduce the airborne transmission of diseases such as tuberculosis and influenza.

By Cath Noakes and Andrew Sleigh
University of Leeds, U.K.

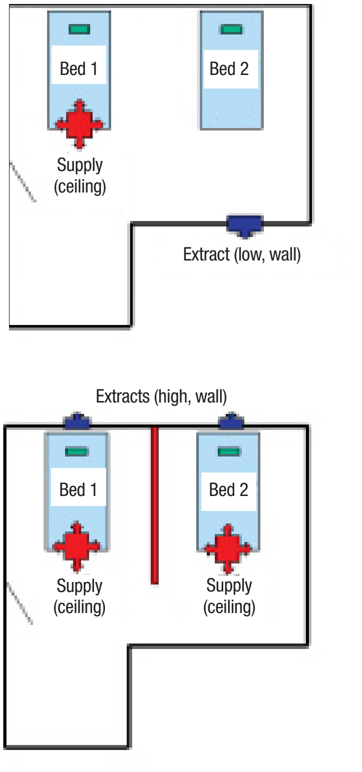
Hospital Nacional Dos de Mayo in Lima, Peru, was the site of a TB ward ventilation system redesign.

Hospital-acquired infection poses a major problem in healthcare facilities around the world. Although many infections are transmitted through hand-to-hand contact, airborne transmission also may play an important role; this is the primary mechanism for a number of infections, including tuberculosis (TB) and influenza. Airborne routes also have been implicated in the transmission of hospital-acquired infections such as methicillin-resistant *Staphylococcus aureus*, *Acinetobacter* spp and norovirus. Successful control of infection involves breaking the chain of transmission. To do so, it is necessary to understand both the mode of transmission as well as the nature of the pathogen and its behavior in the environment.

The role played by airborne transport of pathogens has been the driving force behind the research carried out by the Pathogen Control Engineering Group at the University of Leeds in the U.K. for the past 10 years. The multi-disciplinary team of engineers, mathematical modelers and microbiologists is based in the School of Civil Engineering, with strong links to clinicians at the Leeds Teaching Hospitals and to academics and scientists around the world. Originally set up to investigate ultraviolet (UV) air disinfection devices to combat TB, the group now focuses on understanding airborne transmission routes with a strong emphasis on the hospital environment. This knowledge is used to aid the development of new infection control technologies and to optimize engineering strategies to reduce the risk of disease.

The suitability of a ward ventilation system design was the subject of a recent study carried out using ANSYS CFX computational fluid dynamics (CFD) software [3]. The two-bed ward in Hospital Nacional Dos de Mayo, located in Lima, Peru, is one of a number of similar rooms housing patients with TB. Unusual to a hospital in this part of the world, the wards are mechanically ventilated. Any airborne transmission of TB within the hospital will be strongly influenced by the imposed ventilation flow. As part of a wider project researching TB transmission, led by Dr. Rod Escombe of Imperial College in London, U.K., the CFD study was carried out to examine whether changes to the ward layout and ventilation system could reduce the risk of cross-transmission between patients, staff and visitors in the hospital.

A simplified geometry represented the key features in the ward, including the basic furniture, the ventilation supply and extract vents. Isothermal airflow was modeled on an unstructured tetrahedral grid using a standard k–ε turbulence model. Supply air velocities were defined to ensure a room ventilation rate of 6 AC/h for all simulations, and a pressure of –10 Pa was imposed on the extracts to



Original room layout and ventilation system (top) and proposed new layout (bottom) showing the location of the partition between the two beds, the additional ventilation supply diffuser and the modified extract locations simulate the negative pressure that is maintained in the real facility. As the study focused on the risks of cross-infection, it was important to include a model to represent the release of infectious material from TB patients. To relate the CFD study to published outbreak data, a scalar infectious particle production variable was defined in terms of units of infectious dose, known as "quanta."

To represent a patient's production of TB bacteria, a small inlet condition was located close to the head of each bed. Scalars, representing the infectious particles produced by each patient, were introduced into the room at a constant rate of 14 quanta/hour in order to represent the typical production rate of a pulmonary TB patient.

The CFD study made it quick and easy to compare the impact of a number of proposed modifications to the ward. The original room layout with its single ceiling-mounted supply diffuser and wall-mounted extract resulted in significant mixing of TB contamination throughout the room, demonstrating the high risk of cross-infection between patients. The simple addition of a partition between the two beds yielded an immediate benefit, providing a physical barrier that limited the transfer of infection between the two areas. As a low-cost intervention, this could prove beneficial in resource-poor countries, although it may not be suitable for naturally ventilated environments. Combining the partition with a new ventilation system layout, comprising ceiling supply diffusers above the foot of each bed with wall-mounted extracts at the head of each bed, yielded the best results. Despite the ventilation rate remaining constant, the transfer of infectious material between the two beds was reduced by over 75 percent, representing a significantly reduced risk of cross-infection between patients. These findings were of immediate benefit to the architects redesigning the ward, who based the new ventilation system and ward layout directly on the study results. ■

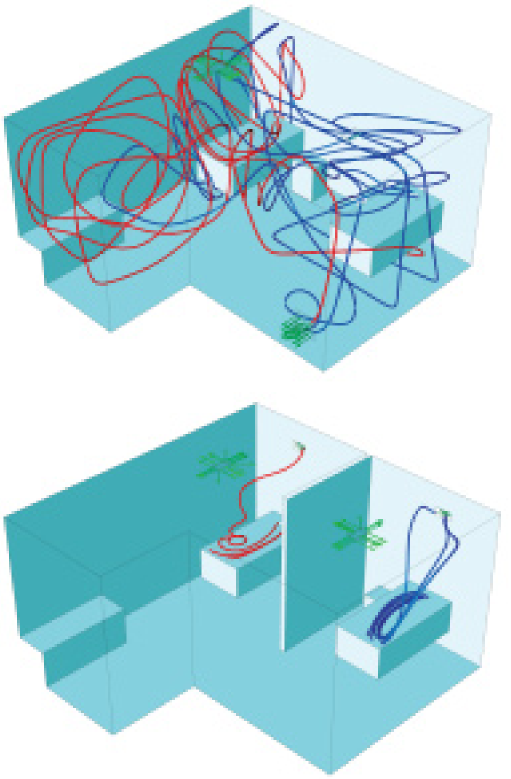
www.efm.leeds.ac.uk/aerobiology

References

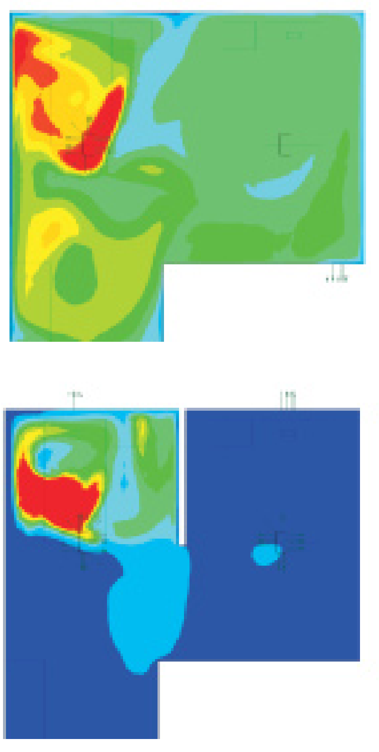
[1] Noakes, C.J.; Sleigh, P.A.; Fletcher, L.A.; Beggs, C.B., Use of CFD Modeling in Optimising the Design of Upper-Room UVGI Disinfection Systems for Ventilated Rooms. *Indoor and Built Environment,* 2006 15(1), pp. 347-356.

[2] Noakes, C.J.; Fletcher, L.A.; Beggs, C.B.; Sleigh, P.A.; Kerr, K.G., Development of a Numerical Model to Simulate the Biological Inactivation of Airborne Microorganisms in the Presence of UV Light. *Journal of Aerosol Science,* 2004, Vol. 35(4), pp. 489-507.

[3] Noakes, C.J.; Sleigh, P.A.; Escombe, A.R.; Beggs, C.B., Use of CFD Analysis in Modifying a TB Ward in Lima, Peru. *Indoor and Built Environment,* 2004, 15(1), pp. 41-47.



Streamlines originating from patients 1 (red) and 2 (blue) show how a partitioned room with modified ventilation system (bottom) more efficiently extracts contaminated air than the original room (top) does.



Contaminant concentration contours, at an elevation of 1.4 m above the floor originating from patient 1. The figure on the top has no partition, while the figure on the bottom uses a partition and ventilation systems local to each patient.

Standing Up Right

ANSYS Multiphysics sheds light on the wonders of the human spine and how to fix it.

By Stavros Kourkoulis, Satraki Margarita and Chatzistergos Panagiotis, National Technical University of Athens, Greece

The spine's intervertebral disc is exposed to a combination of compression, bending and torsion stresses.

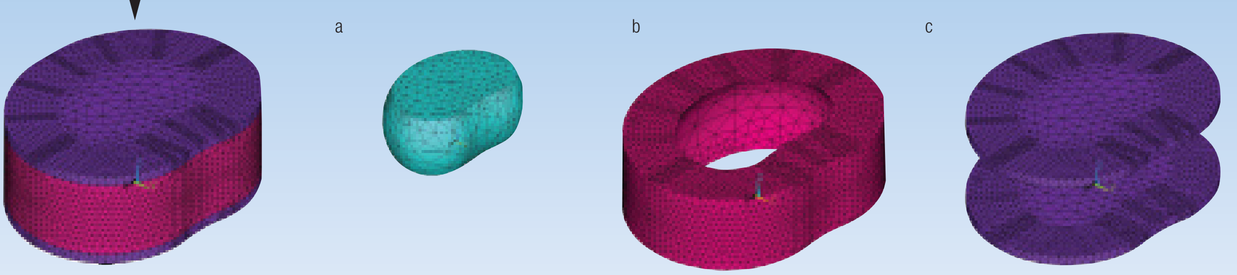
The human spine is a wonder of engineering work, one that is heavily used in daily activities. An important part of it, the intervertebral disc (IVD), is one of the most sophisticated suspension and shock absorption systems ever found. When disorders arise, back pain quickly can become a nightmare. The National Technical University of Athens (NTUA) in Greece conducted a study using ANSYS Multiphysics software that revealed some secrets of how this precious structure works, as well as ways to fix it efficiently when it malfunctions.

Simulating the Intervertebral Disc

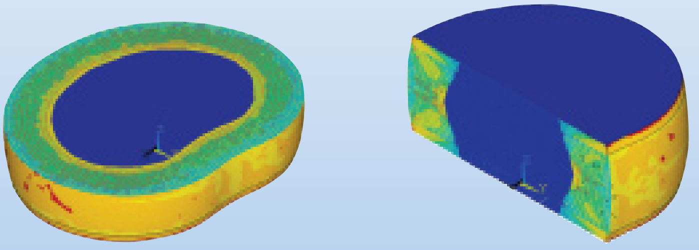
The IVD is located between the vertebrae in the spine. In performing daily activities, it acts as a cushion and therefore is exposed to a combination of compression, bending and torsion stresses. Each disc consists of the nucleus pulposus, a gel-like inner portion of the disc; the annulus fibrosus, the outer portion made of about 20 lamellae of coarse collagen fibers; and the two cartilaginous endplates, composed of hyaline cartilage, located on either side of the nucleus and annulus.

The IVD simulation model comprised four distinct volumes corresponding to the disc's regions: The nucleus was modeled as a nonlinear viscoelastic material in a kidney-like cross section; the two cartilaginous vertebral endplates were considered linear elastic bodies; and the annulus surrounding the nucleus was simulated as dual laminated shell elements whose outer surfaces were viscoelastic in nature. The study analyzed various scenarios in order to determine the contribution of each section of the IVD to the viscous character of the entire structure.

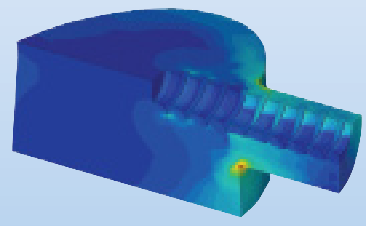
The numerical model revealed that the maximum stresses appeared in the fibers of the intermediate volumes of the annulus, in the vicinities of the endplates. The nucleus was almost stress-free, as expected due to its gel-like nature. The NTUA study also investigated the behavior of the IVD during daily activities; the results found that the reduction of disc height related to a person's 24-hour daily cycle was in very good agreement with the respective experimental data by Tyrell et al (L3–L4 discs) [1].



The numerical model of the intervertebral disc: a) nucleus pulposus, b) annulus fibrosus and c) cartilaginous vertebral endplates



The von Mises stress distribution through the center of the disc horizontally (left) and at the point of minimum vertical cross-sectional area (right)



The distribution of the Mises equivalent stress in a typical vertebra for a pull-out displacement of 0.02 mm

Studying the Surgical Remedy

Spinal stabilization using pedicle screws and rods (or plates) is one of the most common invasive treatments for spinal disorders and injuries. In this procedure, the surgical team implants screws posteriorly into a number of vertebrae and bolts them to a rod or plate. This assembly actively fixes the vertebra in place, with respect to each other, and thus stabilizes that section of the spine. After such a procedure, some serious problems can still exist. Pain in the IVD adjacent to the fixed vertebrae can occur due to failure of the spinal instrumentation, from either a fracture in structural elements or a loosening of the screws. Experimental and clinical studies alone cannot provide a complete view of the mechanical behavior of such complex structures. Numerical simulations introduce a unique tool for the thorough and parametric study of such systems.

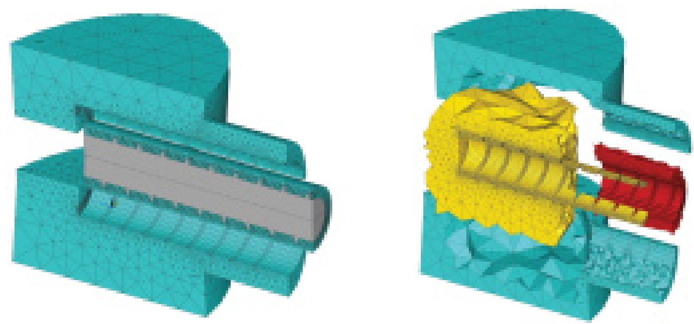
From the moment a pedicle screw is implanted into the vertebra, the bone begins to regrow around the screw. This regrowth leads to the eventual complete unification of the bone and the implant, which occurs about two years postoperatively. A fundamental requirement for the success of this procedure is the stability of the screw's fit into the bone. NTUA used mechanical simulation to investigate the influence of the vertebra structure and screw specifications — such as depth of implantation, pitch and inclination of the thread — on the value of the force required to loosen the screw from the spine.

The parametric study assumed that the vertebra consisted of cortical, subcortical and cancellous bone as suggested by measurements of bone mineral density of typical human lumbar vertebrae. The simulations estimated the force required to produce a pull-out displacement of 0.02 mm, the stress distribution onto the bone, and the contact pressure on the bone–screw interface. The results indicated that the pull-out resistance could be amplified significantly by ensuring that the screw was anchored into the regions of stronger materials located near the cortical shell. Furthermore, the parameter found to have the strongest influence on the pull-out force was the screw pitch. For pitch values varying from 2 to 5 mm, the pull-out force increased linearly by approximately 30 percent. The variation of the screw depth and the thread inclination had limited impact on the pull-out force. A comparison of the numerical results with the experimental results found them to be in very good agreement, within the tolerance of experimental error.

The main advantage of the numerical models lies in the accurate simulation of both the structure and the shape of the various portions of the biological disc or vertebra as well as of the constitutive behavior of the different materials. In order to further improve the accuracy of these numerical analyses, researchers must develop studies using models of increasing sophistication adapted to specific groups of people with morphology and properties varying with age, sex, type of activities, degenerations and other factors. ■

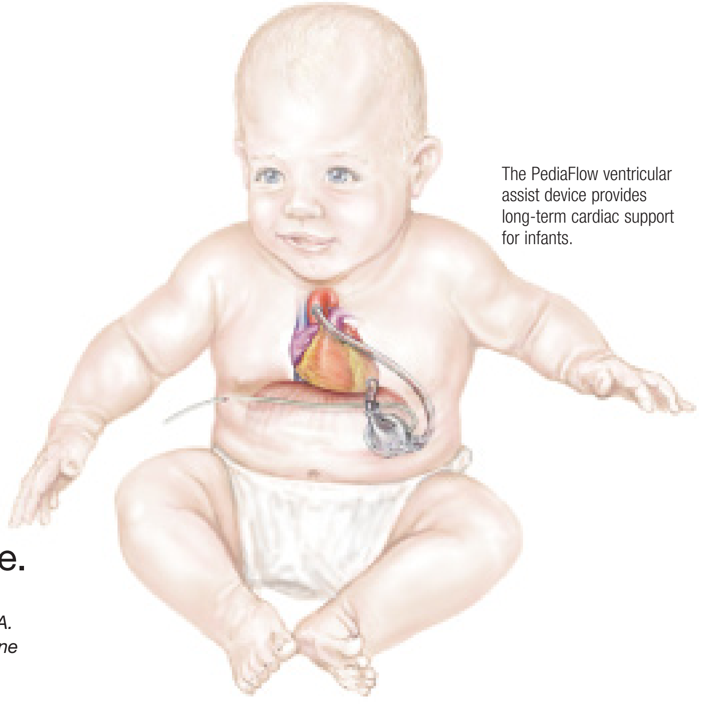
References

[1] Tyrell, A; Reilly, T; Troup, J., Circadian Variation in Stature and the Effects of Spinal Loading, *Spine*, 1985, 10(2), pp. 161-164.



The two phases of model construction: (left) the screw and surrounding bone implanted into the vertebra and (right) the regions of the vertebra (yellow: cancellous bone; red: subcortical bone; blue: cortical shell)

Designing with **Heart**



The PediaFlow ventricular assist device provides long-term cardiac support for infants.

CFD-based design optimization for a miniature ventricular assist implant can shave years off the medical device development cycle.

By Jingchun Wu, LaunchPoint Technologies, Inc., California, U.S.A. and Harvey Borovetz, McGowan Institute for Regenerative Medicine Pennsylvania, U.S.A.

An important challenge facing the design of turbodynamic ventricular assist devices (VADs) intended for long-term cardiac support is the optimization of the flow geometry to maximize hydraulic efficiency while minimizing the peak shear stress in the blood flow. High efficiency reduces the required battery size while low shear reduces the number of red blood cells that are ruptured by the pump. A pediatric heart-assist pump is particularly challenging. Due to its small size (about 28 mm diameter by 51 mm length), the design laws for adult-sized pumps do not apply, and they cannot be scaled. Therefore, the design of pediatric blood pumps must rely on modern design approaches to optimize the flow path. Computational fluid dynamics (CFD) has been widely used in the field of artificial heart pumps for the analysis of internal flow because it offers an inexpensive and rapid means of acquiring detailed flow field information that is expensive and painstaking through in vitro testing. LaunchPoint Technologies, Inc., in the United States, which developed the first magnetically levitated (maglev) heart pump (the Streamliner ventricular assist device that reached animal trials in 1998), finds that CFD is a powerful tool in the performance assessment and optimization of artificial heart pumps.

LaunchPoint has developed a CFD-based design optimization approach that integrates internally developed 3-D inverse blade design methods, parameterized geometry models, automatic mesh generators and mathematical models of blood damage with the commercial ANSYS CFX solver. The system provides rapid optimization for various types of centrifugal, mixed-flow and axial-flow blood pumps. The ANSYS CFX solver was chosen because of its robustness for computations with multiple frames of reference (MFR) (the coupling between rotating and stationary components).

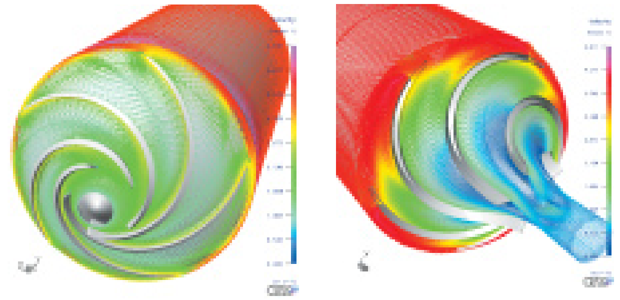
A new LaunchPoint VAD, PediaFlow™ is intended to deliver a flow rate of 0.3 to 1.5 l/min against 100 mmHg pressure rise to neonates and infants weighing 3 to 15 kg. The PediaFlow was designed with a magnetically suspended, mixed-flow style impeller with a single annular flow gap between the rotor and housing to avoid unfavorable retrograde flow and separation. The shear stress transport (SST) model, a low Reynolds number turbulence model, was selected for the turbulent flow simulation, which was justified by the representative Reynolds number of ~30,000 based on the impeller outlet diameter and the pump tip speed. Although blood exhibits non-Newtonian behavior at very low shear rates, many studies have shown that blood can be modeled as a Newtonian flow at a shear rate larger than the threshold of a 100 s^{-1}. The shear rate in the computational model of the PediaFlow is much larger than this threshold, so Newtonian blood with a constant viscosity of 0.0035 Pa-s and a density of 1040 kg/s^3 was assumed for the simulations.

The CFD-predicted velocity vectors at both the mid-span blade-to-blade region of the impeller and the vane-to-vane region of the stay-vanes show a very smooth distribution without any vortices at the nominal flow condition for the optimized PediaFlow model. As literature is replete with anecdotal evidence that recirculating flows lead to attachment of platelets to biomaterial surfaces — which in the clinical VAD setting can promote blood clot formation — reverse flows and vortices are undesirable. The CFD results found that a smooth and gradual transition in the secondary flow velocity was present at the curvature of one inflow and outflow cannula geometry. This graduation helps to prevent separation and reversal flow for the primary flow velocity. In addition, the predicted pathlines of representative particles through the entire flow region did not exhibit any vortices.

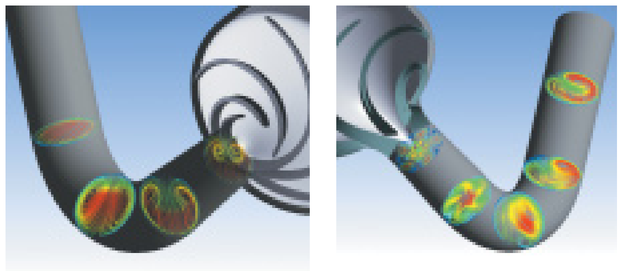
The exposure of blood elements to shear stress above a certain threshold as a function of exposure time can cause hemolysis, which actively breaks open the red blood cells; activate platelets, which can cause clotting problems; and denature proteins, which alters the proteins so they can no longer carry out their cellular functions. Thus, it is desirable to minimize the shear stress that blood passing through the pump may experience. Using the results of the CFD simulation, a plot of shear stress versus exposure time for particles passing through the pump demonstrates relative uniformity within the annular flow gap region, but it is less uniform within both the impeller and stay-vane regions. The overall mean blood damage through the entire domain of the model is divided according to the three main regions of the flow path: impeller, annular gap and the stay-vane. The analysis reveals that the hemolysis level in the annular gap region is highest, accounting for more than 50 percent of the total, while the level of hemolysis in the impeller region and stay-vane region is almost the same, each causing approximately 20 to 25 percent of the total blood damage.

CFD-based design optimization with the integration of the ANSYS CFX solver can significantly reduce the design optimization cycle from years, compared to the traditional trial-and-error methods, to just several months. It provides detailed and useful flow field information from which blood damage may be computed, and it also predicts the hydrodynamic characteristics such as the relationship of developed pressure and efficiency to flow rate. ■

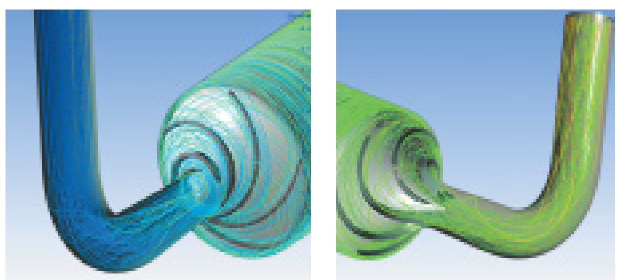
This research was supported in part by NIH Contract No. HHSN268200448192C (N01-HV-48192).



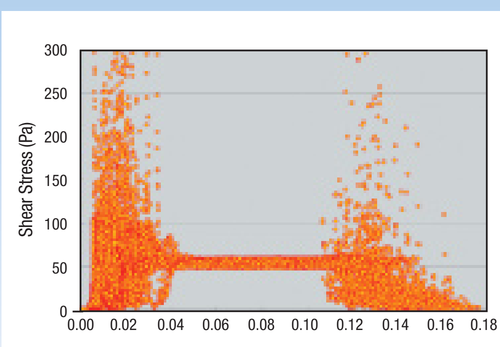
Predicted smooth velocity vectors at mid-span blade-to-blade region of the impeller (left) and mid-span vane-to-vane region of stay-vanes (right)



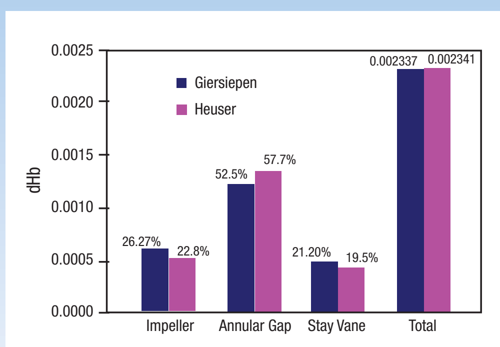
Secondary flow streamlines at sections of inflow cannula (left) and sections of outflow cannula (right)



Pathlines of particles at inflow cannula and impeller side (left) and stay-vanes side and outflow cannula (right)



Shear stress history from impeller inlet to stay-vane outlet

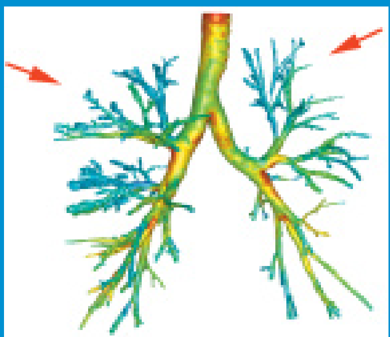


Proportion of total blood damage at different pump components under nominal flow condition

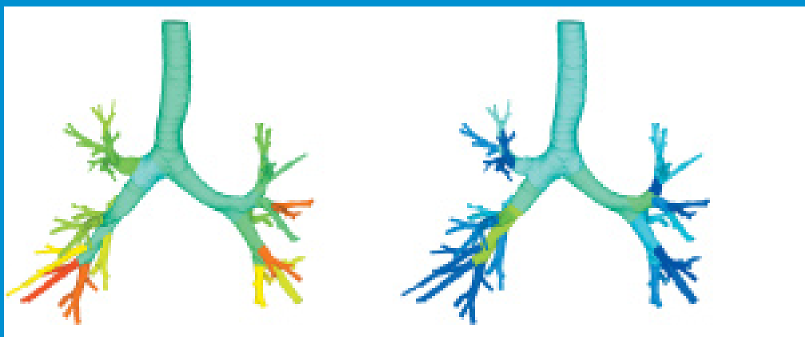
Going with **the Flow**

Functional biomedical imaging through CFD provides a new way of looking at pathological lungs.

By Jan De Backer and Wim Vos
FluidDA nv, Antwerp, Belgium



Reconstructed airway of a patient with cystic fibrosis: The red arrows indicate regions in which inflammation has restricted the airways.



Contour plots show the effect that the use of a bronchodilator has on the local values for airway volume (left) and resistance (right); red indicates high values and blue indicates low values.

Diseases such as asthma, chronic obstructive pulmonary disease (COPD) and cystic fibrosis can have a significant adverse impact on the structure and integrity of the lungs' airways. While functional magnetic resonance imaging (MRI) allows for measurement of air flow, computational fluid dynamics (CFD) provides highly detailed information of local flow characteristics and resistances. The first requirement of a patient-specific analysis is knowledge of the bounding walls of the patient's flow domain — their lung geometry. This type of information usually comes from computed tomography (CT), a scan that indicates detailed information about lung geometry because of the natural contrast between air and the lung walls. The main drawback of CT is that the resulting scan is a static image. Coupling computational analyses of air flow with the lung scan has the potential to provide significant added value to the clinical evaluation of lung function.

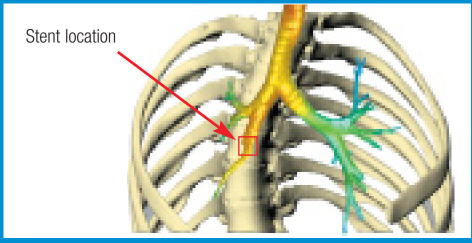
FluidDA, a spin-off of the Antwerp and Ghent universities in Belgium, has successfully developed a workflow for predicting air flow in healthy and diseased lungs using CFD. The fluid and structural dynamics company combines clinical experience and capabilities with numerical simulations to offer a variety of services to the healthcare industry.

The workflow process begins with the conversion of CT scan data into a 3-D computer model of the airway, performed with the Materialise product Mimics. FluidDA then uses TGrid software to create surface and volume meshes and FLUENT technology to simulate and examine the air flow. Flow patterns, relative pressure drops and drug delivery profiles are readily extracted from the simulation results. The resistance distribution — defined as the total pressure drop over various lung segments — also is available.

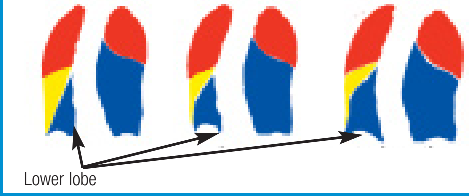
The pharmaceutical and medical device sectors also can benefit from patient-specific flow analysis as a way to evaluate performance and efficacy in a virtual patient population. In clinical studies, it is possible to analyze the effect of bronchodilating medication, which widens lung air passages and relaxes bronchial smooth muscle to ease breathing, on airway volume and flow resistance. A researcher then can begin to establish correlations between drug deposition patterns and clinical outcomes, thereby providing an indication as to why the drug does or does not work. Functional imaging also can be used to assess the placement of intrabronchial devices such as stents and valves.

Coupled with CFD, such imaging can dramatically increase insight into medical assessment and improve the accuracy of medical interventions. ■

For patients with deformation of the spinal column (kyphoscoliosis), simulation can be used to determine the site of obstruction and/or respiratory function.



Obstruction site (and subsequent location) of an intrabronchial stent, which re-inflated the blocked lower right lung lobe. Pressure contours are plotted in the airway.



An increase in the volume of the lower lobe is clear in time following insertion of a stent.

Battle of the Bulge

Rapid prototyping results in a new surgical tool to treat back pain.

By Joe Richard, HydroCision, Massachusetts, U.S.A.
Brenda Melius, consulting firm, New Hampshire, U.S.A.

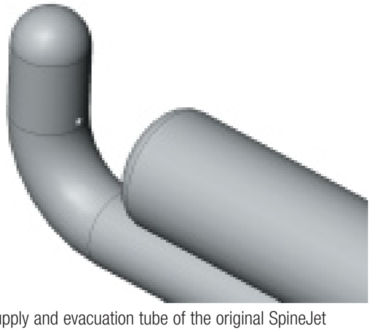
In the United States, back pain is one of the most common reasons for healthcare visits and missed work. Four out of five adults have at least one bout of back pain at some point in their lives.

A common source of pain is from a bulging intervertebral disc impinging on spinal nerves, which can cause back pain or sciatica (pain down the leg) — a condition known as herniated disc. The intervertebral disc is sandwiched between the vertebrae of the back and acts as a shock absorber during spinal movement. The disc is made of two parts: a tough outer wall called the annulus and a gelatinous inner core called the nucleus. Trauma or aging of the disc can cause the annulus to bulge.

Most occurrences of lower back pain resolve with rest and medication. For many people, though, the pain can be debilitating and last for several months to years. Such patients typically require surgery.

Minimally invasive surgical techniques offer many benefits, since traditional back surgery can cause further pain and complications. HydroCision, which develops and manufactures fluidjet-based surgical tools in the United States, used computational fluid dynamics (CFD) to improve a novel minimally invasive surgical treatment called HydroDiscectomy™.

The goal of HydroDiscectomy is to decompress the herniated disc. When performing the procedure, a physician uses a tool called the SpineJet® to remove a portion of nucleus, which debulks the disc and retracts the bulge. The device uses a high-pressure jet of sterile water directed into an evacuation tube. The jet is attuned to cut the softer nucleus but protect harder surrounding tissues such as the vertebrae and the annulus. The water jet naturally provides cutting and a low-pressure Venturi to draw the nucleus to the jet, cut it and aspirate it through an evacuation tube.



Supply and evacuation tube of the original SpineJet
Image courtesy T.G. Communications.

The SpineJet repairs a herniated intervertebral disc by removing a portion of the nucleus. The tool uses the Venturi effect created by high-velocity saline jets to cut and then aspirate targeted tissue. Image courtesy T.G. Communications.

As physicians adopt new technologies, their product demands increase. HydroCision saw CFD as a technology that could reduce development time and improve product performance. Manufacturing limitations with the existing SpineJet nozzle affected the flow divergence, directionality and alignment with the evacuation tube. By redesigning the SpineJet nozzle for better flow characteristics and greater ease of manufacture, the surgical device could be made more consistent and cost-effective. HydroCision's product development team used FLUENT software in analyzing the performance of the existing nozzle geometry. CFD simulations allowed new geometries to be designed and analyzed for performance in a matter of hours to days. Optimization of the device was faster and less expensive than the traditional method of making and testing prototypes.

The CFD model included flow simulations through the supply tube, nozzle orifice and evacuation region. CFD results helped the HydroCision team visualize critical flow characteristics such as the velocity profile, pressure distribution and flow divergence (cone angle).

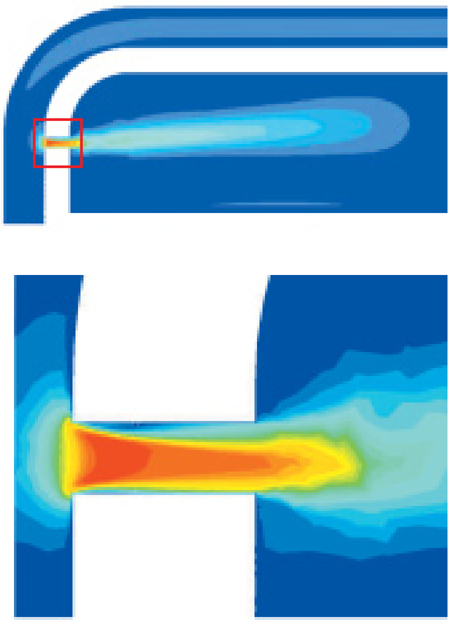
The team modeled six alternate SpineJet designs that incorporated significant changes to the nozzle and/or the supply tube. Engineers selected velocity magnitude and general jet shape as the primary means for comparing the different designs, since these two parameters are considered the most accurate predictors of overall SpineJet performance.

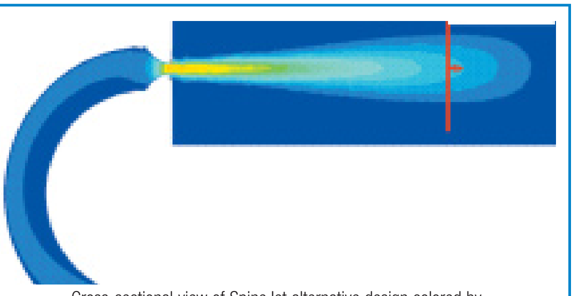
CFD results for the existing SpineJet showed the influence of a sharp-edge orifice and its location on the flow characteristic. As expected, the orifice creates a flow separation at the corner, and a vena contracta is formed. In addition, the proximity of the orifice to the 90-degree-bend in the supply tube and the additional supply tube length past the orifice create a non-uniform flow condition at the orifice entrance. As a result, the region of highest flow velocity is concentrated in the lower portion of the orifice; therefore, the flow is neither symmetrical nor well developed.

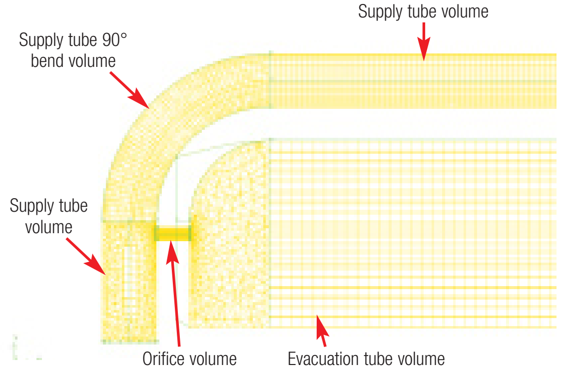
CFD results for the alternate SpineJet designs showed substantial improvement compared to the existing design. Three of the alternate configurations had 20 percent higher mass flow rates than the existing design as well as a 40 percent reduction in cone angle (flow divergence). These designs had general jet shapes that were symmetrical and well developed. They also retained higher flow velocities over longer distances from the orifice exit.

Historically, HydroCision manufactured prototypes of new geometries for testing to examine the feasibility of producing a new and improved design. Although fairly effective, this method was costly (more than $15,000 for each design tested) and time-consuming (taking approximately six months). Furthermore, testing did not always lead to a full understanding of the fluid flow characteristics that occur.

Computer modeling utilizing FLUENT software provides a different approach to the problem. The only expenses are computing and software costs; creating a CFD model and running it takes just a few days. This allows HydroCision to model and refine many designs in a fraction of the time it would take to manufacture and test a single prototype. In addition, computer simulation can yield better insights into the interactions between the geometry and the fluid flow. Finally, the graphics generated by FLUENT software help stakeholders better understand the operation of the surgical tool. ■


Cross-sectional view of all fluid volumes for original SpineJet design (top) with close-up section indicated by the red box at orifice (bottom)


Cross-sectional view of SpineJet alternative design colored by velocity magnitude



Cross-sectional view of all fluid volumes

About the Industry Spotlight

Cover image: Simulation demonstrates shape memory for a cochlear implant. Photo courtesy Cochlear GmbH. Simulation courtesy Fachhochschule Hannover – University of Applied Sciences and Arts, CADFEM GmbH and Dr. Omid Majdani – Hannover Medical School.

For ANSYS, Inc. sales information, call **1.866.267.9724**, or visit **www.ansys.com**. To subscribe to *ANSYS Advantage*, go to **www.ansys.com/subscribe**.

Managing Engineering Knowledge

Web-based solution is aimed at hosting and integrating simulation data, processes and tools for more effective Simulation Driven Product Development.

By Michael Engelman, ANSYS, Inc.

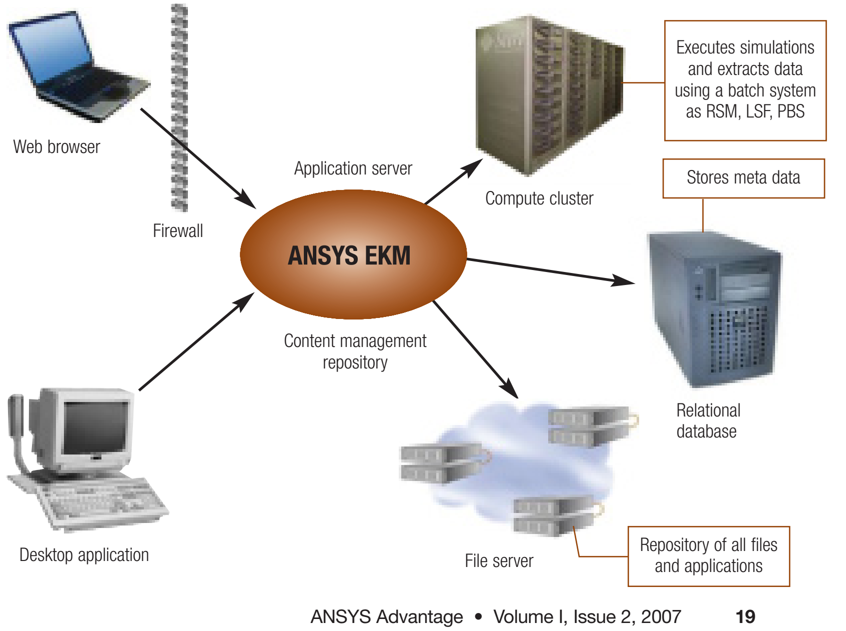
Managing simulation processes and data is a specialized subset of the larger product lifecycle management (PLM) vision. But it is often overlooked or poorly addressed, since managing simulation processes and data is more demanding than the file/document-centric approach of PLM and related product data management (PDM) systems. Simulation data is both richer and typically many orders of magnitude larger than other types of product data: It can be many gigabytes in size and can require sophisticated data reduction techniques. In addition, to extract the true value and knowledge represented by simulation data, a user must capture both the content and the context associated with the product being simulated.

The complexity of the task notwithstanding, the need to manage simulation data and processes is now more important than ever. Robust data management systems have the potential to provide significant benefits to companies by enabling users to access and reuse historical design information and expertise for speeding creation of new designs, providing ways to capture and leverage existing engineering knowledge, and addressing the problems of loss of engineering expertise and protection of intellectual property.

Process management in the context of product engineering essentially means optimizing the design workflow through more effective use of computer-aided engineering (CAE) simulation tools. This can result in a wide range of improvements, including enterprise standards for work procedures, consolidation and automation of best practices, and increased quality and reduction in errors.

Data or knowledge management applies an archiving system to allow for searches based on relevant and descriptive tags that help identify files and their contents. Thus, what is involved is knowledge management — capturing both data content and context — rather than just file or data management. This information can later be mined for insight into the how and why of a design or simulation. A managed simulation environment can address this issue by automating much of the uploading and data entry steps.

The ANSYS Engineering Knowledge Manager (EKM), scheduled for initial release this year, is aimed at meeting these challenges with capabilities for backup and archival, traceability and audit trail, process automation, collaboration, and capture of engineering expertise and IP protection. It is a Web-based design and simulation framework aimed at hosting all simulation data, processes and tools (whether in-house or commercial) while maintaining a tight connection between them. It provides three services: access management to address deployment and collaboration, process management to address integration and process automation, and knowledge management to address the issues associated with simulation data. Adding ANSYS EKM to the capabilities of the ANSYS, Inc. family of simulation products empowers organizations to create enterprise systems and achieve the goal of Simulation Driven Product Development. ■



No-Hassle Kitchen Appliance

Finite element analysis helps redesign a countertop water filter that is easier to maintain, can be injection-molded in half the time and costs a third less to manufacture than previous models.

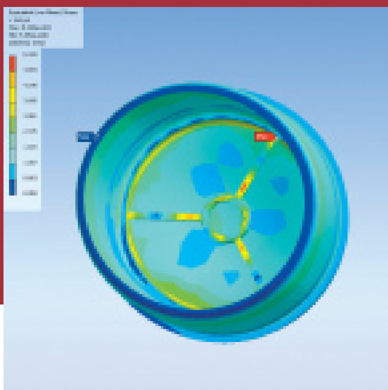
By Matthew Stein, Stein Design, California, U.S.A.

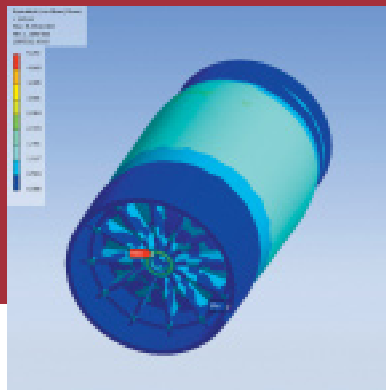
Even with degrees from top technical schools and considerable design experience, engineers find complex parts — especially ones with modern ergonomic curves — difficult to analyze with traditional handbook thermal and stress analysis. As a small one-man design shop, Stein Design completes several such projects each year that benefit from the application of finite element analysis (FEA).

The firm has used the technology to develop a wide range of plastic and cast parts, including water filtration systems, drinking fountains, medical bacteriological filters, emergency chemical drench systems and computer disk drives. Clients include Hewlett-Packard, Seagate, Plantronics and Duraflame — companies that value Stein Design for providing fast-turnaround designs that meet their unique engineering and business requirements. In the development of consumer products in particular, the firm recognizes that product aesthetics and visual impact often are critical elements in the commercial success of a product.

A countertop drinking water filter was redesigned to cut costs while making it easy for consumers to change the carbon filter cartridge and flow meter battery.

In one recent project, Water Safety Corporation of America in the United States commissioned Stein Design to complete a major redesign of their Essence™ countertop drinking water filter, an appliance intended to be attractive as well as effective in turning ordinary tap water into better-tasting, healthier water. The goal was to cut production time and cost while making it easier for consumers to change the carbon filter cartridge and flow meter battery annually. The previous housing had incorporated thick walls to accommodate the hydrostatic pressure of 150 psi required for certification by the National Sanitation Foundation (NSF), a mark recognized for its value in international trade and respected by regulatory agencies at the local, state and federal levels. These thick walls resulted in slow injection molding cycle times, excessive material usage and an undesirably expensive housing. However, arbitrarily reducing material





Left: In spite of the device's 0.27 inch-thick bottom wall and three internal ribs, stress-levels in the original design were excessive.This showed up as red areas on the ribs, as displayed in this color-coded stress plot.

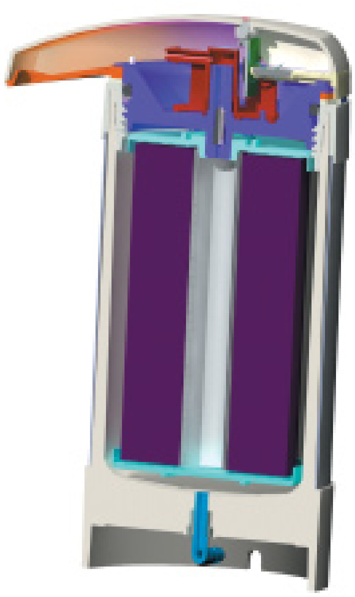
Right: After the iterative process of testing various combinations using ANSYS DesignSpace software, the final design included 12 radial ribs with a thickness of 0.125 inch.

from the overall design could potentially cause part failures leading to water damage of consumers' homes and high warranty costs. To account for these issues, Stein Design used FEA in developing a lightweight, reliable design for an appliance that would be easier for consumers to maintain.

The redesign was started by performing an FE analysis of Water Safety's existing product. When the housing was subjected to an internal hydrostatic pressure of 150 psi, analysis with software from ANSYS, Inc. showed that, in spite of its 0.27-inch-thick bottom wall and three internal ribs, stress levels of 5,360 psi were unacceptably close to the yield strength of the ABS thermoplastic material. In redesigning the housing, one of the primary concerns was reducing this maximum stress to half the material yield strength — thus providing a safety factor around 2.0 — while reducing wall thickness and injection molding cycle time for the parts.

To arrive at an optimal design satisfying these complex requirements, Stein Design performed an iterative process of evaluating different wall thickness and rib combinations. Three-dimensional models were designed in SolidWorks® software and then imported into ANSYS DesignSpace. Once the initial pressure loads and boundary conditions were set for the first model, the project geometry was updated with each new model iteration, making quick work of the analysis of "what if" scenarios. Since this was a highly cosmetic part, the maximum rib thickness was kept to a maximum of 70 percent of the wall thickness to ensure that the part would not display excessive marks where the ribs joined the outer cosmetic surface. Such indentations occur when the plastic cools and shrinks, and they are considered problematic on products that must be highly attractive in nature.

The iterative process of analyzing various rib and wall thickness combinations using FEA yielded a domed surface having a wall thickness of 0.175 inch and 12 radial ribs with a thickness of 0.125 inch minus 1/2 degree of rib draft. Rib height was 7/8 inch at the outside wall and sloped down to 1/2 inch at the inside of the rib hub. The maximum rib stress on the new design was reduced to 2,240 psi, giving a safety factor of 2.3 and exceeding the 2.0 target. At the same time, by reducing the nominal bottom housing wall thickness from 0.27 inch to 0.17 inch, injection molding cycle time was cut by a factor of two and part cost was lowered by more than a third.



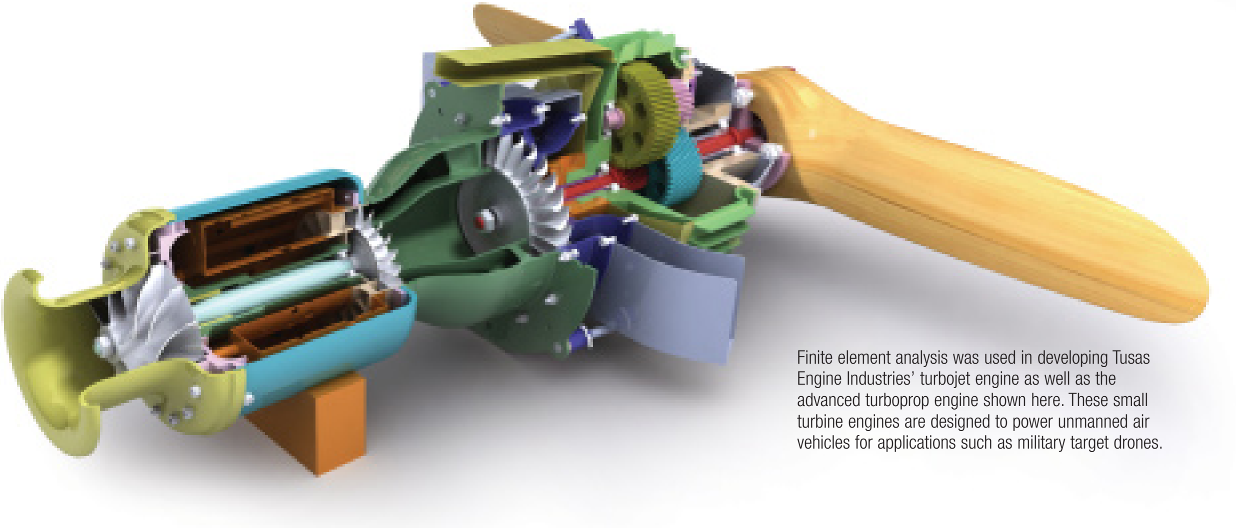
Three-dimensional models of the water filter were designed in SolidWorks and then imported into ANSYS DesignSpace software for analysis of various ribbing configurations and wall thicknesses.

ANSYS DesignSpace software is an integral part of many Stein Design projects — and part of the reason the company has succeeded in the highly competitive engineering consulting business. Small consulting firms with no full-time analysts on staff can't afford to spend a lot of time and money on training to run a complicated FEA program. Engineers who use ANSYS DesignSpace need little training to be highly productive, and the tool interfaces seamlessly with SolidWorks mechanical design software. Stein Design finds it very easy to make quick changes to the part geometry and to regenerate the ANSYS DesignSpace FEA solutions to investigate "what-if" scenarios early in the design process, when design changes have little impact on project schedules and tooling. Even though several months may pass between FEA applications, the software is designed so users can get up to speed quickly in producing meaningful results. ■

Overcoming Big Challenges for Small Turbojet Engines

In developing an impeller for a microjet turbine engine for unmanned drone aircraft, engineers used FEA to reduce stresses by 20 percent, prevent fatigue in high-speed rotating parts and study resonances in the assembly.

By Bulent Acar, Tusas Engine Industries (TEI), Inc., Eskisehir, Turkey



Finite element analysis was used in developing Tusas Engine Industries' turbojet engine as well as the advanced turboprop engine shown here. These small turbine engines are designed to power unmanned air vehicles for applications such as military target drones.

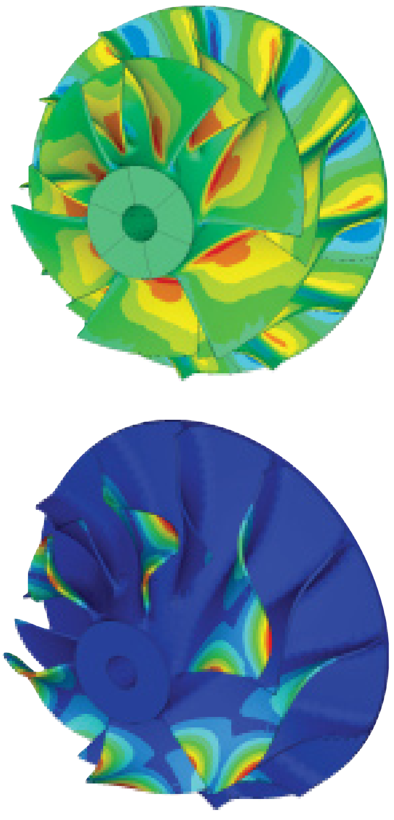
The concept of the unmanned air vehicle (UAV) is thought to have been envisioned first by Leonardo Da Vinci in 1488. The idea was not put into action until World War I, however, when radio control and gyro-stabilization technology were available to make such an aircraft feasible. UAVs became more advanced during the Second World War, when they were used to train anti-aircraft gunners and fly attack missions. Most of these early machines were remote-controlled, full-sized aircraft, but more recent technology advancements have led to the development of miniaturized UAVs, providing opportunities for cheaper, highly functional military aircraft that can be used without risk to aircrews.

One of the most challenging aspects in the development of these small aircraft is designing compact, lightweight propulsion systems for delivering the required performance. In one recent project, Tusas Engine Industries, Inc. (TEI), based in Turkey, used finite element analysis (FEA) in developing the high-speed, precision radial compressor impeller for a microjet turbine engine to be used in UAV applications such as target drones for testing the accuracy of surface-to-air and air-to-air weapon systems.

Recognized as a leader in developing and producing a range of high-quality aircraft engine parts for the worldwide aerospace industry, TEI was established in 1985 for aircraft engine assembly primarily in the Turkish region and later expanded into design, testing and manufacturing of components for gas turbine engines and other precision systems. The firm began advanced research and development activities in 1996; since then, it has participated in major international projects such as the Joint Strike Fighter (JSF) and the A400M Airbus military transport aircraft with the advanced TP400 turboprop engine.

One of the most critical parts of the Tusas TEI-TJ-1X microjet engine, the impeller compresses air entering the engine inlet to a high pressure and delivers it to the combustion chamber. Rotational speeds in the order of 100,000 rpm are necessary to achieve high compression, resulting in design challenges related to vibration, resonance, transonic flow, shock waves in diffusers and high stress levels.

Studies performed for the TEI-TJ-1X using FEA included structural analysis to determine stresses and deformation of the impeller, modal analysis of the impeller and rotor, and rotordynamics analysis of the entire assembly



to study the response of the components to rotational effects. TEI used ANSYS Mechanical software to minimize stress and deformation in their impeller designs. Various combinations of mechanical, fluid and thermal loads were considered. By using this approach, stresses in the critical regions of the impeller were reduced by 20 percent.

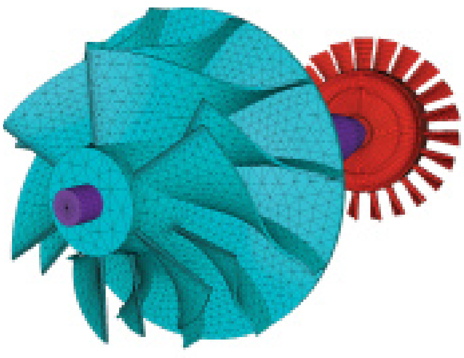
TEI engineers also used ANSYS Mechanical technology to examine the centrifugal and aerodynamic loads that can affect vibration of the blade and potential deformation of its geometry. Such deformation is a major concern in maintaining proper tip clearance — the spacing between the outer edge of the impeller blade and the inlet housing — under the range of operating conditions. If not carefully accounted for, excessive deformation could create the risk of contact between the blades and their housing.

Following the initial structural analyses that minimized stress and deformation, Tusas engineers performed modal analyses to determine dynamic characteristics of the impeller. Analyses indicated that none of the impeller frequencies coincide with any of the resonance frequencies for the engine in the operational range of impeller speeds of 100,000 to 120,000 rpm. Since rotational speed is very high, rotating parts (such as impeller and turbine) can undergo millions of cycles in a relatively very short time. Vibration characteristics of the impeller were investigated in detail to prevent high cycle fatigue (HCF) as well as contact between the impeller blade tips and the stationary inlet as a result of excessive vibration.

TEI performed full rotordynamics modal analysis on the complete assembly, including the impeller, shaft and turbine, to determine the resonant frequencies of each individual component. The most challenging aspect of the full modal analyses was defining realistic boundary conditions for the rotor's bearings and bearing housings, whose stiffnesses substantially affect modal response. In order to calculate the bearing housing stiffness values correctly and precisely, the engineering team created a whole engine model. ANSYS contact elements were used to blend the different mesh patterns of the impeller, shaft and turbine for dynamic analysis of the assembly.

As a result of the analyses, three critical frequencies were determined. The first and second frequencies affect the impeller and turbine respectively, while the last frequency has impact on the shaft. The impeller and turbine critical frequencies are especially important since they may exist in operational range and/or during startup or shutdown cycles of the engine. This led the TEI team to make design modifications, including incorporation of integrated blades. Subsequent tests validated that critical frequencies for the impeller and turbine were within approximately 10 percent of the FEA simulated values, which was acceptable. The shaft-related critical speed occurred 25 percent above the maximum operation speed. Critical shaft speeds could not be validated due to the requirement that rotational speeds were higher than operational speeds. The Tusas engineers noted that, while the test apparatus was operated at its maximum speed, there were no indications of vibration-induced problems related to the shaft.

The radial compressor impeller is one of the most critical parts of the engine. In designing the microjet turbine impeller, TEI engineers used ANSYS Mechanical technology for structural analysis to determine stresses and deformation (top) and for modal analysis in showing displacement at various harmonic frequencies (bottom).



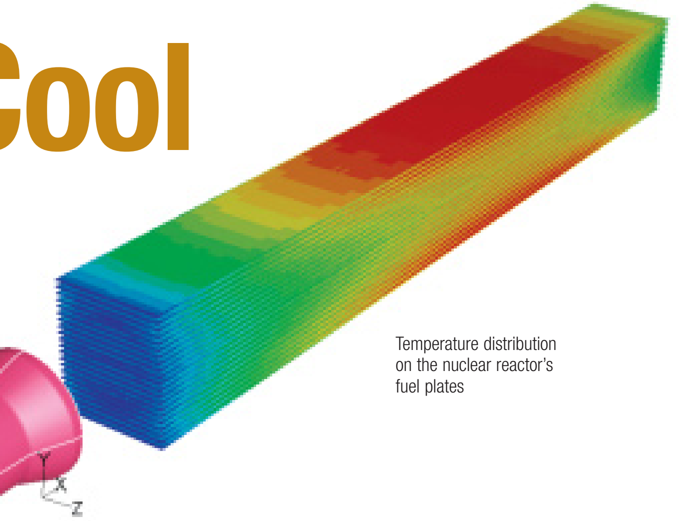
Rotordynamic analysis of the complete assembly was performed to determine resonant frequencies of each individual component, including the impeller, shaft and turbine.

Simulation in the early stages of the development cycle provided valuable insight for quickly identifying potential problems and evaluating alternative solutions. This prevented large numbers of costly and time-consuming late-stage design changes, and it enabled TEI engineers to verify the design with the minimum number of physical tests. Simulation was a critical tool in TEI's successful development of the TEI-TJ-1X microjet engine, which has successfully undergone initial performance tests and is being used as a basis for the design of an advanced turboprop engine TEI-TP-1X, now under development. ■

Keeping It Cool

Modeling fluid flow and heat transfer throughout a nuclear fuel assembly helps prevent reactor burnout.

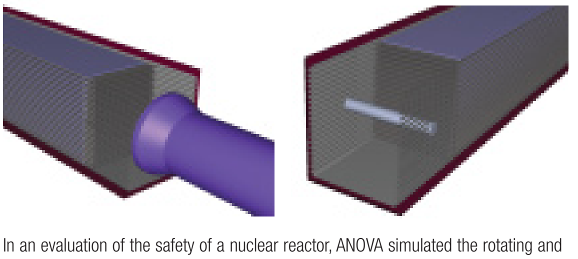
By Fahri Aglar, Turkish Atomic Energy Authority
Ankara, Turkey
Mustafa Ozer Gelisli and Emre Ozturk
ANOVA Ltd., Istanbul, Turkey



Temperature distribution on the nuclear reactor's fuel plates

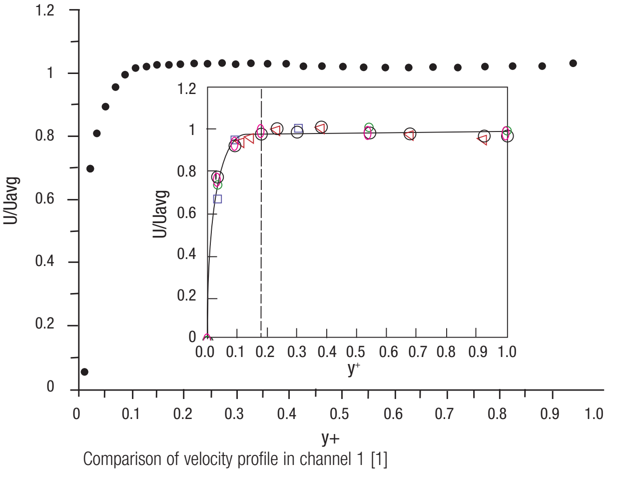
Adequate cooling of fuel in nuclear reactors has always been an important safety concern. The bulk of the radioactive inventory of a nuclear reactor is contained in the fuel elements, and, normally, their integrity can be destroyed only by excessive temperature. Insufficient cooling of the fuel leads to burnout that can cause structural damage, and subsequent leaching of radioactive fission products. Therefore, the main goal of nuclear safety strategy is to avoid an imbalance between the heat generation and heat removal in all operational states. Such imbalance could result from transients in which either the heat generation exceeds the nominal values or heat removal falls below these values. Another cause of imbalance could be the loss of coolant from accidents that result in the partial or total depletion of coolant required for the heat removal. In past investigations of the problems encountered in cooling the fuel used in nuclear reactors, thermal hydraulic studies have been carried out both experimentally and theoretically [1, 2].

As part of its work studying reactor safety, the Turkish Atomic Energy Authority (TAEK) needed to evaluate the flow and heat transfer characteristics of a material test reactor (MTR)-type fuel assembly. As a provider of advanced engineering fluid mechanics solutions in Turkey, ANOVA Ltd. performed this study to assist TAEK in its evaluation.

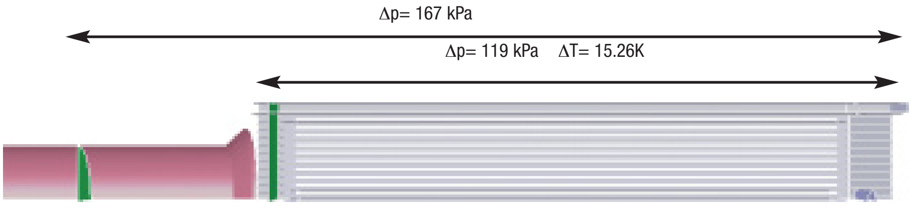


In an evaluation of the safety of a nuclear reactor, ANOVA simulated the rotating and separating flow through the cooling channel and also modeled the wall shear stress and local heat transfer coefficients. Geometry created in GAMBIT 2.3 software of a material test reactor-type fuel assembly shows the diffuser and fuel plates (left) and outlet region (right).

The fuel assembly consisted of plate-type fuel elements, with light water serving as both the coolant and the moderator. Based on geometry and boundary conditions provided by TAEK, ANOVA generated a mesh of the assembly using GAMBIT software. By assuming symmetrical flow and geometry, only one-quarter of the fuel assembly needed to be modeled. When the cross section of the fuel assembly was examined, distinctive geometries with variable cross-sectional area — such as narrow cooling channels, slender fuel elements, and sudden enlargements and contractions — could be seen. Therefore, during the GAMBIT modeling, the fuel assembly was divided into three regions: the diffuser, the fuel plates and cooling channels between them, and the outlet region. A generally hexagonal mesh was developed, and the three regions were connected through non-conformal interfaces. Accurate evaluation of wall shear stress and local heat transfer coefficients at narrow cooling channels was required, which necessitated a boundary-layer meshing scheme. Under these circumstances, and following a sensitivity analysis, ANOVA analysts created a grid containing 2 million cells.



Comparison of velocity profile in channel 1 [1]

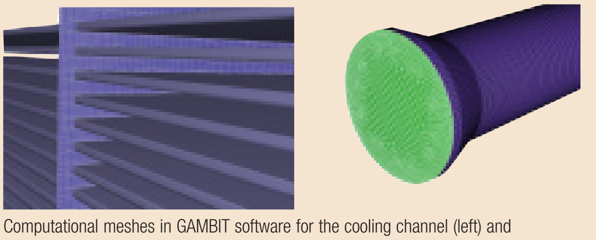


Pressure drop along the fuel assembly

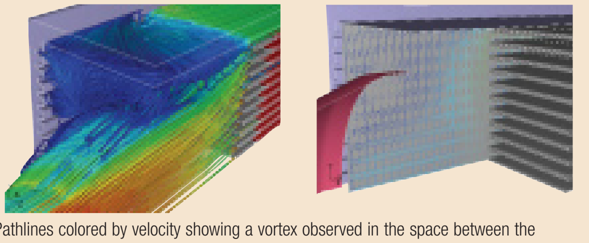
The realizable k–ε turbulence model with standard wall functions was used throughout the computations to exploit its advantage for simulating flow possessing rotation and separation. The ANOVA team performed pressure and velocity coupling with FLUENT software using the semi-implicit method for pressure-linked equations, or SIMPLE. The convection and diffusion terms of the equations of motion were obtained by cell-based discretization. One of the main input variables for the FLUENT simulation was the volumetric heat generation, which TAEK extracted from the neutronic calculations using the WIMS-D/4 and CITATION codes. The power peaking factors, which describe the local power density at the hottest part of a fuel rod, also were estimated and used to correct the volumetric heat generation term.

One of the main concerns of the simulation was the comparison of the velocities at peripheral and central cooling channels. Engineers observed that the magnitude of the velocities at the peripheral cooling channels was slightly lower than the channels located at the center of the assembly. The reason for this became apparent when the influence of the vortex observed in a region between the diffuser and the fuel plates was taken into account. The vortex and its influence are extremely important from the reactor safety point of view, and estimations revealed that this velocity reduction seemed to be negligible. The outcomes related to velocity reduction also matched those obtained from experiment [1]. Further channel-to-channel flow distribution analysis showed that the relative flow rate, evaluated as a ratio of the flow rate in the individual channel to that of the assembly, decreased from the central channel to the outermost channel. The plate-to-plate temperatures showed the opposite behavior; that is, the temperature increased toward the outer channels.

A final point of interest was that the pressure difference between the inlet and the outlet of the fuel assembly was in the acceptable range and did not cause flow instability and phase change during normal operation. The pressure drop along the fuel region was 70 percent of the total pressure drop, which was in accord with experimental data [1]. The FLUENT results thus have been instrumental in understanding the complex 3-D flow in an MTR-type fuel assembly. Such CFD simulations have contributed significantly to the design and licensing of nuclear power systems. ■



Computational meshes in GAMBIT software for the cooling channel (left) and diffuser section (right)



Pathlines colored by velocity showing a vortex observed in the space between the diffuser and plate region (left) and corresponding velocity vectors (right)

www.taek.gov.tr
www.anova.com.tr

References

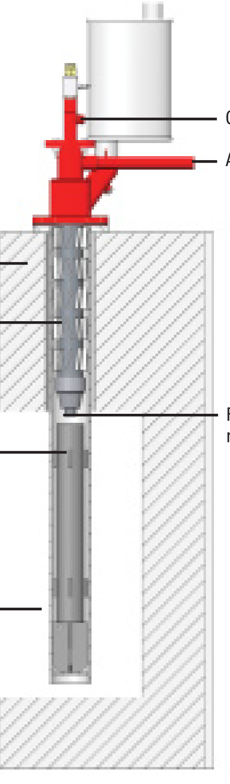
[1] Ha, T.; Garland, W. J., Hydraulic Study of Turbulent Flow in MTR-Type Nuclear Fuel Assembly, *Nuclear Engineering and Design*, 2006, 236, pp. 975-984.

[2] Franzen, F. L., Nuclear Power Plant Operational Safety — Safety Strategy and its Technical Realization, IAEA Interregional Training Course, Karlsruhe Nuclear Research Center, 1981.

The Greening of Gas Burner Design

Simulation assists in developing efficient and environmentally friendly recuperative burners used in heat-treating applications.

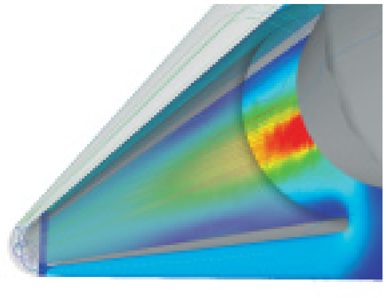
By Örjan Danielsson and Marcus Andersson
Kanthal AB, Hallstahammar, Sweden



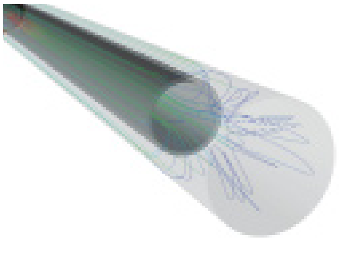
Geometry of a single-ended recuperative burner, in which fresh air and fuel are combusted in an inner tube within a burner assembly while exiting exhaust gases are recovered, passed back through an outer tube that surrounds the inner tube and used to heat the incoming fresh air in a recuperator region

Companies that depend on gas burners for heat-treating materials are challenged with adapting to tougher nitrous oxides (NOx) and carbon dioxide (CO_2) emissions rules and legislation, along with maintaining high efficiency. To address this, Kanthal AB uses a simulation-driven design process to develop innovative products such as the ECOTHAL® single-ended recuperative (SER) burner, the latest addition to the Kanthal family of heating solutions.

The key to success in delivering a low-emission, high-efficiency gas burner lies in well-defined combustion. Delivering too much air reduces heat output and increases the amount of harmful NOx produced, whereas too little air results in incomplete combustion that causes unburned residue in the form of carbon monoxide (CO) and hydrocarbons. In the ECOTHAL SER burner, fresh air and fuel are combusted in an inner tube within a burner assembly while exiting exhaust gases are recovered, passed back through an outer tube that surrounds the inner tube and used to heat the incoming fresh air in a recuperator region upstream of the combustion area. Kanthal used computational fluid dynamics (CFD) to model and optimize flow behavior, gas mixture control and combustion efficiency. CFD simulations using software from ANSYS, Inc., together with physical testing, resulted in an SER burner that had an efficiency of approximately 80 percent — 10 to 20 percent higher than conventional SER burners — while still keeping NOx levels below 50 ppm (or 20 mg/MJ).



Contours of velocity and streamlines predicted by ANSYS CFX, viewed looking into the inner tube from its inlet area



Streamlines indicate exhaust gases that are exiting the inner tube and recovered back through the outer tube of Kanthal's SER burner.

To help ensure that Kanthal provides accurate recommendations regarding procedures for maintaining proper performance, the company uses ANSYS Mechanical software to model creep. Kanthal's burner systems often are mounted horizontally, and creep, or deflection, of the tubes can affect flow characteristics and operation within the burner. The deflection rate typically is measured through physical testing in which a sample tube is placed in a furnace and the deflection is measured at specified intervals. This is a very time-consuming test that can take up to 3,000 hours, or 125 days. On the other hand, when modeling creep, existing test data is used to provide the coefficients for the creep equation that is used in the simulation inputs, and the testing process is simulated in less than a day.

The SER burner from ECOTHAL is the first in a family of five burners. The second burner in the series was designed entirely using CFD in combination with traditional computer-aided design (CAD) software. The ability to go directly from a 3-D CAD model to meshing and simulation within the ANSYS Workbench platform, and then to pass design changes back to the CAD program, greatly improved the speed of product development for Kanthal. By using simulation in conjunction with CAD tools, expensive and time-consuming laboratory testing was kept at a minimum, and development time was reduce by several months. ■

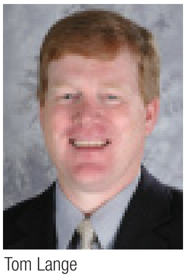
www.kanthal.com

The **Democratization** of Engineering Analysis

To compete successfully in today's business climate, Procter & Gamble makes analysis tools available to rank-and-file engineers as well as to analysts and advanced simulation experts.

By Fred Murrell and Tom Lange, Procter & Gamble Company, Ohio, U.S.A.

Fred Murrell

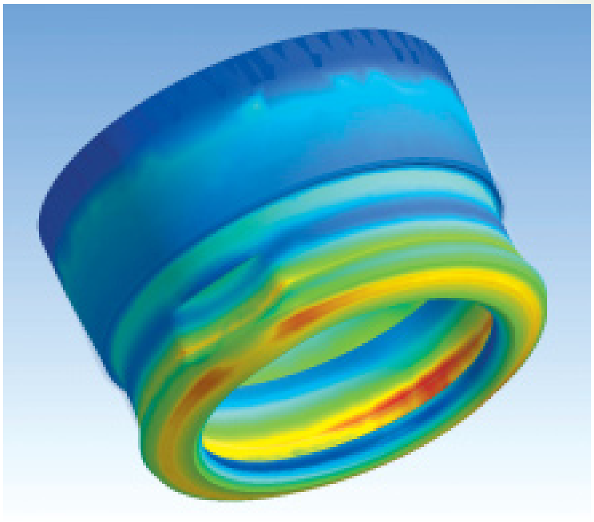


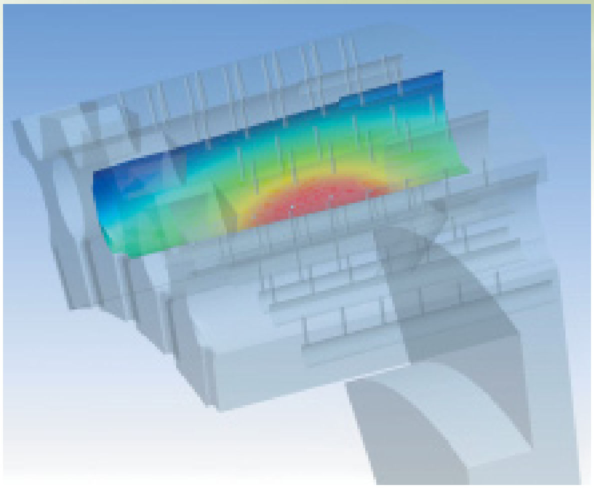
Tom Lange

Early practitioners of techniques such as finite element analysis (FEA) and computational fluid dynamics (CFD) typically were confined to industries in which the risks to human life or mission success were such that the expense could be justified. It is, therefore, no surprise that the first commercial FEA packages came from and were used by industries that could afford access to expensive computational resources — and for which a failed component could have catastrophic results.

As these techniques spread to other industries, computer-aided engineering (CAE) remained the bailiwick of the expert analyst, requiring advanced degrees and long apprenticeships to cope with the difficulties of the technique and to ensure accurate results.

The rapid and unrelenting improvements in hardware, the personal computer and low-cost cluster computing — and technology such as the ANSYS Workbench platform — has truly democratized CAE analysis. A common desktop PC has more than 10 times more computing horsepower than a high-end workstation from just 10 years ago costing 10 times that price. No longer is engineering analysis a luxury that costs many thousands of dollars requiring the services of highly trained experts.





Simulation on the ANSYS Workbench platform was used to determine stresses (left) and thermal distribution (right) in these components of high-speed equipment used in Procter & Gamble Company production operations.

The Procter & Gamble Company (P&G) is best known for its brands. Three billion times a day, P&G brands touch the lives of people around the world. The company has one of the strongest portfolios of trusted, quality leadership brands, including Pampers, Tide, Ariel, Always, Whisper, Pantene, Mach, Bounty, Dawn, Pringles, Folgers, Charmin, Downy, Lenor, Iams, Crest, Oral-B, Actonel, Duracell, Olay, Head & Shoulders, Wella, Gillette and Braun. Chances are that you have used one of these brands recently, if not today.

P&G also is well known for advertising these brands. According to *Advertising Age* magazine, P&G was the largest advertiser in the United States in 2005, spending more than $4.6 billion. Advertising, packaging, display and name recognition are aimed at what P&G refers to as the first "moment of truth," when a customer decides to purchase a product they have never used before. But, as any manufacturer knows, you won't have a customer for long if your product doesn't deliver as promised. If you fail the consumer the first time, you will not be rewarded with repeat business. P&G calls this the second moment of truth, when a customer uses the product and judges whether you have delivered on your advertised promise. This is where the science behind the brands comes into play.

All manufacturers face similar tensions — rapid innovation, keeping down costs and improved time to market. P&G is a leading proponent of CAE technologies in its drive for improved innovation. In fact, in a 2003 conference call with Wall Street analysts, P&G Chief Executive Officer A.G. Lafley stated, "We are significantly expanding capabilities in computational modeling and computer-aided engineering, so we can do an increasing percentage of product and process design through virtual simulation."

In the consumer packaged goods business, this would not have been realistic or feasible just 10 years ago. High-end CAE analysis was then the domain of experts, most likely employed in the defense, aerospace or automotive business. The expert also was armed with complicated, high-end analysis software and an expensive UNIX® workstation. Today, the ubiquity of inexpensive, fast and powerful desktop PC workstations has made the use of CAE analysis available to the rank-and-file engineer in ways unimaginable in the past.

When P&G creates new products, there are three goals — it has to fit, do what it is supposed to do and, most important, make financial sense. The company wants to make the first prototypes virtually and make the physical item only when confident it will work. In the consumer packaged goods business, companies make billions of items and sell them for a relatively small amount. Analysis allows P&G to optimize those products and processes to save a penny or two here and there. The focus is in making lots of high-quality products very quickly.

Just as the needs of individual projects vary, so do schemes for utilizing CAE. P&G has developed a three-tier approach for CAE. Tier 1 consists of a small cadre of experts. They face new-to-the-world kinds of problems that require a great deal of preparation and development. Here a highly trained, advanced-degree individual will stretch the bounds of a high-end commercial code or require specialized codes from national laboratories to solve the problem.

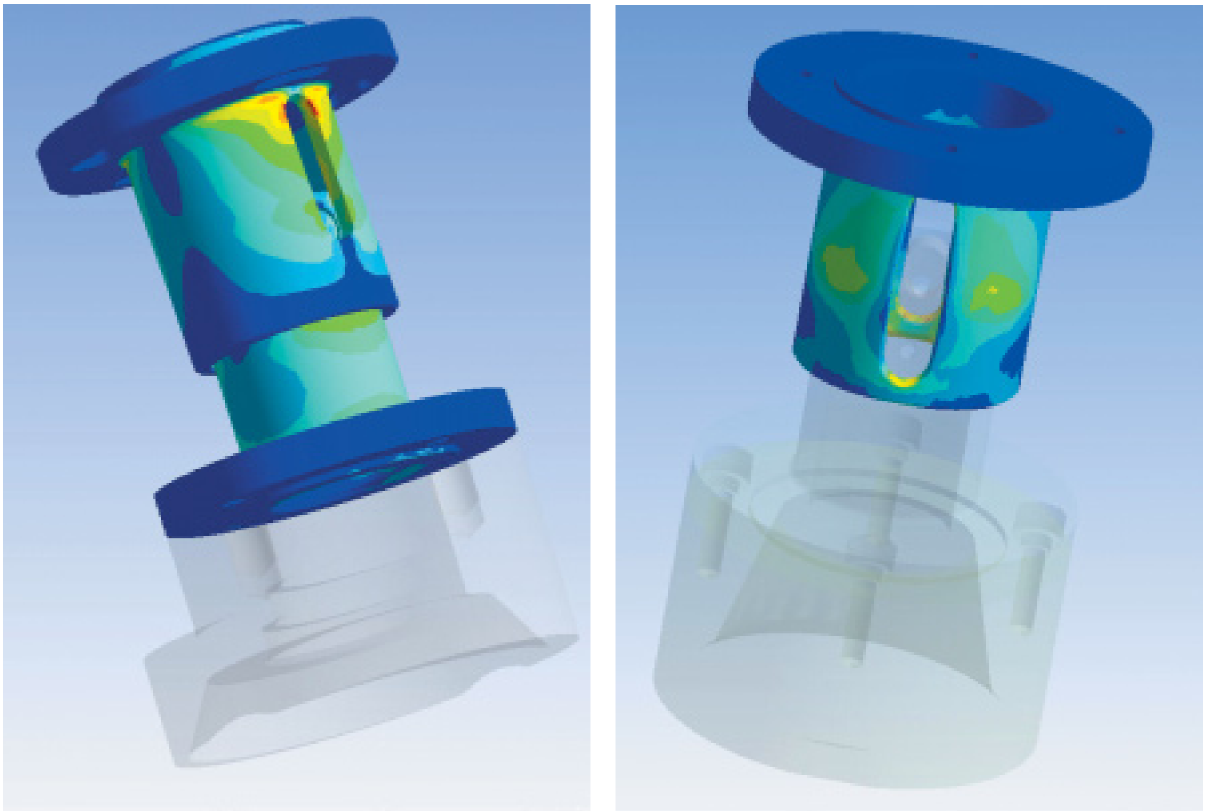
The second-tier analysts use very high-end analysis tools, but the problems are such that the tools can be automated to some extent. A common example at P&G is the analysis of bottles. P&G sells billions of packages each year. Design optimization is critical to maintaining competitiveness and profitability. The sheer numbers of projects annually require a different level of expertise to achieve effective results. P&G has chosen to automate a number of these analyses in a product called the Virtual Packaging System (VPS). VPS is a collection of common analysis tasks that have been developed over the years and automated to a large degree by internally written code. This allows a journeyman analyst to feed various geometries to the system and view the results in short order. The time to complete an analysis is reduced substantially. “This system frees analysts to focus on the physical parameters of the design problem rather than on setting up analytical solutions,” said David Henning, manager of packaging analysis at P&G. Typically, there are three times as many analysts in this category as there are experts.

A third tier is that of the rank-and-file engineer engaged in project work. Occasionally, this individual is faced with the need for an analysis to determine the suitability of a structure for a particular load or other such question. In the past, a call would go to the expert practitioner who may (or may not) have the time or resources to assist. For these types of analyses, P&G uses the ANSYS DesignSpace product as the software of choice. ANSYS DesignSpace was selected after a careful investigation of solutions available in the marketplace.

In making the decision on which software to use, the comparison requirements were ease of use, accuracy, full associativity with a number of 3-D computer-aided design (CAD) systems, and widespread training and support. In the end, ANSYS DesignSpace software was selected. The solution allows for escalation of the problem to ANSYS Mechanical or ANSYS Multiphysics software if a particular analysis requires nonlinear materials, large deformation or advanced contact. The ANSYS Workbench platform also contains tools for convergence studies that serve to ensure an accurate solution.

The ANSYS Workbench environment is available to thousands of engineers and scientists within the P&G organization. Training is available to those who wish to utilize the tools. P&G also finds that more and more new hires are already trained in CAE tools. These software products allow the engineer to rapidly screen numerous designs before having to commit to a physical prototype. The overarching goal is to make sure the first physical prototype has the best chance for success that the engineer can provide.

This translates to fewer, more meaningful tests, decreased innovation cycle times, and, most important, reduced time to market. This is where analysis makes money: in improving the decisions that are made every day and getting a better product to the market faster. ■



Proctor & Gamble rank-and-file engineers routinely use ANSYS DesignSpace software in product development projects. Sample plots here show loads on the slotted concentric shafts of a converting machine assembly, enabling engineers to quickly evaluate the design early in development.

Rotordynamic Capabilities in ANSYS Mechanical

Useful features are available to study vibration behavior in rotating shafts, bearings, seals, out of balance systems, instability and condition monitoring.

By Achuth Rao, ANSYS, Inc.

Rotordynamics is a collective term for the study of vibration of rotating parts found in a wide range of equipment including turbines, power stations, machine tools, automobiles, home appliances, aircraft, marine propulsion systems, medical equipment and more. In these applications, resonant vibration — in which mechanical systems can oscillate excessively when excited by harmonic loads at their natural frequencies — is of particular concern. These large-amplitude vibrations can bend and twist rotating shafts, leading to premature fatigue failure in these components as well as bearings and support structures. Also, deformation of shafts and other components can cause rotating systems to impact adjacent parts in which clearances are tight, causing potentially catastrophic damage in high-speed equipment.

Analysis of rotating systems typically involves the study of many different variables related to vibration including the critical rotational speeds that set up natural-frequency resonances, the response of the entire system to unbalanced loads and instabilities, deflection of the shaft during vibration, torsional vibration in which shafts also twist around their axes, and flow-induced oscillations produced by fluids moving through the system. Calculation of these and other vibration-related variables can be performed in ANSYS Mechanical software using some of the most advanced rotordynamics simulation capabilities available in commercial finite element analysis (FEA) codes.

Rotordynamics usually is best studied in the rotating frame of reference, in which Coriolis terms are used in the equations of motion to describe rotational velocities and accelerations. Introducing these Coriolis terms for static, modal, harmonic and transient analysis provides a modified equation of motion:

$$[M]\{\ddot{u}\}+([G]+[C])\{\dot{u}\}+([K]-[K_c])\{u\}=\{F\}$$

[M], [C] and [K] are the structural mass, damping and stiffness matrices, respectively. $[K_c]$ is the spin softening matrix, and [G] is a "damping" matrix contribution due to the rotation of the structure or the Coriolis term.

This modified equation of motion is at the foundation of performing the most common types of rotordynamics analyses.

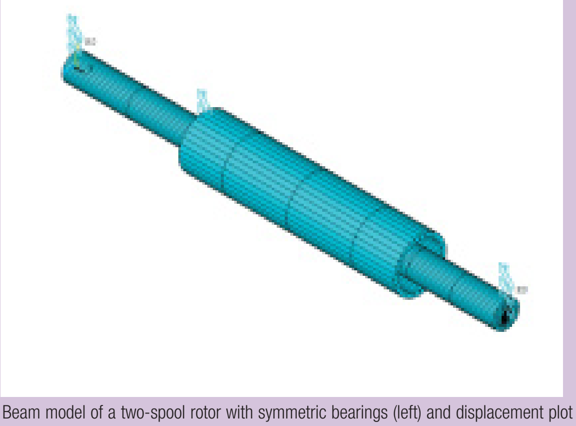
Modal analysis: When components are spinning, the Coriolis term adds nonsymmetric terms that introduce forces to the system, causing natural frequencies to split and shift up and down. These natural frequencies must be determined, therefore, to avoid excitations at the critical speeds. Modal analysis predicts how speed affects frequency by running at speeds from zero rpm up to the maximum rotational velocity of the system.

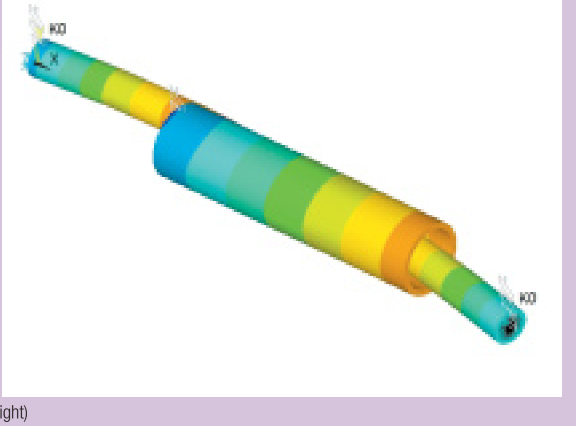
Harmonic analysis: A harmonic analysis sweeps through a range of frequencies to determine how the system responds to various rotating speeds and excitation forces. Again, the Coriolis terms shift the frequencies, and damping plays a greater role. If the excitation is different from the rotating frequency, ANSYS Mechanical technology offers options to scale it up or down.

Static and transient analysis: Static and transient analyses determine loads exerted on structures, joints and bearings of rotating structures. This can be done as a static analysis (by applying initial conditions to specify velocities) or transient dynamic simulation in which the Coriolis effects are included.

Case in Point: Beam Model Analysis of a Multi-Spool Rotor

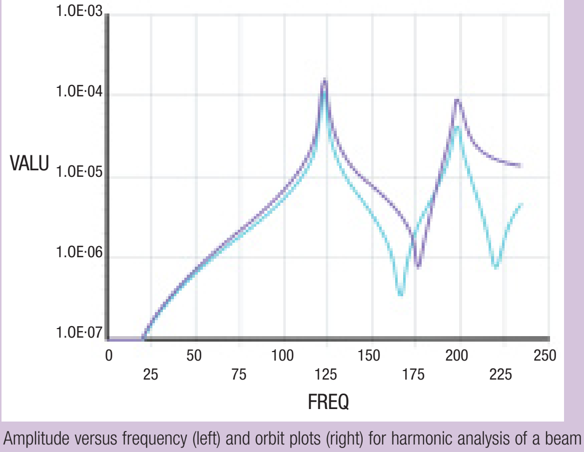
The following is an example of a harmonic analysis of a two-spool rotor on symmetric bearings with unbalance force. An unbalance is located on the second disk of the inner spool, and harmonic response is calculated. The example uses an excitation frequency that is synchronous with the rotational velocity of the structure. ANSYS Mechanical software calculates the rotational velocity Ω of the structure from the excitation frequency and an unbalance excitation force ($F = \Omega^2 * Unbalance$) is applied on the nodes.

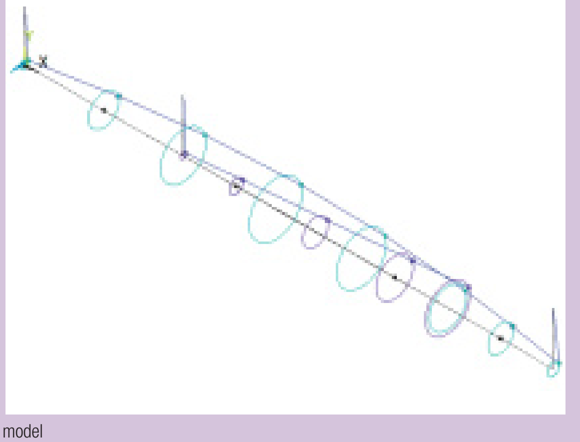




Beam model of a two-spool rotor with symmetric bearings (left) and displacement plot (right)

In a typical rotordynamics harmonic analysis, quantities of interest such as nodal amplitude as a function of frequency, orbit plots at a given frequency and displacements plots at a given frequency are often output as part of the analysis.





Amplitude versus frequency (left) and orbit plots (right) for harmonic analysis of a beam model

Capabilities for Rotordynamics Analysis

ANSYS software offers a complete set of capabilities for studying the dynamics of rotating machinery.

Solids, shell and beam elements: For decades, rotordynamics has been performed with in-house and commercial codes using beams and masses. For most rotor assemblies, this still is the most efficient and the most accurate method. However, sometimes a system does not lend itself to this type of approximation. ANSYS Mechanical software provides a unique solution to address such issues using 2-D and 3-D solid and shell elements to accurately model rotating machinery starting from computer-aided design (CAD) geometry.

Bearings and damping: In real-world rotating systems, bearings are not infinitely stiff, and the friction and lubricant in them introduce damping. Also, springs in these systems often have stiffness that varies with speed and direction. The same goes for damping. ANSYS Mechanical offers spring-damper elements like COMBI14, or the newer COMBI214 for modeling bearings in rotor dynamics, allowing users to specify stiffness and damping ratios for their particular systems.

Stationary and rotating frames: ANSYS Mechanical software provides both rotating and stationary reference frames for rotor-dynamics analysis. The primary application for a stationary frame of reference is a case in which a rotating structure (rotor) is modeled along with a stationary support structure. The primary application for a rotating frame of reference is in the field of flexible body dynamics in which, generally, the structure has no stationary parts and the entire structure is rotating.

Unbalance response: ANSYS Mechanical allows users to specify whether the excitation frequency is synchronous or asynchronous with the rotational velocity of a structure. New capabilities in the software such as the SYNCHRO command update the amplitude of the rotational velocity vector with the frequency of excitation at each frequency step of the harmonic analysis.

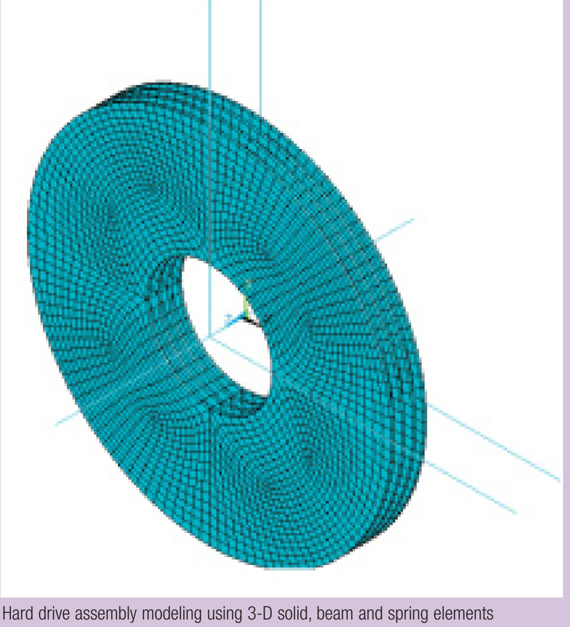
Campbell diagram: The primary post-processing tool for rotordynamics work is the Campbell diagram showing how vibration modes split because of whirling. The Campbell diagram assists users in finding the critical speed for a rotating synchronous or asynchronous force as a function of rotation speed.

Whirl orbit plot: When a structure is rotating about an axis and undergoes vibration motion, the trajectory of a node around the axis generally is an ellipse designated as a whirl orbit. ANSYS Mechanical software provides plotting tools of the whirl for beam/mass and solid rotordynamic models. The orbit (ANHARM macro) can be animated for further examination. ■

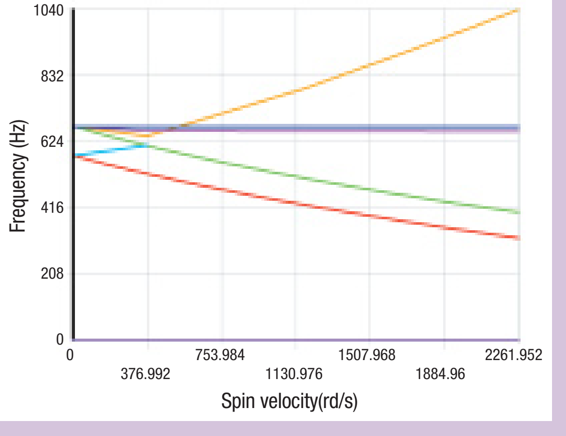
The author would like to thank the development and technical support team at ANSYS, Inc. and Eric Miller from Phoenix Analysis & Design Technologies (PADT) for their efforts and contribution to this article.

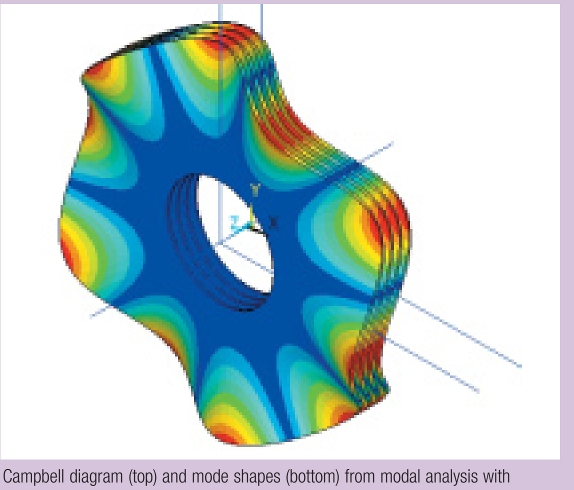
Case in Point: Solid Model Analysis of a Hard Disk Assembly

In a hard disk assembly, modal analysis is run to predict how speed affects frequency by running at zero rpm and then several speeds up to the maximum rotational velocity the system is expected to see. The primary post-processing tool for modal analysis is the Campbell diagram.



Hard drive assembly modeling using 3-D solid, beam and spring elements





Campbell diagram (top) and mode shapes (bottom) from modal analysis with Coriolis effects

Submodeling in ANSYS Workbench

By Dave Looman, ANSYS, Inc.

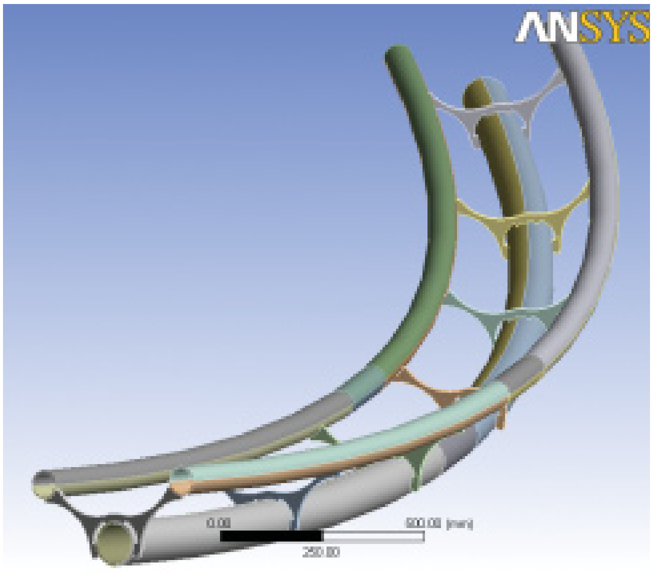


Figure 1. Full CAD model of a curved tubular assembly
Application courtesy Klaus-Dieter Schoenborn, CADFEM.

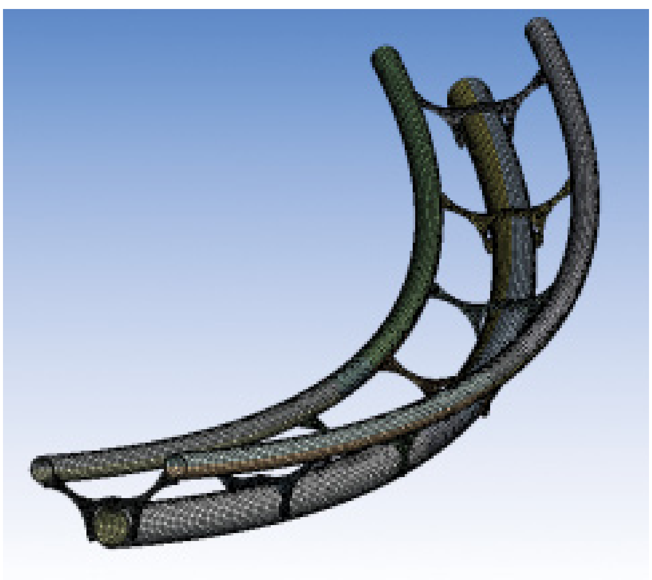


Figure 2. Full global model

Submodeling utilizes two separate models. A full or global model representing the entire structure is used to transform global loads to local deformation. The submodel includes the local geometric details with an appropriate mesh density. The submodeling algorithm then interpolates the deformation from the global model to the submodel "cut boundaries" and solves for the local stress state.

This method typically requires extensive planning and documentation of the workflow, especially if many submodels and numerous load cases are involved. In addition, setup of a submodel may take considerable time. However, the ANSYS Workbench tree and efficient computer-aided design (CAD) interaction make the procedure easier. With small ANSYS Parametric Design Language (APDL) enhancements applied to the model tree, the submodeling technique may be combined with the ANSYS Workbench Geometry handling and process documentation. Thus, a workflow can be presented that covers the whole process from CAD to fatigue analysis in five steps.

Step 1. Build/import the model from CAD. To illustrate this process, a sample analysis is performed to determine stresses on a tubular welded assembly with a regular pattern of joints. It represents a small sample section of a repetitive structure that forms the track of a roller coaster. The model shown in Figure 1 has been created using ANSYS DesignModeler software.

The loading on this structure is caused by a trolley rolling along the two upside tubes. This loading is transferred to the larger tube via the joint elements and passed to the supporting structure. Loading is generated by gravity and centrifugal forces.

To obtain accurate stress in a local region, submodeling separates local analysis from the global model. This allows mesh refinement in a region that might not be possible on the full model without exceeding size limits.

Step 2. Mesh and solve the global model. Each part was meshed independently and connected with surface-to-surface contact. Contact pairs may be part of the interpolated region in the full model, as long as they are completely enclosed by the submodel cut boundary. Figure 2 shows a sample mesh with surface-to-surface contact.

Step 3. Analyze the global model to identify critical spots. The full model gives a general impression of the structural deflection and is used to find the location of critical spots of interest. The structural result is saved and maintained in database and results files in the parent directory. The global solution is needed for interpolation of the submodel boundary conditions.

Step 4. Generate the submodel from CAD data and mesh. Once critical spots are known, submodels such as the one shown in Figure 3 may be created from the original CAD geometry. This is done by cutting the model and suppressing the remaining solids. This process automatically achieves geometrical consistency since the remaining solid does not change its location with respect to the global coordinate system. In order to interpolate displacements from the global model to the submodel, a Named Selection should be created on the cut boundaries, which will be referenced in the macro described next.

Step 5. Interpolate boundary conditions from the global model to the submodel and solve. The interpolation is done by inserting a small macro into the environment branch of

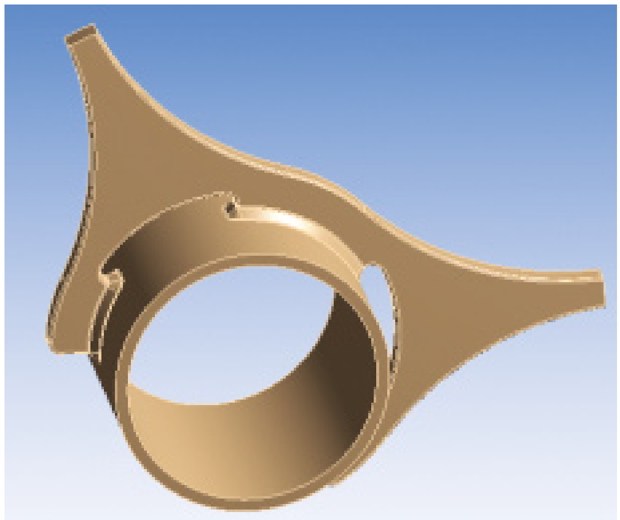


Figure 3. Submodel geometry of a critical structural detail

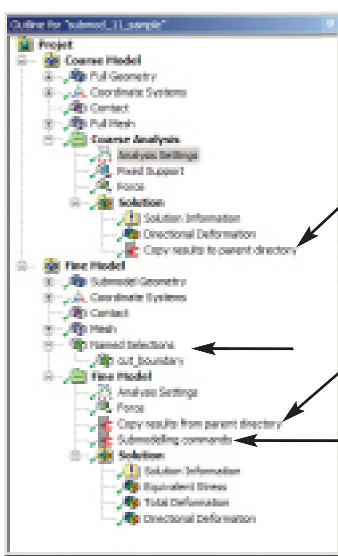


Figure 4. Simulation tree showing the submodeling procedure

```
!!! ANSYS 11.0 Procedure !!!

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Copy results to parent directory !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
/COPY,file,rst,,coarse,rst,..\..\
/COPY,file,db,,coarse,db,..\..\

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Copy results from parent directory !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

/COPY,coarse,rst,..\..\,coarse,rst
/COPY,coarse,db,..\..\,coarse,db
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Submodeling Commands !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

FINI
/PREP7
CMSEL,s,cut_boundary
NWRITE
ALLSEL

SAVE,fine,db
FINI

/POST1
RESUME,coarse,db ! OPEN COARSE MESH
FILE,coarse,rst ! DEFINE RESULTS FILE NAME

SET,1
CBDOF

FINISH
/PREP7

RESUME,fine,db     ! RESUME SUBMODEL FROM FILE
/INPUT,,cbdo

FINI
/SOLU
```

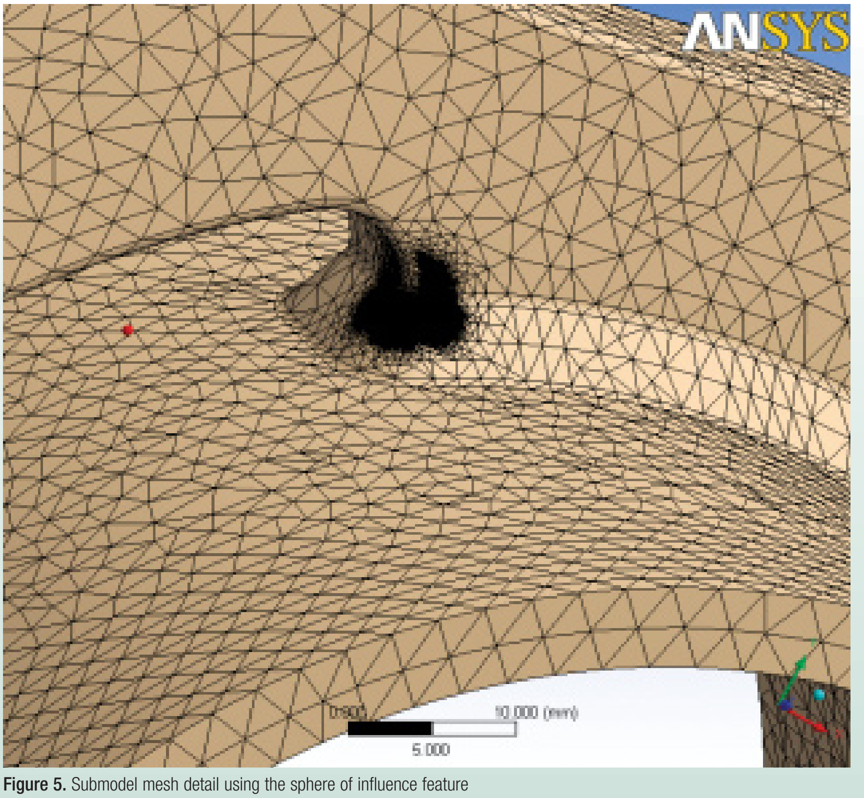


Figure 5. Submodel mesh detail using the sphere of influence feature

the submodel tree as shown in Figure 4. This macro resumes the full model database and results file, and it performs the displacement interpolation (CBDOF command). After that, the submodel is restored, the interpolated boundary conditions are read and the submodel is solved. Note that any external loads present in the submodel (including gravity effects or temperature loading) also should be applied. ANSYS Workbench Simulation solves the model and performs post-processing just as on any regular model. From the interpretation of the resulting local stress state in the submodel, the critical locations now may be reviewed with greater fidelity.

For this example, the initial submodel mesh is found to be still too coarse to accurately predict fatigue life from the resulting stress, so a locally refined mesh is needed. The "sphere of influence" method of the ANSYS Workbench platform is ideally suited to obtain the type of mesh needed. Figure 5 shows the refined mesh on the submodel.

This step is then repeated simply by creating the "sphere of influence" tab on the mesh branch and solving. The interpolation now is performed on the new FE mesh since the macro overwrites any files that were created on a previous run. The interpolation is done from the original results file, which still resides in the parent directory. Equivalent stress in the submodel then can be solved.

In the submodeling process, model consistency is maintained by using ANSYS DesignModeler software to create both full models and submodels. Capturing the process in the tree clearly archives the analysis and allows a user who later opens the database to understand immediately what was done. Note that an arbitrary number of submodels may be created and solved by interpolation from a single run of the global structure. All of these submodels may be included in the model tree, and variants may be studied without having to repeat the whole process. Users should remember to review and compare stresses at the cut boundaries between the global model and submodel to verify that the cut boundary is far enough from the region of interest. Submodels may be altered using the bidirectional CAD interface to ANSYS DesignModeler software or other CAD programs. ■

For more information, refer to chapter 9 on submodeling in the ANSYS Advanced Analysis Techniques Guide.