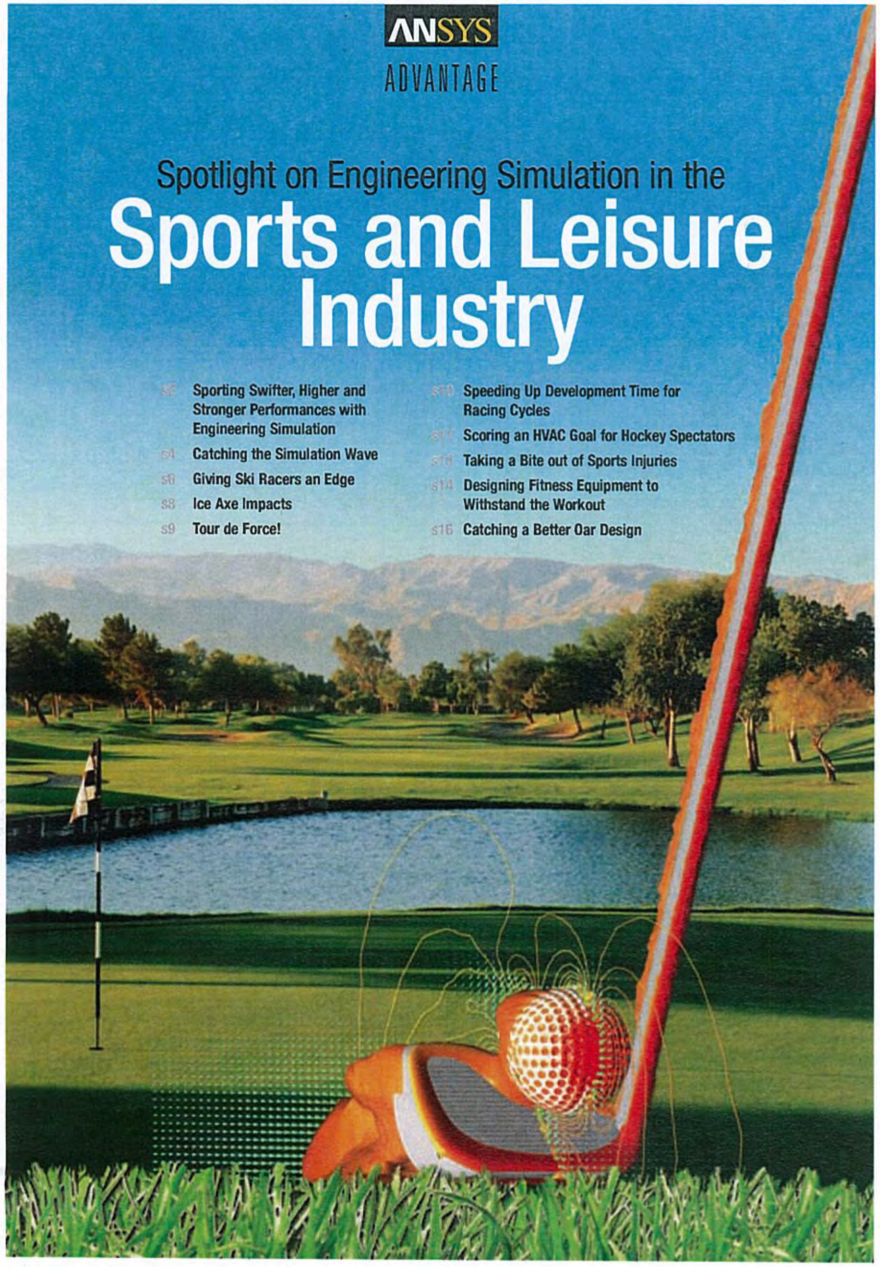


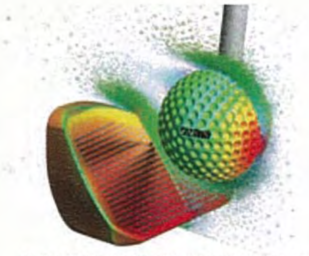
Spotlight on Engineering Simulation in the Sports and Leisure Industry

---

Sporting Swifter, Higher and Stronger Performances with Engineering Simulation

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

By Keith Hanna, ANSYS, Inc.

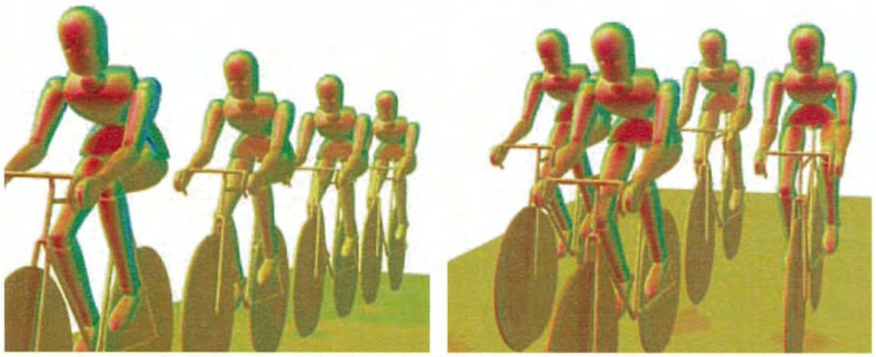

Close up of airflow vectors and surface pressure predictions on a golf ball and #3 iron just after being struck, computed using ANSYS CFX software


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

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

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

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



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

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

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

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

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

Suggested Reading

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

Catching the **Simulation Wave**

Surfers are using engineering simulation to improve their gear.

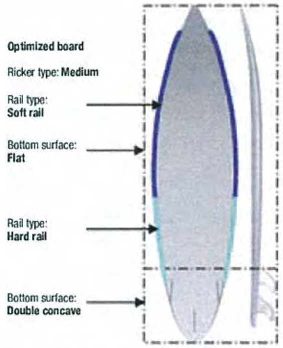
Those passionate about a sport are always trying to enhance the experience for themselves and for others. For years, the search for a perfect board has been a fervent quest among surfing enthusiasts. A surfboard's performance is based on the board shape as well as the shape and size of the fins. Determining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

Designing a **Better Surfboard**

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

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

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



A diagram of a typical surfboard, showing parameters such as the side rails and bottom surface types

The image on the left shows water volume fraction contours on the top of the board surface with the free-surface around the board. The image on the right shows water volume fraction contours on the bottom of the board surface with the free-surface around the board.

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

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

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

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

Developing **More Adaptive** Fins

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

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

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

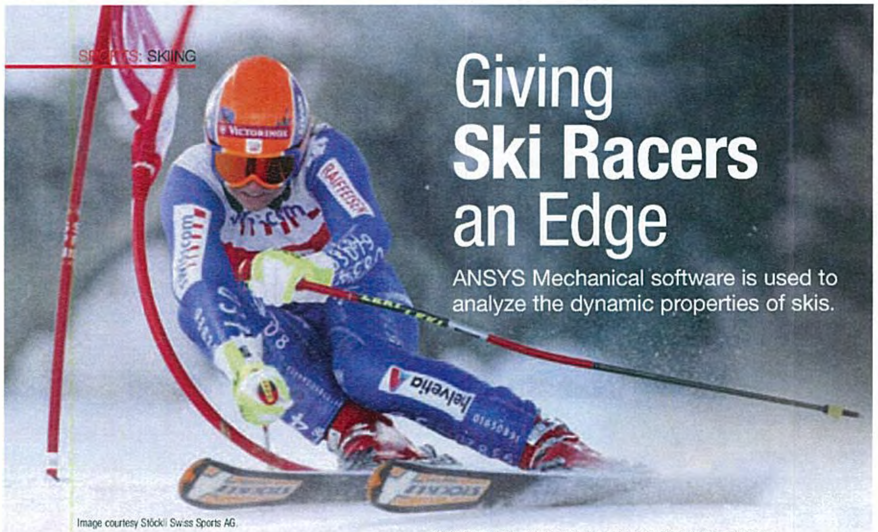
Surfboard fins are complex three-dimensional hydrofoils. Creating solid models of fins, which are essentially scaled down versions of aircraft wings, can take a long time using commercial CAD programs. As a remedy, custom software called Fin Designer™ (www.datdesigner.co.uk/index.htm) has been developed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

An experimental apparatus has been built to measure lift and drag forces on objects in an existing square-section flume, and FLUENT software is successfully predicting these forces at experimentally limited Reynolds number flows, substantially lower than those found in reality. However, once the CFD models have been fully validated, the research team will use FLUENT tools to examine higher flow rates (up to 12 to 15 m/s) that are not achievable in the laboratory using the current equipment.

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

Contours of dynamic pressure on a plane slicing through the fin

Pathlines, colored by velocity magnitude, illustrate the strong recirculation that occurs on the downstream side of the fin.



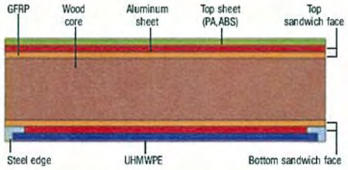
Giving **Ski Racers** an Edge

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

Image courtesy Stöckli Swiss Sports AG.

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

*[1]Laboratoire de Technologie des Composites et Polymères (LTC)
Ecole Polytechnique Fèdèrale de Lausanne (EPFL), Switzerland
[2]WSL, Institute for Snow and Avalanche Research (SLF), Switzerland*



Cross-section schematic of a ski's sandwich structure. Image courtesy Stöckli Swiss Sports AG.

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

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

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

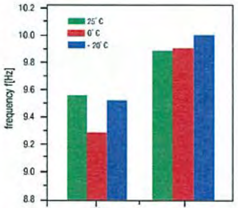
experimental measurements on skis with (referred to as ski+) and without (referred to as ski-) a topsheet.

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

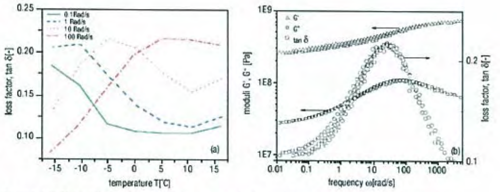
Dynamic mechanical analysis (DMA) was used to measure the damping capacity (loss factor) of the topsheet as a function of temperature at various frequencies. Each damping curve displayed a clear maximum value, or peak, at a temperature that increased as the frequency increased. At a frequency of 10-15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

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

The strong influence of the topsheet on the overall dynamic properties of the ski not only shows that thin layers of viscoelastic materials can have an important influence on the damping behavior of the structure, but also implies that certain specific resonance frequencies can be selectively damped by using polymers with damping peaks in the vicinity of the target frequency. In future work, it will be necessary to introduce more complex models that take into account the viscoelastic nature of the polymeric components. With knowledge of the dynamic response of skis to their constituent materials in hand, designers will have the opportunity to move one step closer to selectively improving ski performance for specific race conditions. In addition to purely mechanical aspects, special attention increasingly is being paid to the athlete–equipment interaction, i.e. the influence of the skis' mechanical properties on the "feel" and the perceived performance of the skier [5]. ■



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



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

References

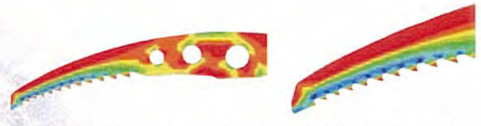
[1] Clerc, C.; Gaertner, R.; Trompette, P.: Computer Aided Design of Skis. Finite Elements in Analysis and Design 5: 1-14, 1989.

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

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

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

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



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

The first principal stress 4.6ms after the initial impact in a transient FEA analysis; the highest tensile stress is apparent at the root of the teeth.

Ice Axe Impacts

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

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

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

There have been a number of recorded failures of ice axes, most of which involve fracture brought on by fatigue [1,2]. Typically, fatigue cracks grow from the serrated edge, particularly from the root of a tooth. An initial finite element study using ANSYS software revealed that a steady-state load applied normal to the end of the pick results in a compressive stress along the serrated edge of the pick with the teeth acting as stress raisers. Fatigue cracks, however, are not normally initiated in areas of compressive stress, but more often are initiated if a tensile stress is present. This can occur if the compressive yield stress for a material is exceeded, resulting in a residual tensile stress. If a residual tensile stress is present at the root of the ice axe teeth, subsequent impact cycles would repetitively expose the tooth area to alternating compressive and residual tensile stresses, ideal conditions for crack initiation and growth. The majority of fatigue literature reports on cracks that are initiated at the site of impact loading, though some also focuses on the phenomena of cracks that are initiated at stress concentrations remote from the impact site.

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

References

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

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

Tour de Force!

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

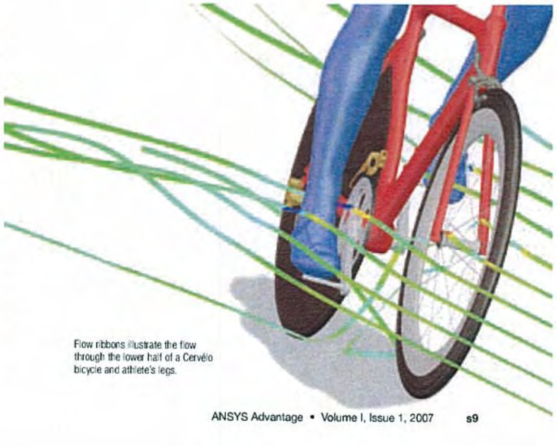
By Keith Hanna, ANSYS, Inc.

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

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

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

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



Flow ribbons illustrate the flow through the lower half of a Cervélo bicycle and athlete's legs.

Speeding Up Development Time for Racing Cycles

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

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

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

Image courtesy Trek Bicycle Corporation

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

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

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

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

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

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

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

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

Scoring an HVAC Goal for **Hockey** Spectators

CFD is used to design ventilation systems for sports arenas.

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

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

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

The Hodynka Arena in Moscow; CFD was used for designing the indoor air conditions.

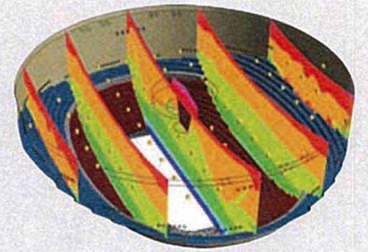
The Tsherepovets Arena, on the other hand, was designed for 6,000 spectators and the indoor air conditions are controlled by a mixing ventilation system. Because both arenas also will be used for other events, such as concerts, CFD was used to better understand the interior conditions and flow behavior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

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

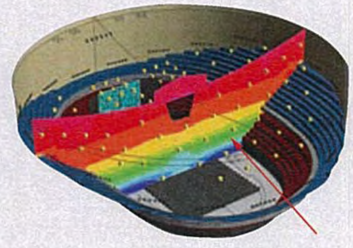
For the ice hockey arena simulations, ANSYS ICEM CFD software was used to build the model, and ANSYS CFX tools were used to simulate and visualize the flow. The first round of simulations that is typically performed for large scale building projects such as these involves individual air supply devices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

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

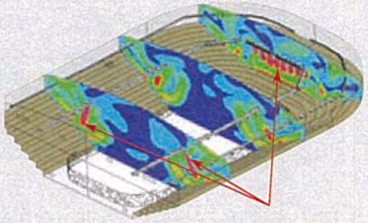
The goals of the arena simulations were to incorporate the parameters that most affect the flow field and ensure that target thermal conditions prevail in zones that are heavily occupied by people at any given time. Given these needs, the challenge was to design the supply air distribution so that fresh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



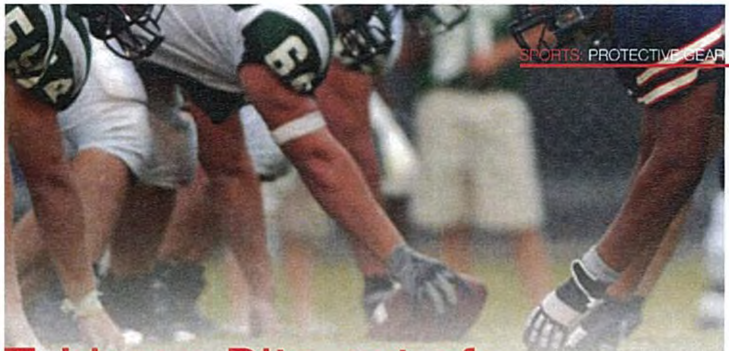
Temperature contours in the Hodynka Arena, showing stratification on several planar slices, during an ice hockey game in summer conditions with displacement ventilation



Temperature stratification during a concert event in summer conditions with displacement ventilation; the arrow points to the area where cool supply air flows to the occupied zone on the field.



Velocity profile in the Tsherepovets Arena during an ice hockey game with mixing air flow from the roof zone; the arrows point to the areas where supply air jets flow to the occupied zones of the stands.



Taking a Bite out of Sports Injuries

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

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

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

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

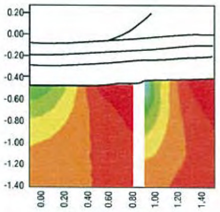
The precise mechanisms by which the device provides such protection are still not well understood, however, and in particular there are no rigorous criteria by which their performance can be assessed or compared. Indeed, the degree to which they protect teeth and surrounding structure has not been thoroughly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

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

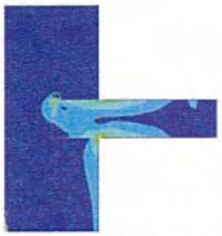
For support, the mouthguard typically is shaped to fit closely around the teeth. This design allows a concentrated load applied to the front surface to be shared by neighboring teeth. The support depends on the rigidity of the guard and its ability to resist local deformation, which is in conflict with the requirements for good cushioning behavior.

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

The cushioning effect of the soft layer of the mouthguard was analyzed with an explicit finite element code to predict impact force given the geometry, material properties and impact velocity.



The supporting effect of the mouthguard and biomaterials was studied with ANSYS Structural. Contact elements represented the low friction contact between the mouthguard and teeth.



Cantilever beam models were used to determine stresses in teeth, socket and gum tissue resulting from impact.

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

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

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

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

Reference

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

Designing **Fitness** to Withstand the

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

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

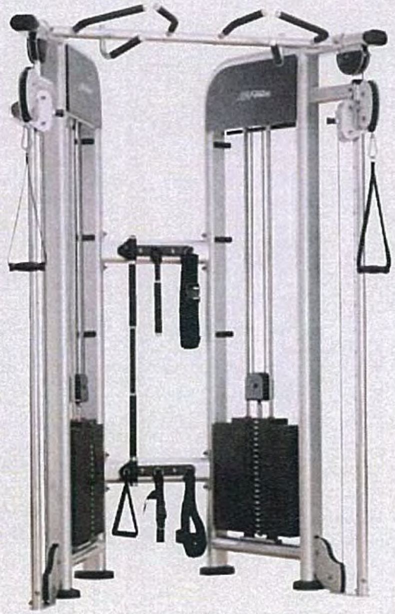
The fitness boom is evident at gyms around the world, as people of all ages take to the mats, weight machines, and cardio stations. Life Fitness is one of the world leaders in developing and manufacturing advanced fitness equipment for the home market, fitness facilities and training centers. Their commercial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

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

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

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

Equipment Workout


Bushings get a real workout in equipment such as this Life Fitness Cable Motion series of strength training machines. Twenty adjustments per column on the equipment's dual adjustable pulley system create a variety of exercises, and the mechanism allows for higher speed movements for sport-specific training.

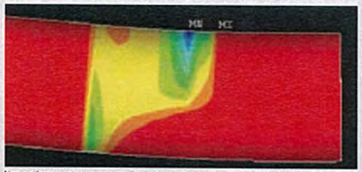
and bushing, however, which develops extremely high non-uniform pressure distributions that are difficult to determine from traditional methods.

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

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

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

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

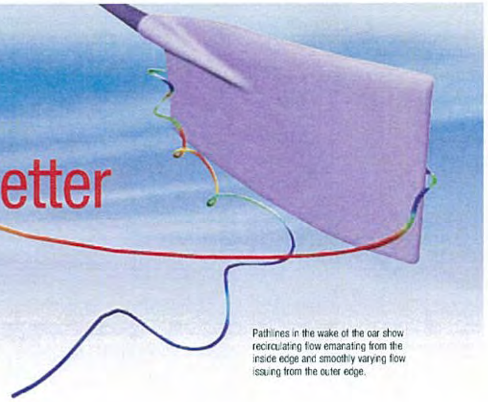


Non-uniform contact pressure distribution between the shaft and bushing enables engineers to accurately calculate bushing wear rates and thus guide decisions on the selection of component materials and surface finishes to maintain overall reliability of the fitness equipment.

Catching a Better Oar Design

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

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


Pathlines in the wake of the oar show recirculating flow emanating from the inside edge and smoothly varying flow issuing from the outer edge.

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

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

The design effort used a mixture of spreadsheet modeling and CFD analysis. Initially a spreadsheet model of the performance of a typical racing eight was developed. This tool was calibrated with race data and was used to link the ultimate performance of the eight to the characteristics of the oar. CFD analysis was used to examine the performance of the oar in detail. The approach used ANSYS ICEM CFD software to develop the computational model and ANSYS CFX tools to obtain the flow solution. Using the k-ε turbulence model, the oar blade was assumed to be fully submersed in water, and its motion was simulated at three locations: the catch, when the oar first enters the water; the middle of the drive; and finally, the finish, just before the oar is removed from the water. The open-water boundaries were placed far from the blade to avoid edge effects. Using a rotating frame of reference, the pivot point was adjusted for each of the three positions to most accurately simulate the compound lever behavior of an oar as it propels a boat through the water.


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

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


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

About the Supplement

Cover image: The flow field in the vicinity of a golf ball immediately after being struck by a club

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**.