ANSYS

ADVANTAGE™

Excellence in Engineering Simulation

VOLUME VI | ISSUE 2 | 2012

SIMULATION FOR SPORTS

10 Dressed for Success

15 Floating on Air

23 Good Vibrations

Welcome to *ANSYS Advantage*! We hope you enjoy this issue containing articles by ANSYS customers, staff and partners. Want to be part of a future issue? The editorial team is interested in your ideas for an article. Contact us.

The Editorial Staff, *ANSYS Advantage*
ansys-advantage@ansys.com

Realize Your Product Promise™
ANSYS is dedicated exclusively to developing engineering simulation software that fosters rapid and innovative product design. ANSYS technology enables you to predict with confidence that your product will thrive in the real world. For more than 40 years, customers in the most demanding markets have trusted our solutions to help ensure the integrity of their products and drive business success through innovation.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.

For ANSYS, Inc. sales information, call 1.866.267.9724.
Email the editorial staff at ansys-advantage@ansys.com.
For address changes, contact AdvantageAddressChange@ansys.com.

Simulation in the News

ANSYS TO ACQUIRE ESTEREL TECHNOLOGIES

ANSYS, Inc., www.ansys.com
May 2012

ANSYS has signed a definitive agreement to acquire Esterel Technologies, a leading provider of embedded software simulation solutions for mission-critical applications.

Esterel provides software and systems engineers with a solution to accurately model and simulate the behavior of the embedded software code to gain insight earlier in the design process and to trace that process to its requirements.

Embedded software is the control code built into the electronics in aircraft, rail transportation, automobiles, energy systems, medical devices and other industry products that have central processing units. For example, today's complex systems, like aircraft and automobiles, can have tens of millions of lines of embedded software code — for cockpit displays, engine controls and driver assistance systems. This spans both graphical user interface code and controls code.

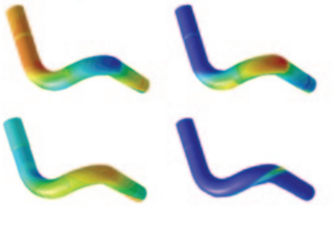
Esterel complements the ANSYS vision by extending capabilities to encompass both hardware and software systems. The combined solutions will enable customers to gain greater insight into the behavior of the embedded software as it interacts with the hardware — or physical plant — including electrical, mechanical and fluidic subsystems. This combination will accelerate development and delivery of innovative products to the marketplace, while lowering design and engineering costs and improving product safety.

The transaction, anticipated to close in the third calendar quarter of 2012, is subject to customary closing conditions and regulatory approvals.

ENGINEERING SEAFLOOR OIL AND GAS PROCESSING BENEFITS FROM SIMULATION

Offshore, www.offshore-mag.com
Vol. 72, Issue 5

Engineering simulation technology can now simulate many subsystems to examine novel concepts and configurations, as well as physical conditions that equipment faces in the real world. One application is sand management, which targets the abrasive impact on downhole tools, processing and transport equipment, and pipelines. ANSYS technology is used to analyze a range of parameters, including fluid proprieties, particle size and distribution, wall materials, geometry, particle loading, and local flow and particle velocity impact.



A STEP AHEAD: JOHN DEERE

:K Magazin, www.k-magazin.de
March 2012

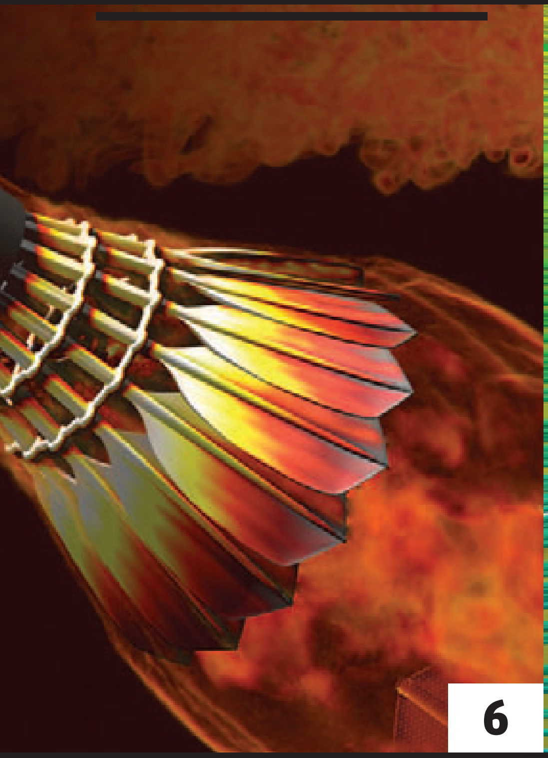
High reliability is a key development goal of the John Deere works in Germany. "With the help of ANSYS simulation software, we can provide our customers with sophisticated products," said Michael Gölzer, in charge of simulation, instrumentation and test facilities. "We can easily compare different design options with respect to functionality and performance in real time." The Mannheim facility uses ANSYS tools for a wide variety of calculations, ranging from individual components and linear–elastic investigations to complex nonlinear analysis. "We now have our second generation of cabs designed completely virtually — there's just a final check with a real prototype," explained Mario Patino, head of the structural mechanics group.

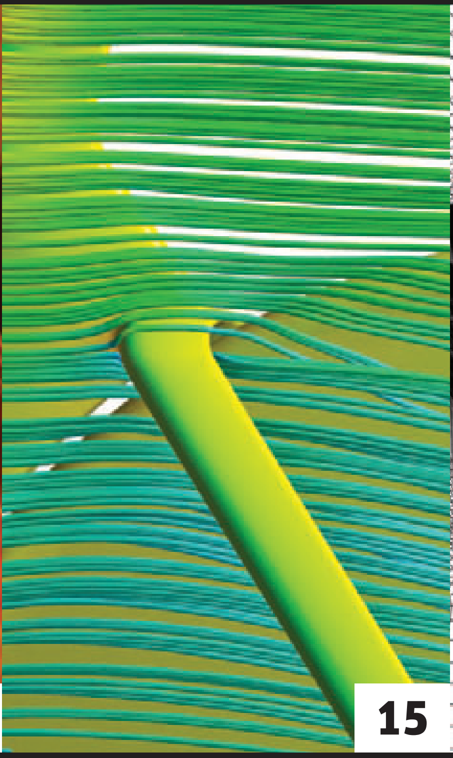
MECHANICAL MEETS ELECTRICAL

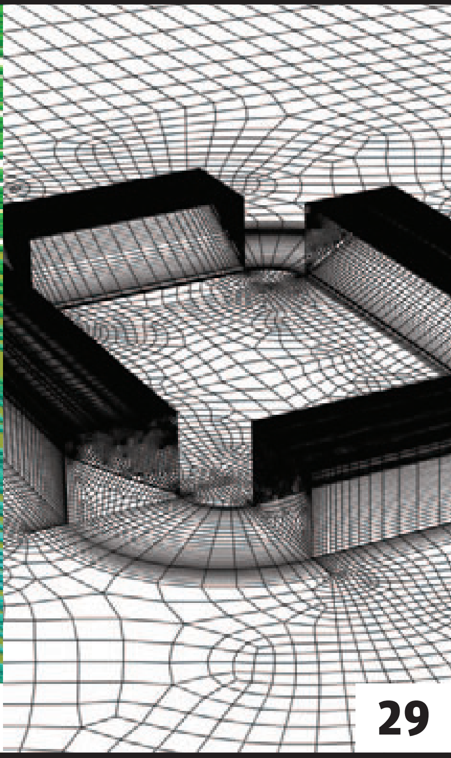
Chip Design, chipdesignmag.com
Spring 2012

Two engineering worlds are intersecting, driven by power, heat and signal integrity. Physical effects can affect many levels of a much larger system. "Inside of an engineering organization, someone near the top has to worry about the entire system," said Larry Williams, director of product management for electronics at ANSYS. "They have to think about boundaries between systems and subsystems, or between mechanical engineering and electrical engineering, because many firms are organized that way. When building a system, the optimum design can be found by considering the system as a whole, and additional margin is often found at those boundaries." As the silos break down, the possibilities are mind-boggling.

Inside of an engineering organization, someone near the top has to worry about the entire system.
– Larry Williams
ANSYS







FEATURES

6
BEST PRACTICES
Become a Champion
Athletic innovations like Olympic swimsuits and racing yachts might seem worlds away from everyday business challenges. But there is much to learn from sports leaders.

9
AUTO RACING
Driving Victory
HPC solutions enable high-fidelity insight into car performance in shorter turnaround times for Red Bull Racing.

10
SWIMMING
Dressed for Success
Engineers at Speedo utilize ANSYS to develop the Speedo Fastskin³ Racing System.

15
CYCLING
Floating on Air
A new racing bike designed with ANSYS CFD won Commonwealth Games Gold and Ironman New Zealand.

20
SAILING
Charting a Confident Course
Engineers at Emirates Team New Zealand rely on CFD from ANSYS to optimize sail performance for the seven-month Volvo Ocean Race.

23
TENNIS
Good Vibrations
HEAD Sport delivers world-leading tennis racket performance with simulation.

26
GOLF
On the Ball
Finite element analysis helps to improve golf ball rebound velocity by 5 percent, resulting in longer drives in a game measured by inches.

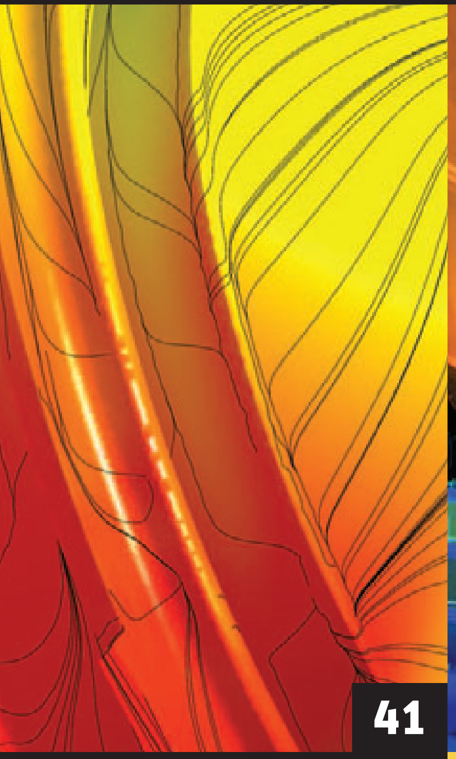
29
CONSTRUCTION
Dry Run
Simulating wind and rain around a stadium determines the best design for keeping spectators dry.

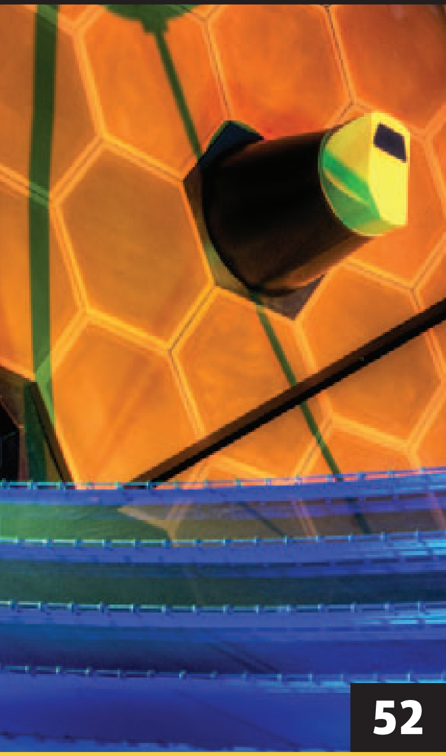
33
ROWING
Germany Rocks on Water
Advances in boat equipment design via simulation help to sustain Germany's Olympic excellence.

ABOUT THE COVER

Speedo's Fastskin³ Racing System includes a cap, goggles and swimsuit designed with the help of ANSYS engineering simulation. When the R&D team completed goggle optimization, the new designs showed significant performance gains over the company's previous design. **PAGE 10** IMAGE COURTESY SPEEDO INTERNATIONAL.


36


41


52

SIMULATION@WORK

36
CONSUMER GOODS
Business is Booming
Multiphysics simulation helps to design a thermal diffuser that converts explosion into long-lasting heat.

39
INDUSTRIAL EQUIPMENT
Pumping Up Sustainability
Flow simulation optimizes the energy requirement of pumps.

41
AEROSPACE
Engineering Simulation Takes Flight
Parker Aerospace uses ANSYS technology to reduce time and costs — as well as risk — in the design of aircraft systems.

46
ELECTRONICS
Wearing a Wire
Simulation helps to optimize body-worn wireless devices for an emerging class of applications.

DEPARTMENTS

50
ANALYSIS TOOLS
To Live Forever
Fatigue simulation can increase product life, reduce costs and decrease risk.

52
TECH TIP
Introducing APDL Math Commands
A new set of commands in ANSYS Mechanical APDL provides access to powerful matrix manipulation features.

WEB EXCLUSIVES

These additional articles are available at *www.ansys.com/exclusives/212.*

TENNIS
The Ball's in Your Court
A team of researchers at Sheffield Hallam University have developed models with ANSYS LS-DYNA software that accurately simulate ball-on-tennis-racket impacts. The team is using these models to investigate the individual and combined effects of many different design variables and to develop insights for improving the design of tennis rackets produced by Prince Sports.

THOUGHT LEADER
Profile of a CFD Pioneer: The Man Behind the CFD Methods
Numerical methods developed by Professor George Raithby have made a significant impact in both academia and the world of commercial fluid dynamics simulation. This article discusses some of his many contributions to CFD as a leading researcher, teacher and educator.

BEST PRACTICES
Modeling Threaded Bolted Joints in ANSYS Workbench
Although bolted joints are extremely common, they can be difficult to model accurately without using best practices. An industry specialist provides some valuable suggestions for using engineering simulation to understand bolted joints to reduce failure.

BECOME A CHAMPION

By **Thierry Marchal**, Industry Director, Sports, ANSYS, Inc.

The Centre for Sports Engineering Research at Sheffield Hallam University, working with Badminton England, the sport's national governing body, used ANSYS CFD software to understand the specific flow-field differences between traditional goose feather shuttlecocks and synthetic versions. ANSYS Fluent simulation revealed the influence of individual feathers, a result that would not have been attainable by other means. Image shows the total pressure over the surface of the shuttle with a volume rendering of vorticity magnitude in the shuttle wake.

Athletic innovations like Olympic swimsuits and racing yachts might seem worlds away from everyday business challenges. But there is much to learn from sports leaders.

This issue of *ANSYS Advantage* features a number of fascinating stories about how innovators such as Speedo and HEAD are changing the face of sports today.

Why are these stories so interesting? Perhaps it's because when we engineers watch a world championship event or a Formula 1 race, we feel that we have some secret knowledge of the technology at the heart of the competition. Or maybe it's because we like impressing our friends at the tennis court or the golf course with some little-known insights about how engineering simulation has shaped the equipment we're using. Whatever the reason, articles about engineering racing yachts and swim goggles never fail to capture our collective interest.

At the same time, these stories might seem worlds removed from the engineering challenges we face every day. How many of us can expect to see a world record broken or a championship podium mounted, based on our own product development efforts?

However, there is much to learn from engineering for sports, which you can apply in your own day-to-day engineering simulations. Design teams at sporting goods companies face the same challenges you do: shorter product lifecycles, growing competition, pressures to drive time and costs out of product development, and ever-increasing regulation.

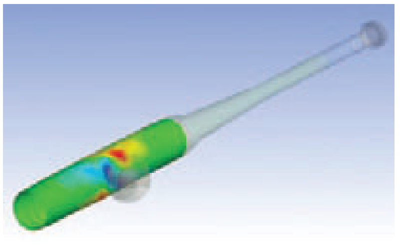
For most athletic equipment designers, the ultimate rewards include not only competitive victories but a larger share of consumer markets, higher long-term profits, and the lower warranty costs that go along with product integrity. These innovation leaders are not very different from your own company. And today, all

Design teams at sporting goods companies face the same challenges you do: shorter product lifecycles, growing competition, pressures to drive time and costs out of product development, and ever-increasing regulation.

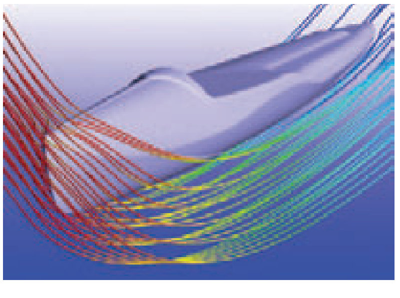
organizations are dealing with the same set of engineering imperatives.

ROBUST DESIGN: A PRIORITY FOR EVERY TEAM

One company that can yield many lessons is Red Bull Racing, a longtime ANSYS customer and vocal proponent of engineering simulation. Only five years after introducing its first Formula 1 car, Red Bull Racing won back-to-back championships in 2010 and 2011. In a sport in which milliseconds matter, the aerodynamics of a million-dollar race car are just as important as driver skill — and Red Bull has acknowledged this fact by using engineering simulation to drive extremely robust vehicle design.



The same composites and explicit dynamics technology used to simulate a baseball bat can be used to design other lightweight, high-impact products.



Using computer simulation, Intelligent Fluid Solutions designed a more stable and better-draining surfski for Red7 without compromising speed. Top racers confirmed improved times, and sales of the product increased.

At Red Bull, much of the vehicle — from airflow and stress points to tires and suspensions — is designed and analyzed virtually before prototypes are ever built. This saves significant costs and time over traditional wind-tunnel tests, and it also provides an incredibly clear picture of how the finished vehicle will behave. Potential performance shortfalls are identified far earlier in the process, which allows for the kind of game-changing, rapid innovation that drives Red Bull into the victory lane, week after week.

Another world-class innovator, Emirates Team New Zealand, focuses on robust design in creating yachts for grueling events like the well-known America's Cup and globe-spanning Volvo Ocean Race.

By relying on computational fluid dynamics (CFD) simulations and parametric analysis, the team is able not only to optimize the performance of its all-important sail inventory but also to ensure that the boat's hull and all appendages are built to maximize the sails' contribution. As yachts embark on races that can span nearly 40,000 miles of uncertain wind and weather conditions, it's obvious why Emirates Team New Zealand places a premium on robust design and product integrity.

AMPLIFYING RESOURCES: FAST TRACK TO RESULTS

Emirates Team New Zealand espouses another universal engineering priority: making the most of product development resources. With only months to design an entirely new yacht for a specific race, the team depends on the power of engineering simulation to analyze thousands of sail configurations, rapidly and reliably. By automating CFD runs and relying on the country's largest high-performance computing (HPC) cluster, Emirates Team New Zealand amplifies its internal resources to meet pressing design schedules and support fast, ongoing innovation.

Large or small, specialized design teams face incredible competitive pressures. Many other athletic equipment and apparel manufacturers have taken this lesson to heart. For those companies focused on winning, engineering simulation is a critical tool for maximizing throughput and launching innovations before their competitors do.

HEAD Sport, a leading maker of tennis rackets, leverages simulation to energize and accelerate its product development efforts, maximizing investments. While HEAD once was able to evaluate only one design prototype each week, today this innovative company uses ANSYS software and a parametric design approach to evaluate a million racket iterations in the same time period. HEAD's efforts to amplify its resources have paid off. Last year, tennis star Novak Djokovic won three Grand Slam titles using a lightweight, high-strength HEAD racket that was designed via engineering simulation.

For companies focused on winning, engineering simulation is a critical tool for maximizing throughputs and launching innovations before their competitors do.

SYSTEMS-LEVEL DESIGN: A WINNING STRATEGY

Today, pioneers in engineering simulation are applying a systems-level, multiphysics approach to not only optimize each individual product component, but to ensure that all components are working together as a high-performing system. This holistic design approach is taking the world of sports by storm.

Speedo is known for its innovations in swimwear, particularly its application of CFD science to maximize fluid flows and minimize drag around competitive swimmers. As you'll see in these pages, Speedo has recently developed an entire "racing system" that incorporates goggles and swim caps into its winning formula. For the first time, Speedo is combining structural analysis with its CFD studies to design revolutionary new swimming equipment that works as a complete system.

Similarly, Avanti Bikes has developed design-award– and championship-winning bicycles by simulating components individually and then as a system, optimizing the machines for the best performance. In a sport in which every second counts, systems-level simulation could help make the difference between winning and placing.

Bicycle designers are using engineering simulation to transform traditional bicycles into advanced, fine-tuned and customized equipment. Today three European universities — Katholieke Universiteit Leuven, Eidgenössische Technische Hochschule Zürich and the University of Eindhoven — are taking this one step further, assuming a systems-level view of competitive bicycling teams.

These academic researchers are focusing on optimizing the position of cyclists on their bikes to reduce drag. Recently, they scanned every member of the Belgian national bicycling team so that each individual's physical profile — and ultimate race performance — could be improved via engineering simulation.

Coupled with ongoing improvements in bicycles, helmets and other equipment, this broader perspective has the potential to revolutionize the sport of bicycle racing. We will continue to follow these developments with great interest — and monitor other ways in which systems-level simulation is impacting the world of athletics.

JUMP-START YOUR OWN INNOVATIONS

Perhaps the stories in this issue of *ANSYS Advantage* will indeed provide fascinating insights that you can share around the swimming pool or tennis court.

But most of all, I hope you find these real-world examples inspiring as you undertake your own Simulation-Driven Product Development efforts.

There might not be a Grand Slam title or a world championship medal in your own future — but, by using engineering simulation to drive innovation in your company, you can emerge as a champion and a hero. Learning how these innovation leaders are focusing on robust design, resource amplification and systems-level simulation can help you and your team to achieve world-class results, whatever your arena.

Using engineering simulation to drive innovation in your own company, you can emerge as a champion and a hero.

A collaborative team of European academic researchers may be the first to apply a systems-level approach to optimize performance of competitive bicycling teams. Drag minimization coupled with ongoing equipment improvements by manufacturers promises to take the sport to an entirely new level.

COURTESY KATHOLIEKE UNIVERSITEIT LEUVEN, TU EINDHOVEN AND ETH ZURICH.

DRIVING VICTORY

By *ANSYS Advantage* Staff

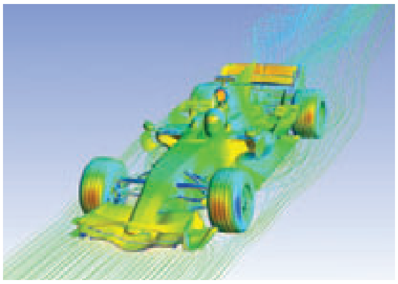
HPC solutions enable high-fidelity insight into car performance in shorter turnaround times for Red Bull Racing.

Red Bull Racing has been widely acknowledged as producing the fastest cars currently in F1. In 2011, fueled by race car design improvements made using simulation software from ANSYS, Red Bull Racing won both the Drivers' and Constructors' titles of the season's Formula One Championship. The team had won the same titles in 2010.

Using ANSYS high-performance computing (HPC) solutions, the Red Bull Racing team continually and quickly optimized its car design in a virtual testing environment on the team's suite of multicore compute clusters. The insights gained gave them a decisive speed advantage on the racetrack.

While fluid dynamics simulation technology is widely used to predict and manage air flows around F1 cars to increase performance, time restrictions placed on all teams require quick, reliable and efficient simulations that also maintain a high degree of accuracy. As a long-term user of ANSYS HPC solutions, Red Bull Racing has benefited from recent technology advances. The team engages high-speed processors (CPUs) and related technologies to solve computationally intensive problems. Because ANSYS delivers significant HPC enhancements with each new release, the Red Bull Racing team is able to solve not only many more smaller models within the same time period, but also to more quickly solve full-car models that contain hundreds of millions of cells.

"ANSYS HPC technology has ensured that we can test and implement changes quickly and competitively," said Nathan Sykes, CFD team leader at Red Bull Racing. "This allows us to turn around simulation results for multiple designs between race qualifications on Fridays and Saturdays, and give our aerodynamics team the important and reliable evidence they need to base vital engineering decisions for the final races on Sunday. To retain freedom to innovate and adapt the car quickly, we rely on a robust modeling process. This puts new designs on the track quickly. To accomplish our goal, we continually need to leverage technologies that help us introduce and evaluate new ideas. With a significant reduction in process times over the last three years, ANSYS HPC solutions have continued to be the tool of choice for us."



ANSYS CFD is used to optimize aerodynamics for Red Bull Racing.

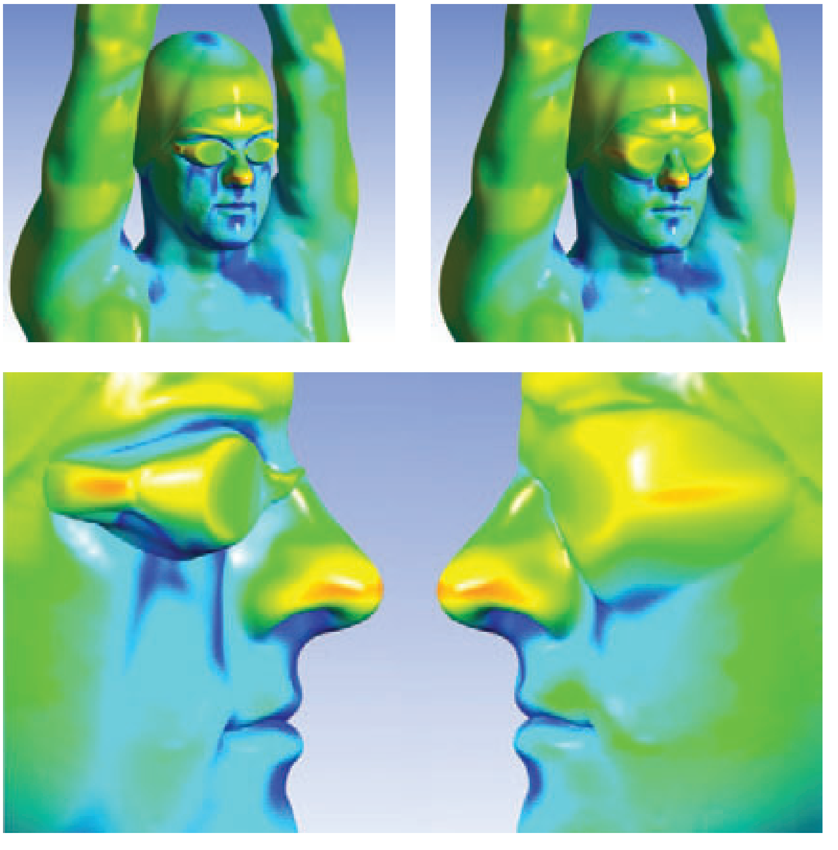
DRESSED FOR SUCCESS

By Stephen Silvester, Technical Services Engineer, ANSYS, Inc., and *ANSYS Advantage* Staff

Engineers at Speedo utilize ANSYS to develop the Speedo Fastskin^3 Racing System.

Speedo's Fastskin^3 Racing System is the world's first swimsuit, cap and goggles designed as a single, cohesive unit.

Engineering simulation has been absolutely critical in launching this world-first concept. *– Dr. Tom Waller*



In analyzing the near-surface flow velocity fields of existing goggle design (left) and a next-generation design concept (right), Speedo engineers were able to see an immediate improvement in hydrodynamic performance.

For the past decade, Speedo has utilized the science of computational fluid dynamics (CFD) to improve the performance of its competitive swimsuits, and the company's R&D efforts have produced remarkable results.

Among the notable developments realized over the course of the partnership between ANSYS and Speedo has been the groundbreaking Speedo LZR RACER® suit. In optimizing this swimwear, Speedo used ANSYS Fluent CFD software to predict fluid flows around a swimmer's body in the outstretched glide position, identifying areas where the slowing effects of drag were likely to occur. During the 2008 Beijing Summer Olympics, 47 gold medals — and 89 percent of all swimming medals — were won by athletes wearing Speedo LZR RACER.

Since 2008, Speedo's research and development efforts have extended far beyond the swimsuit. The company's in-house global research and development facility, Aqualab®, has expanded its use of ANSYS simulation software to include optimizing the caps and goggles worn by swimmers, thus creating a comprehensive suite of products that provide an enhanced competitive edge. The result of these years of research is the Fastskin^3 Racing System®.

A WORLD BREAKTHROUGH

Late in 2011, Speedo debuted its latest swimwear innovation, Fastskin^3 — a cap, goggle and swimsuit engineered to work together as one revolutionary racing system. Research indicates that this innovative system of products, when worn together, reduces full-body passive drag by up to 16.6 percent, improves oxygen economy by up to 11 percent, and reduces active body drag by up to 5.2 percent.

"Engineering simulation has been absolutely critical in launching this world-first concept," said Dr. Tom Waller, head of Speedo's Aqualab. "For the first time, competitive swimmers can utilize a cohesive, hydrodynamic solution that will help them cut through the water with maximum efficiency."

In designing the Fastskin^3 Racing System, Aqualab utilized data from more than 1,200 separate simulations conducted using multiphysics software from ANSYS together with more than 90 human head scans. These simulations revealed the importance of proper fit in optimizing hydrodynamics; therefore, the racing system utilizes a new anatomical marking system called Fit Point Markers. The Aqualab team identified strategic fit points across the cap, goggles and suit that indicate how to align the racing system optimally on the human body. Swimming comfort is maximized, while resistance and drag are minimized.

According to Waller, CFD software from ANSYS played a critical part in developing all three components. "ANSYS has been a technology partner and valuable consultant to Speedo's Aqualab team for years. In developing the Fastskin^3 Racing System, software from ANSYS gave us the confidence that our designs would perform as expected in the real world — and saved us a huge amount of resources that otherwise we would have had to invest in physical testing."

ENVISIONING THE FUTURE

After witnessing the huge impact that hydrodynamic suits made on swimmers' competitive performance in 2008, the

In free-surface simulations that replicated a dive, researchers recognized that its existing goggle design produced significant air entrapment and generated turbulence that would negatively impact a swimmer's downstream performance.

Aqualab team began to explore how the same principles could be applied to other technical equipment, including goggles.

As a first step, Aqualab team members carried out research into the hydrodynamic performance of existing Speedo goggle technologies. A CAD assembly of a current goggle design was imported into ANSYS DesignModeler for cleanup and preparation for surface meshing. Optimal Solutions' Sculptor™ morphing technology was used to fit the goggle surface mesh over the facial contours of a full 3-D scan of an athlete's body. ANSYS Tgrid provided robust prism-layer generation over the complex organic surfaces, coupled to an efficient and high-quality hex-core volume mesh.

Initial simulations performed under steady-glide conditions indicated that localized forces on the goggles contributed to a significant force "spike" that impacted flow-field uniformity downstream. Turbulence generated by the existing goggle assembly produced flow conditions that could negatively impact a swimmer's speed.

DIVING DEEPER INTO PERFORMANCE

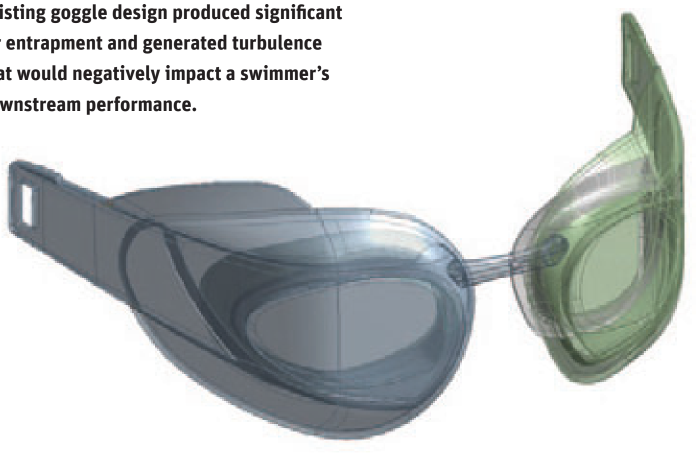
Once Aqualab established the hydrodynamic implications of goggle design, the team initiated a process of simulation-driven development and optimization.

This involved testing a next-generation goggle design under the dynamic conditions of competitive swimming, in which goggles are subjected to different pressures from all directions — including high-impact forces experienced during diving. To obtain a more complete understanding of the real-world performance of the new goggle design, the team used ANSYS Fluent to simulate an athlete piercing the water's surface.

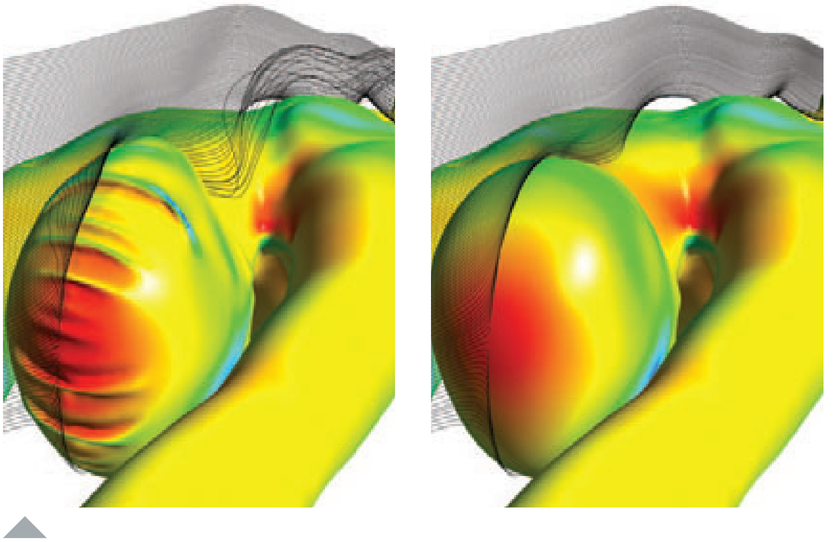
Free-surface simulations demonstrated two key performance characteristics of the goggle design. First, when compared with existing goggle technologies, the new concept registered a far lower impact force. Second, the new goggles produced significantly less air entrainment and, consequently, less turbulence that would negatively impact downstream performance. The Aqualab team believed that this transient free-surface work was vital to understanding the true potential of their new goggle design.

In subsequent parametric simulations, Aqualab team members refined the overall design of their advanced goggles to ensure both regulatory compliance and maximum hydrodynamic efficiency. The team tested the structural response of the goggle design, evaluating alternative polymer materials using ANSYS DesignModeler and ANSYS Mechanical. Researchers incorporated the ANSYS Workbench platform to ensure a robust design.

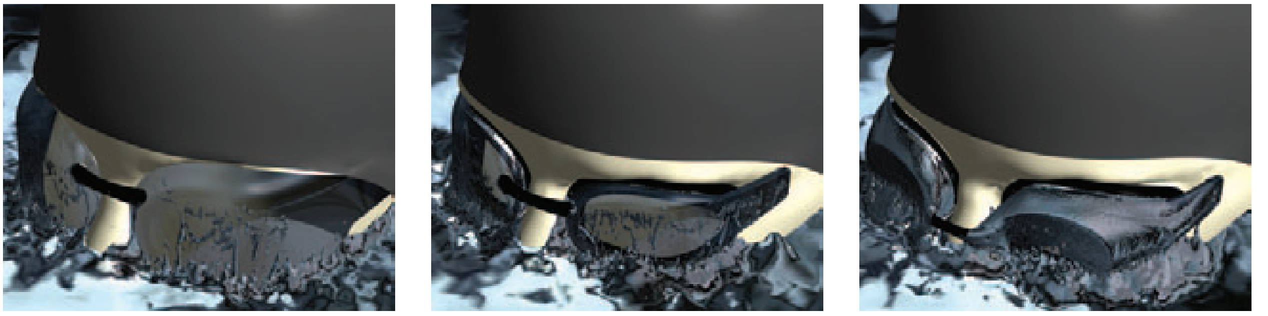
"The parametric and multiphysics analysis capabilities of ANSYS software helped our team rapidly achieve answers and optimize the overall fit of the new goggle design, as well as understand the contribution of each individual product component," said Waller. "This was the



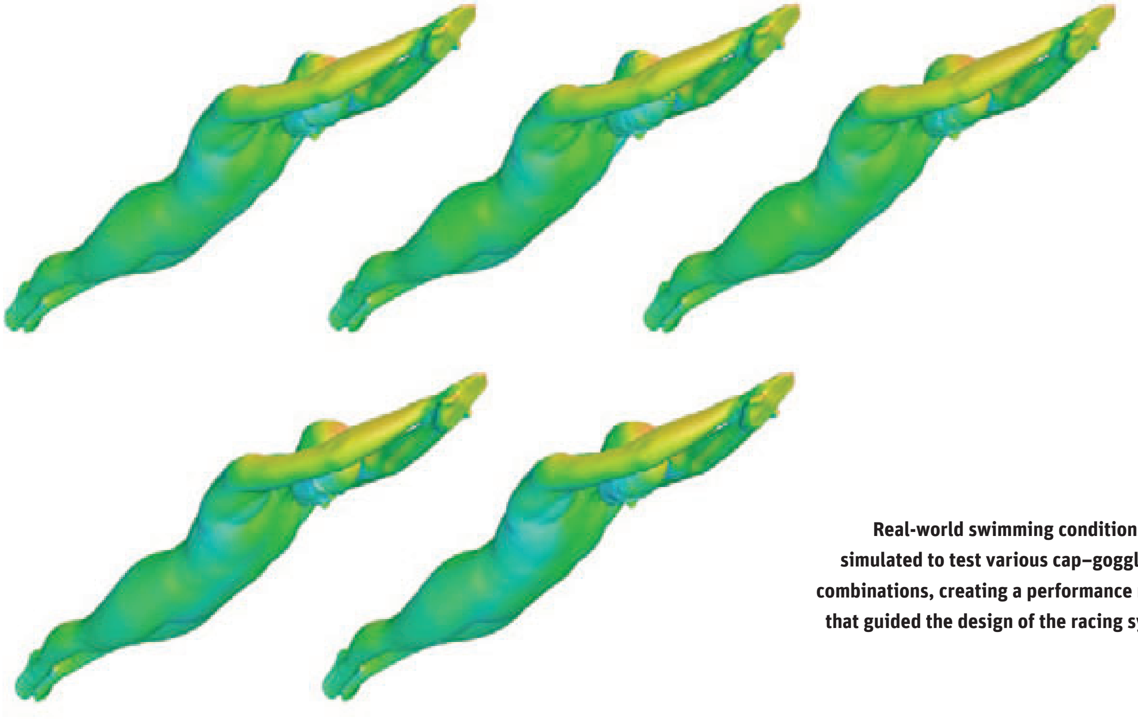
Speedo relied on ANSYS DesignModeler to refine the overall shape of its next-generation goggles, including lens position relative to the eye, bridge design and fit, side arm assemblies, and goggle seals. Shown is a CAD image of the goggles.



A CFD simulation using ANSYS Fluent revealed significant flow-field disruptions — and fluidic drag — caused by an inefficient swim cap shape.



Left to right: Speedo's Super Elite and Elite goggles represent significant performance gains over the company's existing Aquasocket goggle technology.



Real-world swimming conditions were simulated to test various cap–goggle–suit combinations, creating a performance matrix that guided the design of the racing system.

Speedo can rely on ANSYS for the technology that will drive ongoing product innovation. – *Dr. Tom Waller*

first time we integrated structural analysis, which enabled us to explore new materials that are lighter in weight yet maintain the strength needed to withstand the effects of diving and other high-impact forces."

When Aqualab's engineering simulation exercises were completed, the team had reduced impact forces on the goggle assembly by more than 60 percent when compared with Speedo's traditional goggle design.

HEADED FOR SPEED

With goggle optimization efforts under way, designers at Speedo's Aqualab focused on another critical component: the caps worn by competitive swimmers.

An initial study analyzed 90 head scans, through which engineers could study the full variation of actual swim cap fittings. From this data set, the team characterized distinct cap shapes — primarily dictated by how wearers contained their hair. These head shape variations were evaluated for drag by morphing a reference model to replicate the typical cap shapes identified through the scanning study.

These initial simulations indicated that an optimization of hair position within the cap shape could lead to a global gain of 1.7 percent. The results of this study led Speedo designers to develop an entirely new product: a hair management system designed to be worn in conjunction with a swim cap, ensuring that hair is optimally positioned to minimize drag.

Speedo Fastskin^{3} is a total system designed to work together from head to toe. While the goggles, cap and suit are individual items, I feel like the design of this racing system helps make a difference. *– Natalie Coughlin*

A SYSTEMATIC APPROACH

The prototypes for new cap and goggles were designed in conjunction with a new swimsuit, designed in line with guidelines revised in 2010 by the world governing body of swimming, FINA (Fédération Internationale de Natation), calling for swimsuits that cover less of the swimmer's body than the previous generation of whole-body suits.

Both male and female professional athletes were scanned in the outstretched glide position, creating a design matrix of 45 permutations of suit, cap and goggle — including existing, alternate and optimized prototype designs.

These high-resolution scans were quickly prepared for analysis using script-driven ANSYS Tgrid wrapper and volume meshing technology, before being passed on to ANSYS Fluent for analysis and ANSYS CFD-Post for post-processing.

The simulations conducted represented typical real-world swimming conditions. The resulting performance matrix enabled Aqualab designers to identify the most successful suit–cap–goggle combination, as well as to establish individual performance figures for each component.

Continued refinements to the goggles based on these whole-body simulations resulted in the final Fastskin^{3} Super Elite and Elite goggle designs. A significant reduction in the localized forces on the goggles delivered a global 2 percent performance gain; furthermore, it improved comfort during impact with the water surface and contributed to making the goggles fit stably and securely during the high-speed, dynamic maneuvers typical of competitive swimming.

Following the final product development work, Aqualab conducted a number of pool tests on the new suit, cap and goggles using professional athletes, including Michael Phelps (USA), Rebecca Adlington (GBR) and Ryan Lochte (USA).

On testing the suit, Phelps said, "Speedo Fastskin^{3} makes me feel completely at one with the water. I feel confident, I feel comfortable, and I feel like I am wearing the fastest."

CONTINUED IMPROVEMENTS ON DECK

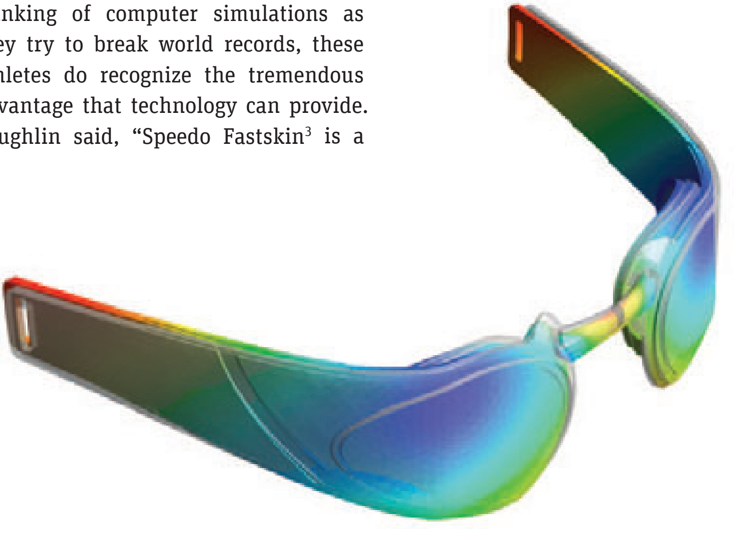
The development of this racing system marks the first time that Aqualab designers have moved beyond CFD analysis, incorporating a multiphysics approach that also considers structural analysis. According to Waller, this expanded approach to simulation matches the broader portfolio of products that Speedo is optimizing today.

"In 2008, Speedo established the dominance of our friction-reducing, hydrodynamic swimwear," noted Waller. "Moving forward, we want to answer the whole-body needs of competitive swimmers with a line of products that maximize overall performance and speed. The Fastskin^{3} Racing System is a revolutionary development, and we will continue to utilize ANSYS solutions to drive even greater product performance gains. Whatever the needs of swimmers, they can rely on Speedo to meet those demands — and Speedo can rely on ANSYS for the technology that will drive ongoing product innovation."

While professional swimmers like Natalie Coughlin (USA) may not be thinking of computer simulations as they try to break world records, these athletes do recognize the tremendous advantage that technology can provide. Coughlin said, "Speedo Fastskin^{3} is a total system designed to work together from head to toe. While the goggles, cap and suit are individual items, I feel like the design of this racing system helps make a difference."

Whatever the future brings, one thing is certain: The ANSYS–Speedo partnership will continue to be a driving force in swimwear innovation.

"The innovations we've debuted since 2004 would not have been possible without the design analysis and verification capabilities of ANSYS software," said Waller. "ANSYS has been a key partner in helping to drive groundbreaking new products such as the new Fastskin^{3} Racing System. I'm confident this collaboration will continue to impact the future of competitive swimming by driving ongoing innovation."



ANSYS Mechanical provided the capability to study flexibility and strength of three different polymer materials that were candidates for the goggles' nose bridge.

FLOATING ON AIR

By David Higgins, Product Design Engineer, Avanti Bikes, and
Larisa Marsh, Director, Dynamic Sports Engineering (DSE)
Auckland, New Zealand

PHOTO: PAUL SLOPER

A new racing bike designed with ANSYS CFD won Commonwealth Games Gold and Ironman New Zealand.

Alison Shanks' Commonwealth Games gold medal–winning ride on the Evo II

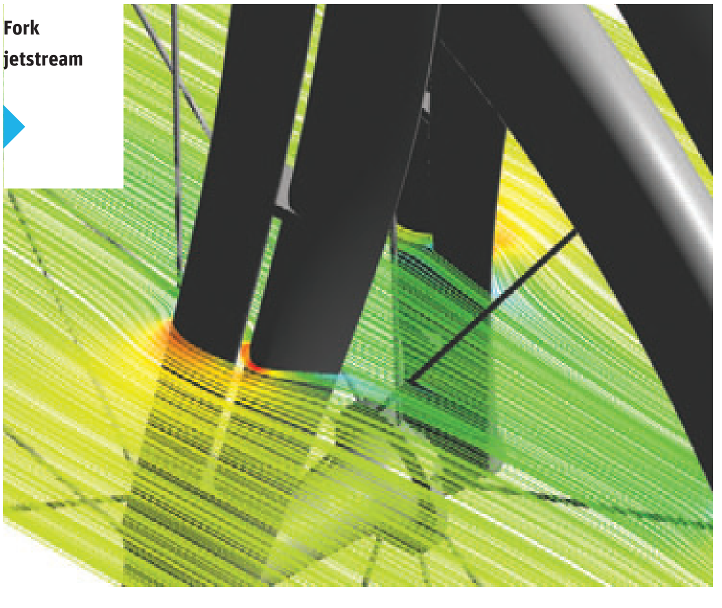
In a sport in which every fraction of a second counts, the Chrono Evo II and Pista Evo II are Avanti Bikes' fastest racing machines ever. At the 2010 Commonwealth Games, Alison Shanks rode her Avanti to capture gold in the women's 3,000 meter individual pursuit; Cameron Brown rode his to victory at the Ironman New Zealand Triathlon in 2011. The project recently received the highest honors at the New Zealand Best Design Awards and a prized 2012 International Red Dot Design Award.

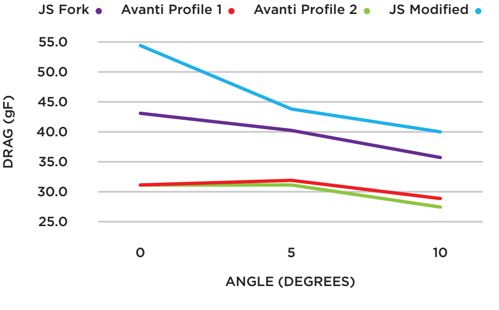
Frames for the time trial/triathlon (Chrono) and track (Pista) racing bikes were developed with fluid dynamics software from ANSYS. To help design the bikes, Dynamics Sports Engineering (DSE) engineers first used ANSYS CFD-Flo computational fluid dynamics (CFD) software to optimize the aerodynamics of individual components, such as the head tube, fork, bottom bracket and seat stay attachment. Next, engineers performed a systems simulation as they combined components into a full structure and tested the aerodynamic performance of the assembly at crosswind angles from −20 degrees to +20 degrees. Engineers manually iterated through a series of designs to an optimized solution that provided about a 20 percent reduction in drag compared to previous models. This drag reduction could decrease a rider's time by approximately one minute over one hour of race time.

The project recently received the highest honors at the New Zealand Best Design Awards and a prized 2012 International Red Dot Design Award.

Avanti's Chrono Evo II

Fork jetstream





Fork drag (isolated): improvement in fork drag at various wind angles

LEADING RACING BIKE MANUFACTURER

Avanti released its first bike in 1985 and now offers more than 110 models, distributed primarily in New Zealand and Australia. Avanti contracted DSE, a small New Zealand engineering consulting firm that comprised two engineers at the time, to help design its new bike from an aerodynamic standpoint. Founding DSE engineer David Higgins has since gone to work for Avanti.

DSE engineers used CFD-Flo, which is specifically geared to product designers and driven by the power of ANSYS CFX. The software provided direct access to the initial design of the bike produced in Pro/ENGINEER®. A wizard guided engineers through the simulation physics setup, including specification of boundary conditions, and verified that all necessary information was entered. CFD-Flo includes leading-edge technology that provides fast run times and quick project turnaround to efficiently integrate CFD into a design process using the ANSYS Workbench environment.

DSE began working on the aerodynamic properties of the components in the approximate order that these bike parts meet the airstream. The Union Cycliste Internationale (UCI), the governing body of bicycle racing, specifies that frame tubes must be no more than three times as long as their width with a maximum depth of 80 mm and a minimum width of 25 mm. The frame design consists of two connected triangles, one toward the front of the bike and the other toward the rear. The key difference between the time trial/triathlon and track versions of the bike is that the former is designed for typical outdoor wind conditions while the latter is designed for the zero-wind conditions experienced indoors.

FORK AND FRONT JUNCTION DESIGN

To create each fork blade, the company first evaluated a proprietary fork design consisting of paired airfoil sections; engineers determined that this was not the optimal solution for the bike. The proprietary design directs the airstream away from the wheel. In reality, track bikes like the Avanti Pista Evo II use disk wheels, for which it is desirable to attach the flow to the back side of the wheel (so the flow envelops the wheel). Engineers decided to use a conventional fork and evaluated different fork depths and widths at a number of different airstream angles, targeting the optimized angle at about +10 degrees. Based on the results, they developed a series of new fork designs. The end result: The company delivered a design that provided an isolated drag force of 27 grams of force (gF), compared to 36 gF for the proprietary fork design (which had, to this point, been considered state of the art).

Engineers then moved on to other frame components, starting with the front junction, where the front tube attaches to the top tube just below the handlebars. In the previous-generation design, the brake was attached to the rear of the fork, the handlebars were above the top tube, and the top tube slanted toward the front of the bike. Based on analysis results, engineers evolved the design so that the handlebars aligned horizontally with the top tube, and the brake was internal to the fork. Housing the brake inside the streamlined fork structure reduced the turbulent effects of the brake. Additionally, both the time trial/triathlon and track bikes are much deeper front to back. The new design improves handling wind coming from the side of the bike, converting it into side force rather than drag.

REAR JUNCTION DESIGN

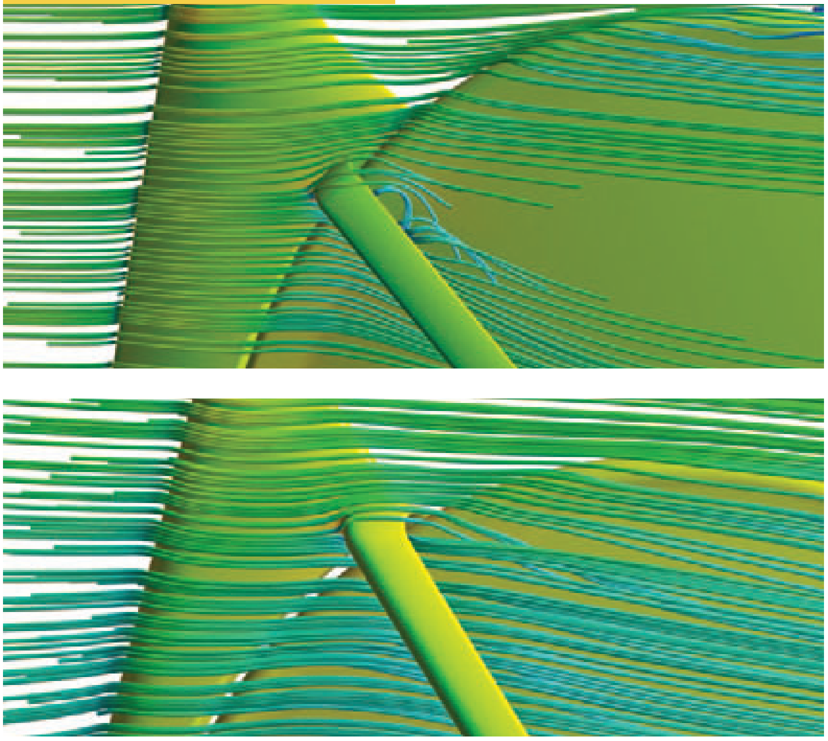
Next, engineers addressed the rear junctions of the bikes, where the seat tube, top tube and seat stays come together. The previous-generation bikes had a relatively wide frame member with the rear brake attached near the top of the seat stays. CFD showed relatively poor aerodynamic performance for this area, so they redesigned it from scratch, reshaping the seat tube to match the rear-wheel profile. The team tried nine different ways to attach the seat stays to the seat tube, ending up with a symmetric National Advisory Committee for Aeronautics (NACA) profile. Engineers also redesigned the bottom bracket area, where the pedals connect to the frame. Housing the rear brake behind the bottom bracket and under the chain stay greatly reduced drag compared to having it in a conventional road bike position at the top of the seat stays.

Engineers also provided complete internal cable routing, making both versions of the Evo II the first bikes to have every cable integrated with the frame. The cables are routed internally from the levers, through to the derailleurs and brakes including through the handlebars and stem. Internal cables offer reduced drag as well as enhanced visual aesthetics.

COMPLETE STRUCTURE DESIGN

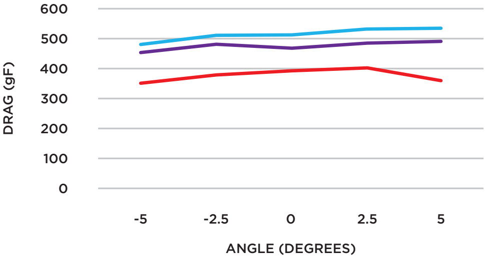
Once the engineers were satisfied with the bikes' individual elements, they combined the components as a system and analyzed the entire structure. It took approximately eight hours to solve each complex full-bicycle model. The initial results were good; still, the team made further improvements with some additional fine-tuning at the systems level. The first prototypes were then built and tested in a wind tunnel at various wind angles. Testing confirmed that the new bicycle designs developed with ANSYS software delivered 20 percent less drag than the previous-generation designs.

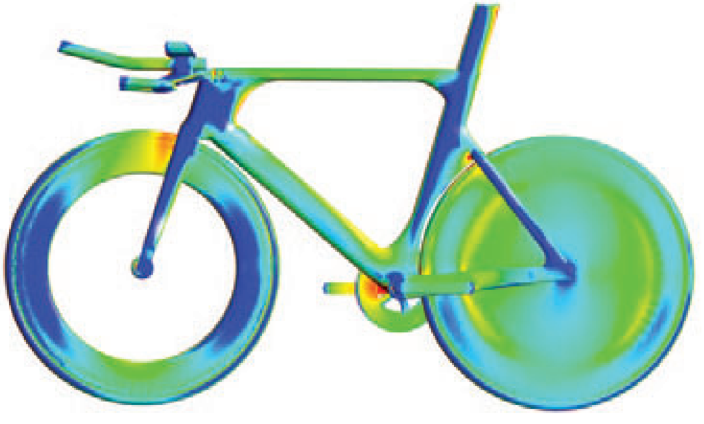
Testing confirmed that the new Avanti Evo II bicycle designs developed with ANSYS software delivered 20 percent less drag than the previous designs. This could reduce a rider's time by one minute for every hour of riding.



(Top) In each version of seat-stay design, flow was not attaching to rear wheel for the Pista Evo II. (Bottom) Simulation of final version of rear seat stay shows improved flow around wheel.

Track Bikes Alone — Normalized to 30 mph (Drag Converted to Bicycle Axes): There was a 20 percent reduction in drag from earlier versions of the bike (light blue) and the later version (dark blue). CFD simulation is shown in red.





The bicycle components were simulated individually and as a complete system so that design tweaks could be made to optimize performance.

RAVE REVIEWS

Avanti's resulting Evo II bikes have received rave reviews from the bicycle press. *Biking Australia* reported, "The result is a fantastic-looking bike that is a far cry from Avanti's previous models. Avanti has taken advantage of the latest technology available to bring to the market a bike wanting for nothing. The carbon fiber frame is the result of an exhaustive design process. It takes the concept from sketch to reality via numerous steps that, no matter how seemingly insignificant, can be crucial in the way the final-functioning bike behaves both in response to riders' commands and to the environmental conditions."

The judges of the product category from the New Zealand Best Design Awards described Avanti's award-winning bike as "a world-class product demonstrating the highest standards of build, technology and New Zealand design ... This amazingly light performance machine demonstrates meticulous attention to detail. The design confidently represents New Zealand in one of the world's most demanding and exacting sport and recreational arenas."

Reference

www.dseng.co.nz

Avanti has taken advantage of the latest technology available to bring to the market a bike wanting for nothing. – *Biking Australia*

The Avanti Evo II bicycles — Chrono (top) and Pista (bottom) — were designed using simulation to deliver the performance required by world-class athletes.

Avanti Riders

Avanti Triathlon and Time Trial Riders

Hamish Carter
Olympic Champion and Commonwealth Medalist

Cameron Brown
10-time Ironman Champion

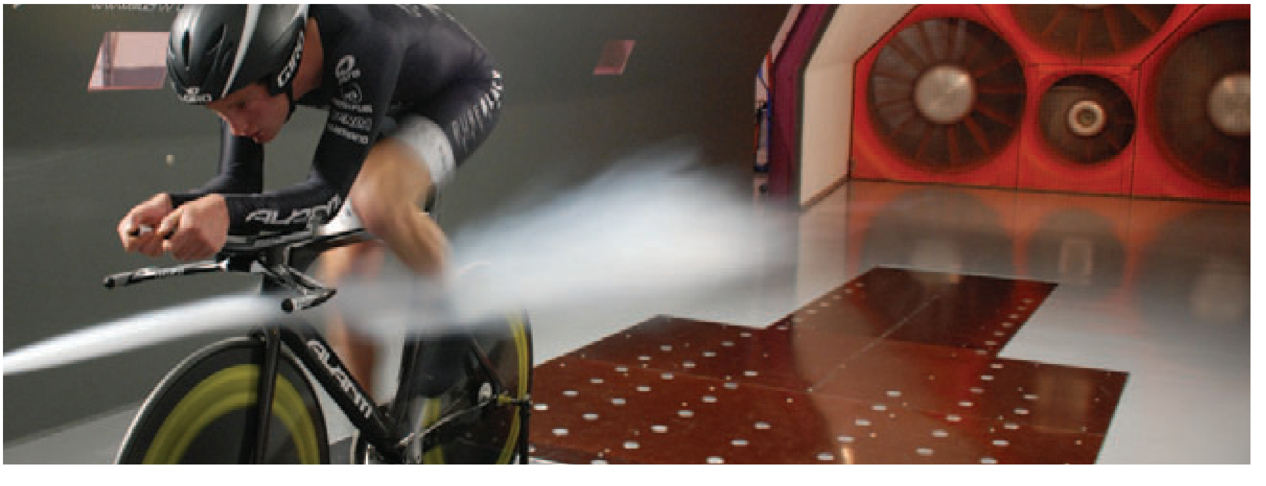
Bevan Docherty
Olympic Medalist

Avanti Track Riders

Alison Shanks
World Champion and Commonwealth Gold Medalist

NZ Women's Pursuit Track Team

Hayden Godfrey
World Champion



Wind-tunnel testing correlated well with simulation predictions.

Charting a Confident Course

By Nick Hutchins, CFD Engineer
Emirates Team New Zealand
Auckland, New Zealand

Engineers at Emirates Team New Zealand rely on CFD from ANSYS to optimize sail performance for the seven-month Volvo Ocean Race.

The Volvo Ocean Race is one of the most grueling sports events in the world. In this event, held every three years, professional teams of 11 sailors guide their yachts over more than 39,000 miles of uncertain ocean conditions — a challenging route that spans the globe and includes some of the world's most treacherous seas, including those around Cape Horn.

From October through July, Volvo Ocean Race crews sail up to 20 days at a time, as they complete nine separate legs of the competition. Crew members face a daunting regimen of freeze-dried meals, waves up to 10 meters, and temperature swings from −5 C to 40 C.

While the race brings daily challenges for athletes, this demanding event also poses special problems for yacht designers. Race rules restrict each yacht to only eight sails for any given leg, compared to the 13 sails an America's Cup Class yacht typically carries for a two-hour race. Crew members manipulate the eight sails in navigating the transocean route, where wind conditions can range from becalmed to force 10 gales, originating from any point on the compass.

This combination of few sails and uncertain conditions means that Volvo Ocean Race competitors are often forced to sail at challenging headings and wind speeds without ideal sails for such conditions. The yacht's hull and appendages must be designed to compensate. Consequently, early in the boat design process, yacht designers must obtain a good understanding of the effects of different sail inventories and their capabilities.

SAIL PERFORMANCE: A CRITICAL EDGE

For the 2011–12 Volvo Ocean Race, Emirates Team New Zealand — a world leader in yacht racing — designed a boat backed by advanced technologies from ANSYS, with financial sponsorship provided by Spanish footwear manufacturer Camper. Team New Zealand has used ANSYS software for the past 14 years, achieving considerable success in applications of both aerodynamic and hydrodynamic design.

To understand the contribution of sail designs throughout the 39,000-mile Volvo campaign — and to feed this data back into the overall yacht design process — Emirates Team New Zealand engineers subjected these designs to an intensive combination of wind-tunnel testing and computational fluid dynamics (CFD) simulations using ANSYS CFX software.

With only a four-month design window for the entire yacht, the team placed a premium on using engineering simulation software from ANSYS to obtain fast and reliable CFD results. To

PHOTO: CHRIS CAMERON/CAMPER2011-12/VOLVO OCEAN RACE S.L.U.

establish a physical basis for CFD simulations, Emirates Team New Zealand engineers began their sail studies in a unique twisted-flow wind tunnel at the University of Auckland. Over the course of three days, they carried out 216 tests, fully covering the team's initial sail inventory across the range of wind speeds and angles expected during the race.

The team mounted small cameras on the model yacht, looking up at the sails. Using a specialized tool called VSPARS that calculates sail shape and position in real time, the designers were able to output full details about each sail stripe and rig deflection to an on-deck display. The digital images were used to generate accurate sail shapes to seed subsequent CFD analyses. They focused on getting a large collection of data for a wide range of sails, so engineers could quickly get a velocity prediction program (VPP) up and running — and understand the implications for the yacht's overall design.

RAPID RESULTS FUELED BY HPC

Next, block-structured grids were generated using ANSYS ICEM CFD Hexa. This approach produced a high-resolution boundary layer mesh around the mast and sails, while retaining manageable grid sizes, on the order of 8 million to 10 million cells. Since grid generation can be time consuming, this process was automated to increase productivity. By using Perl scripting running ICEM CFD and CFX-Pre in batch mode, the entire process took approximately 10 minutes for a 9 million-cell grid.

Emirates Team New Zealand engineers then relied on high-performance computing resources to each day evaluate dozens of sail combinations under a multitude of wind conditions. The team's Dell cluster is the largest private-sector computational cluster in New Zealand. It is optimized for CFX simulations with a smart queuing system that has increased Emirates Team New Zealand's design efficiency considerably.

As the hull and appendage design proceeded, the sail design process continued as well. Emirates Team New Zealand engineers used CFD studies to continuously refine the aerodynamic elements of the sails, to best suit the overall hull package being developed for both on- and under-water elements. This was a repeated, closed-loop process to ensure that the yacht's hull and appendages matched with the sail inventory.

ANSYS technology allowed Emirates Team New Zealand designers to analyze more than 2,000 different sail shapes and trims.

PHOTO: CHRIS CAMERON/CAMPER2011-12/VOLVO OCEAN RACE S.L.U.

Physical tests in a twisted-flow wind tunnel produced images that seeded subsequent CFD simulations of sail performance.

Since the eight sails on board the Camper yacht have to perform as promised under any wind conditions, robust design is critical. While physical tests in the twisted-flow wind tunnel were excellent at comparing the sails' overall performance, it could be difficult for engineers to determine why one sail performed better than another — and what design modifications could bring even further gains. CFD simulations, on the other hand, demonstrated exactly how and where performance gains could be made. Simulations were precise and repeatable, allowing for rapid parametric analyses. Subtle changes in trim and sail shape significantly improved the overall VPP results and ensured that the sails would work in tandem with other yacht components to deliver the highest possible performance for the entire boat.

TOOL FOR PREDICTABLE PERFORMANCE

The range of shapes and sizes associated with sail design is very broad. Designs for upwind sailing are very different from the ones used in downwind conditions.

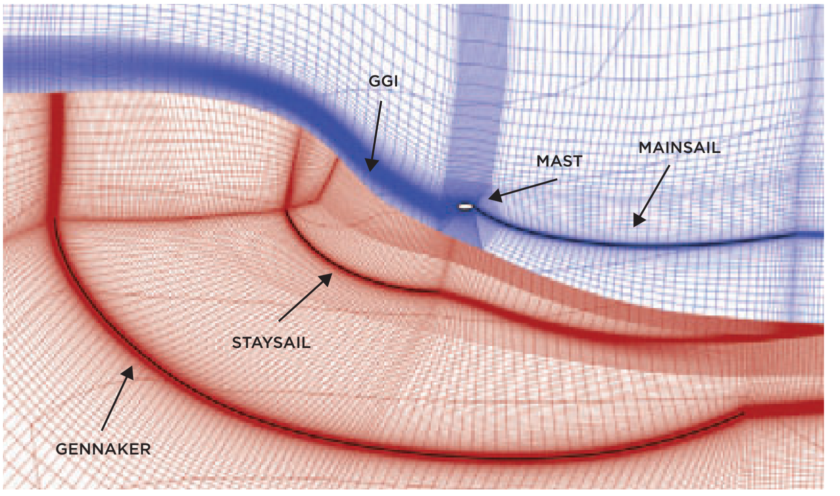
Sails must be smaller under conditions of very high wind velocity. Whatever the size and shape of the sails, the key to predicting sail performance is accurately simulating turbulence and behavior of the boundary layer close to the sail surface. This requires well-validated turbulence-modeling capabilities.

Emirates Team New Zealand's sail designers find the k-ω shear stress transport (SST) turbulence model invaluable in conducting daily aerodynamic studies. Key to efficient analysis of sails is achieving good performance across a range of flows, from low to high shear, without requiring changes to mesh topology. Automatic wall functions in the CFD software are also important, as they allow homogeneous turbulence modeling across different resolution meshes for the various sail components. The implementation of Menter's SST model within CFX provides design accuracy, robustness and speed — and the technology remains in a class of its own for yacht sail analysis.

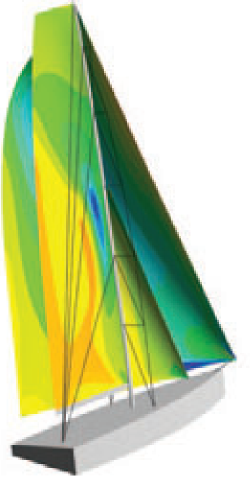
Another important facet of sail design is analyzing deformation in the sail. In upwind situations, in which the sails are relatively flat and stiff, aerodynamic pressures can be obtained from a panel method. But for large spinnakers used in downwind conditions, accurately mapping deformation of the sail and subsequent change in aerodynamic performance is critical. To accomplish this, the design team coupled the ANSYS software's aerodynamics capabilities with the sail manufacturer's in-house structural analysis code.

The smooth workflow from geometry to ANSYS ICEM CFD meshing software to the CFD solver allowed the designers to rapidly perform a large number of simulations. The results provided information on sail performance under varying aerodynamic conditions, which was fed back into the structural design of the sails. Throughout preparation for the Volvo Ocean Race, designers performed coupled fluid–structure interaction assessments, an iterative loop in which CFD analysis provided a new sail shape, which was then subjected to structural analysis.

The team's short design cycle and tight budget placed a premium on fast turnaround, from initial concept through final analysis. In meeting these demands, the generalized grid interface capability in CFX proved invaluable, allowing high-quality ANSYS ICEM CFD hex meshes to be built around different sail components — and often re-used when designers were making minor trim changes. In all, ANSYS technology allowed Emirates Team New Zealand designers to analyze more than 2,000 different sail shapes and trims in developing sails for the 2011–12 Volvo Ocean Race.



Block-structured grids of the sails generated based on images from wind-tunnel tests. To retain high-quality cells and a simple blocking topology, Emirates Team New Zealand engineers made use of generalized grid interfaces (GGIs).



The CFD simulations included sails, boom, mast and approximate hull geometry. CFD results allowed the team to model rigging windage by sampling velocities at discrete locations along each rigging element — and then pinning these elements to the CFD geometry.

CONFIDENT COURSE

As Camper Skipper Chris Nicholson eases through the doldrums in search of wind during the challenging months of the Volvo Ocean Race, ANSYS simulation software is probably the last thing on his mind. As the Camper yacht blasts through 10-meter waves deep in the Southern Ocean, the sail trimmers are unlikely to be thinking about the supercomputer sitting in a carefully climate-controlled server room in Auckland, relentlessly churning through sail design iterations. Nevertheless, ANSYS software helped to set a confident course toward victory in the Volvo Ocean Race.

A successful Volvo campaign lies in the confluence of a myriad of fields, from engineering design and analysis to sailing skill and sound meteorology. Using ANSYS fluid dynamics has removed a lot of the guesswork from Emirates Team New Zealand's sail design process, allowing team members to be much more precise in their engineering and performance analyses. That has led to a very robust sail design and a high level of integrity for sails when subjected to uncertain conditions once the yacht is under way.

CFD analysis alone can't guarantee a win in the demanding Volvo Ocean Race, but ANSYS technology helps to drive reliable sail performance — and an overall yacht design that maximizes the potential of those all-important sails.

GOOD VIBRATIONS

By Stefan Mohr, R&D Manager, Predevelopment
HEAD Sport GmbH, Kennelbach, Austria

Tennis champion Maria Sharapova using her HEAD racket

HEAD Sport delivers world-leading tennis racket performance with simulation.

The highly-competitive tennis equipment market is characterized by an extreme emphasis on performance and very short product cycles. HEAD Sport has succeeded in this market through a process of continual innovation, for which the company uses advanced simulation technology to make ever-increasing performance improvements. In the past, HEAD engineers used the trial-and-error method to improve racket design but were constrained by the amount of time required to build and test each prototype as well as the limited information yielded by physical testing. More recently, HEAD has made substantial performance improvements and accelerated the pace of innovation by using ANSYS simulation technology to evaluate virtual prototypes in far less time than was previously required. The result is innovative designs that help world-class athletes win, such as the YOUTEK™ IG SPEED MP 18/20 racket used by Novak Djokovic to win three Grand Slam tournaments in 2011, vaulting him to the position of the world's number-one player.

MEETING STRENGTH AND STIFFNESS CHALLENGES

One of the basic challenges of tennis racket design is creating a frame that weighs only a few hundred grams yet can resist the tension force of 200 Newtons on each of its 76 holes, for a total force of 15,200 Newtons, or 3,417 pounds — the weight of a large automobile. HEAD developers have addressed this challenge by using carbon and glass-fiber reinforced polymer (FRP) materials with specially engineered orthotropic properties that provide the stiffness needed for racket performance and the strength required for durability. Furthermore, a top-level player generates a serve speed velocity of 250 kilometers per hour, producing a peak force of 600 Newtons applied perpendicular to the strings for a period of 3 or 4 milliseconds. During this period, the ball accelerates from 0 to 250 kilometers per hour. HEAD engineers use dynamic analysis to simulate this impact by applying the force of the ball to the strings in the form of a force-time function. In some cases, they simplify the calculation by converting the dynamic force to a static force.

One of the problems with previous racket design methods was that when a prototype racket passed a stiffness test, engineers never knew how close it came to collapsing. Using ANSYS Mechanical, HEAD engineers can determine the stress on each individual layer of the composite laminate to a high level of precision and add or subtract small amounts of material to provide exactly the strength needed without any wasted material.

PHOTO: ©HEAD/PAUL ZIMMER

The result is that, today, HEAD can build a frame weighing only 200 grams that withstands both string tension and ball impact forces. In reality, a 200-gram racket is too light for playing tennis, but it gives engineers freedom to add other features to the racket.

MODAL SHAPES AND FREQUENCIES

Modal shapes and frequencies are becoming increasingly important in racket design because the way a racket "feels" is passed to the hand via vibrations. For example, a professional tennis player once came to HEAD and said that he was unhappy with a racket that had been built specially for him but could not explain why. HEAD engineers used ANSYS Structural to evaluate the modal shapes and frequencies of the racket and made changes to give it just the right feel. Over time, researchers have identified specific modal shapes and frequencies that provide a solid feel and stable racket, so they can address the feel of the racket from the beginning of the design process.

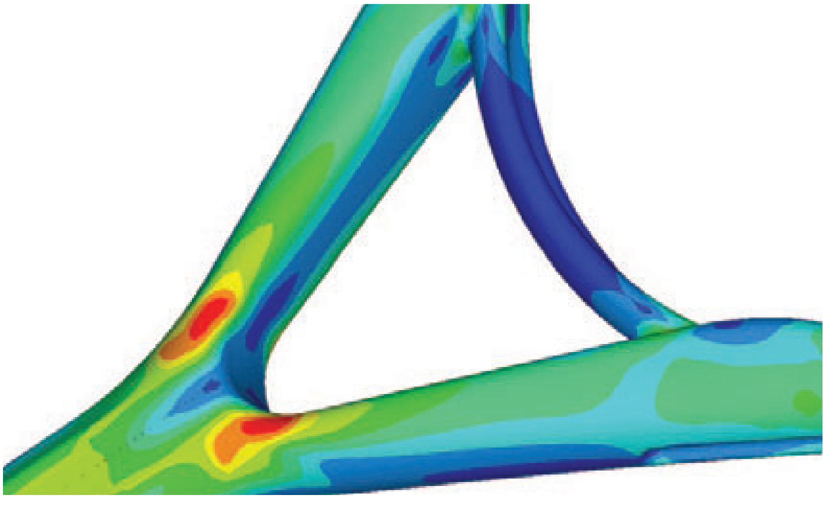
It's also important to ensure that the proposed racket design does not have any modal harmonics that might be excited during normal use. For example, the first mode of vibration of the racket strings is typically around 500 Hz. Engineers analyze modal harmonics of proposed racket designs with ANSYS software to be sure that higher modes are not too close to this number. Higher modes play an important role in the sound generated by the racket. HEAD engineers optimize the mode shapes and frequencies to produce a sound that will provide the right feedback to the player.

The tennis racket's bumper is a plastic part that fits on the head to protect against abrasion and impact. The bumper is injection-molded as a 2-D shape, then bent into a complex 3-D shape during assembly. This part is engineered so that the pre-stresses induced when it is bent into its final shape will hold it in place. If the pre-stress is too high, the bumper will buckle; if too low, the bumper will come loose. Engineers use ANSYS Mechanical to simulate the assembly process and ensure that just the right amount of pre-stress is incurred.

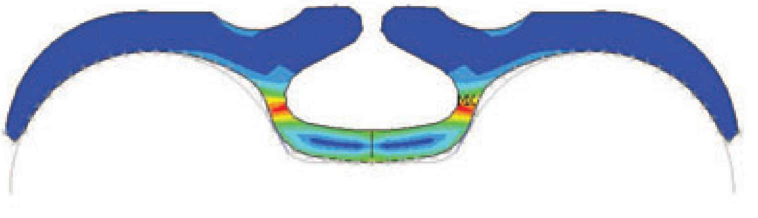
DESIGN OPTIMIZATION

When HEAD engineers first began using simulation, they evaluated a single design iteration, made changes based on the results, and ran a new simulation. It took approximately one week to evaluate the performance of each iteration. Today, they frequently use a nonlinear optimizer with a genetic algorithm to generate design iterations then run ANSYS Mechanical simulations automatically in a batch process to identify the optimal solution. The typical goal is to find the lightest structure with the highest stiffness and strength that meets other design constraints. Recently, engineers used optimization to evaluate 1 million design concepts in about a week to improve the design of a composite structure. The ability to analyze such a large number of design alternatives made it possible to optimize performance to a level that would never have been possible in the past. The optimizer frequently identifies design concepts that are such a departure from traditional engineering

It formerly took HEAD one week to evaluate the performance of a single design iteration. Today, engineers can evaluate 1 million design concepts in approximately a week.

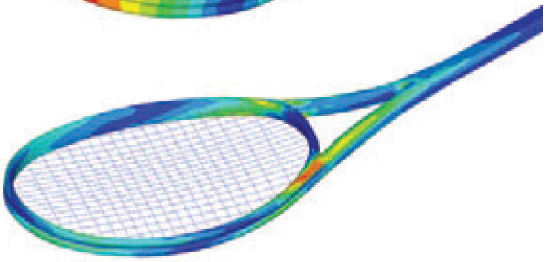


Shaft area of a tennis racket under bending load. The stress level is too high, so engineers reinforced the design.

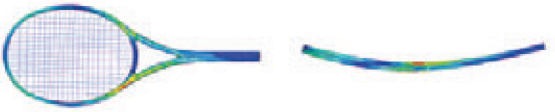


Simulation of bumper verifying that it has the right pre-stress to fit well on the frame

First bending mode of a strung tennis racket in elongation. Hitting the ball at the blue node line on the strings does not excite this mode. This is the sweet spot of the racket.

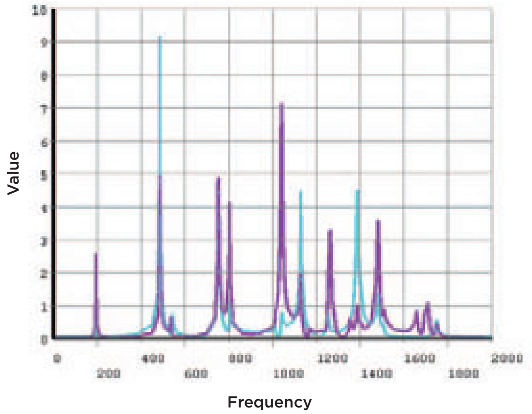


First bending mode in tension



Second bending mode of a strung tennis racket

Torsion mode of a tennis racquet illustrating how a non-center hit introduces torsion



Harmonic analysis shows frequency spectrum of a strung tennis racket with a special excitation load. The spectrum shows all excited natural modes up to 2kHz. Blue represents the center strings and purple the off-center strings.

Simulation has played a significant role in the dramatic improvements that HEAD has achieved in the performance and durability of its tennis rackets over the past decade.

practices that the team would never have thought them applicable.

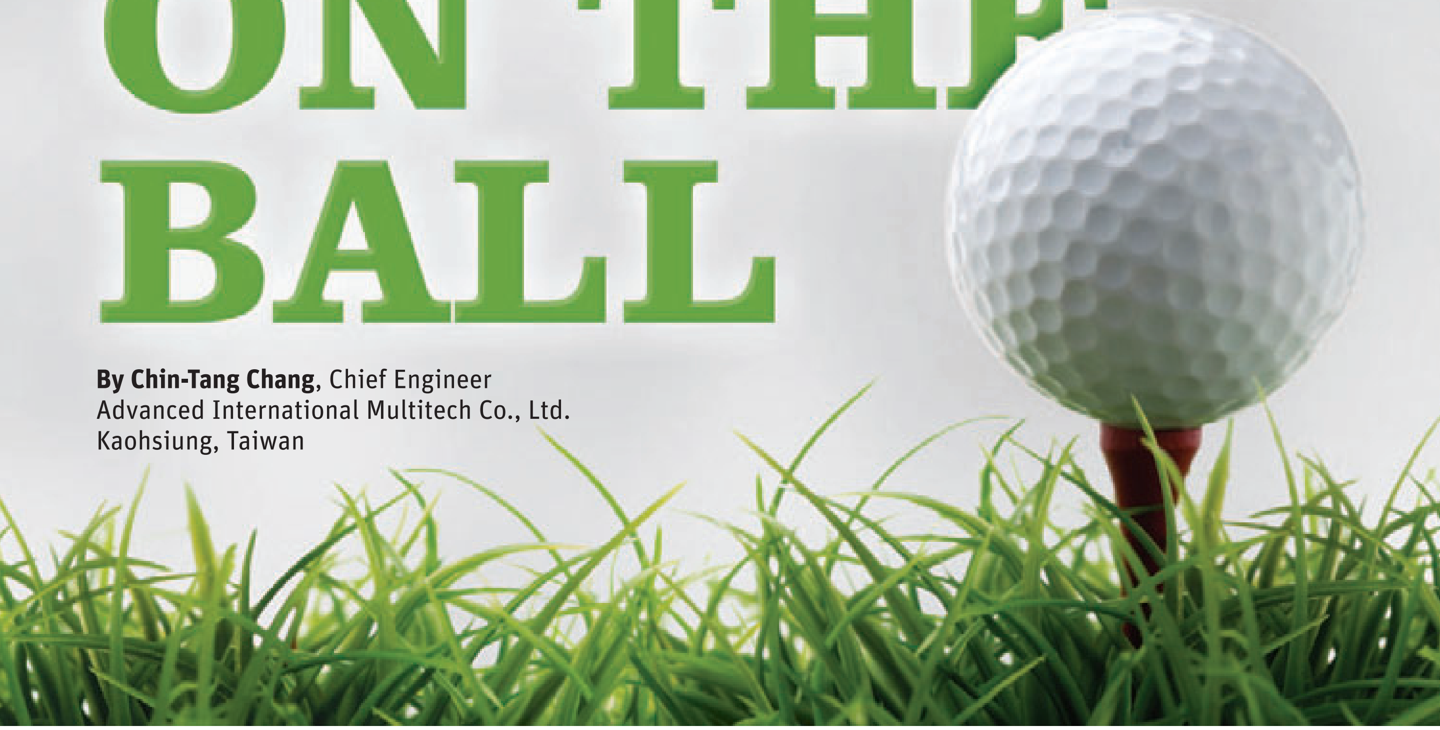
HEAD has developed a challenging series of quality tests that measure stiffness and strength, which every racket must pass before entering the market. For example, the racket must withstand a drop test onto concrete without any damage. The team at HEAD has developed a series of ANSYS Parametric Design Language (APDL) scripts that automatically simulate each test on a new design. The scripts import the CAD model of the racket, generate an FE mesh, orient the mesh, perform the five tests, and generate a report that provides detailed results. These scripts enable engineers to quickly and easily evaluate each proposed design from a quality standpoint, all from the early stages of the product development process.

Simulation has played a significant role in the dramatic improvements that HEAD has achieved in the performance and durability of its tennis rackets over the past decade. The detailed and accurate design information provided by ANSYS Mechanical combined with the experience of HEAD engineers has made it possible to produce extremely lightweight designs that can withstand the enormous forces generated by string tension as well as the impact of the ball on the racket during the serve. Engineers also use ANSYS Mechanical to tailor other performance characteristics, such as the vibration and sound generated by its rackets. The results can be seen in the performance of HEAD rackets in the hands of players such as Djokovic, who after a remarkable series of victories over the world's top players is widely considered to be one of the greatest tennis players of all time. ∧

HEAD Sport is supported in this work by ANSYS channel partner CADFEM.

ON THE BALL

By Chin-Tang Chang, Chief Engineer
Advanced International Multitech Co., Ltd.
Kaohsiung, Taiwan



Finite element analysis helps to improve golf ball rebound velocity by 5 percent, resulting in longer drives in a game measured in inches.

In the highly competitive golf and cycling sectors, differences of just a few percentage points in equipment performance can make the difference between a successful product and an also-ran. Advanced International Multitech Co., Ltd., is a leading producer of golf balls, golf clubs and biking equipment. The company uses ANSYS LS-DYNA software to evaluate a wide range of club head and golf ball designs to improve equipment performance. For example, the business optimized a new line of clubs to make the heads thinner but just as strong as the previous version. By improving both club heads and balls, the company has improved rebound velocity in its current line of products by 5 percent. The resulting improvement in driving distance contributed to a successful product launch, which in turn helped the company to achieve nearly 40 percent sales growth in its recently completed fiscal year.

Advanced International Multitech produces balls for many world-famous brands. The company produced approximately 36 million golf balls last year for Callaway, its largest customer. Advanced International Multitech began operation of an $11 million (U.S.) plant in southern Taiwan in 2010. It is currently running one ball production line in the new plant, increasing capacity from 100 million to 125 million balls per year. Once fully equipped, the plant will provide the capacity to vault the firm into position as the number-one golf ball manufacturer in the world.

EVALUATING DESIGNS

Over the last several decades, substantial improvements have been made in drivers and other golf clubs so that they hit the ball farther and do not require a perfectly on-center strike to achieve good results. To limit the distance that a golf ball can travel, the United States Golf Association (USGA) has placed limitations on the coefficient of restitution (COR), a number between 0 and 1 that represents the percentage of energy transferred from the club to the ball during impact. A COR of 1 represents a perfectly elastic collision, with all of the club's energy being transferred to the ball. A COR of 0, on the other hand, means that none of the energy in the club is transferred to the ball. The USGA limits COR to 0.830. The most comprehensive method of measuring COR is to use an air cannon to fire the golf ball at a club head and measure the ball's before-and-after velocity.

This USGA restriction has forced manufacturers of balls and clubs to re-evaluate designs and determine methods of optimizing performance within these constraints. Golf is a game of inches, so tiny differences in performance between different balls and clubs can make the difference between winning and losing a match — and between success and failure in the marketplace. Physical testing is the gold standard for determining performance differences. But it is expensive and time consuming, as it requires building a prototype of every potential design of interest. Achieving a superior design in today's highly competitive market often requires evaluating thousands of potential designs, so physical testing is not a practical approach for early design stages when many different alternatives are being considered. Of course, physical testing is still essential in the later stages of the design process, when the choice has been narrowed down to a few options.

APPLYING SIMULATION

Golf equipment manufacturers have begun using simulation as their go-to design tool. But the impact of a club head hitting a golf ball is a challenging simulation task because it produces extreme changes in loading conditions and severe deformations in a very short period of time. The most popular finite element analysis software programs use the implicit method, which has difficulty in accurately predicting extremely nonlinear events. The explicit method, on the other hand, captures the physics of short-duration events for products that undergo highly nonlinear, transient events. Algorithms based on first principles accurately predict complex responses, such as large material deformations and interactions between bodies and fluids with rapidly changing surfaces.

ANSYS LS-DYNA is a leading explicit finite element analysis program for the simulation of short-duration events. It has been used in most of the pioneering research aimed at simulating golf ball and club head impacts by most golf equipment companies. The value of this software package has been increased by incorporating it within the ANSYS Workbench environment, which reduces the time required to perform analysis through bidirectional CAD connectivity, geometry cleanup tools, automatic meshing, and quick and simple definition of initial and boundary conditions.

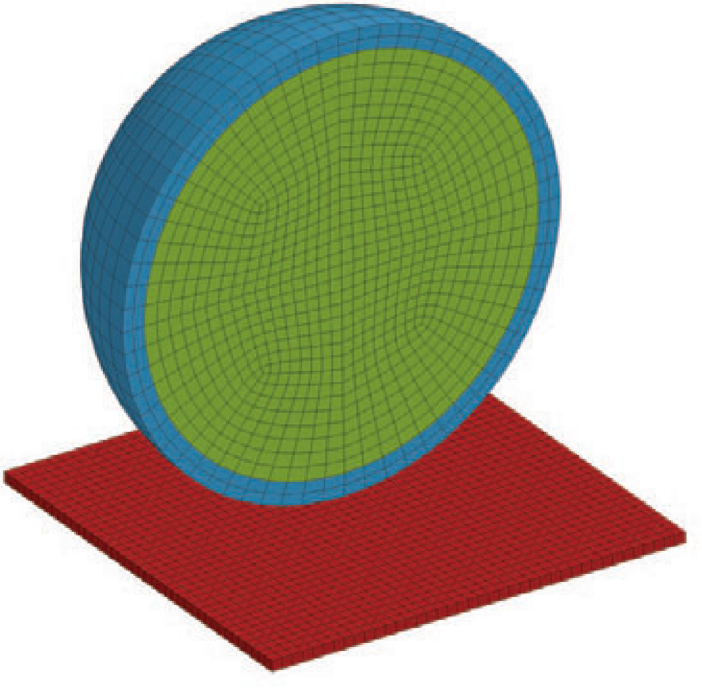
Advanced International Multitech engineers recently used ANSYS LS-DYNA to design a new club head and golf ball. Engineers began by performing physical testing to evaluate the properties of the different materials that were being considered for use in the ball. They modeled the ball as a three-piece system — cover, mantel and core — with Mooney-Rivlin hyperelastic properties and about 10,000 hexahedral elements. The team modeled the club head as titanium alloy plate with about 250,000 tetrahedral elements. The speeds used for the initial club velocity were representative of typical male swing speeds, ranging from about 40 to 50 meters per second (90 to 110 miles per hour).

The analysis investigated different dimensions and material properties for the ball's cover, mantle and core. The same analysis was used to look at the effects of club head geometry. The model took about one hour to run. Advanced International Multitech engineers ran parametric analyses over a wide range of geometries and boundary conditions. The simulation results showed ball velocity at a few inches past the point of impact, the same point at which ball velocity is measured in physical testing. This made it very convenient to evaluate the accuracy of the simulation by comparing the results with experiments. The simulation predictions closely matched physical testing.

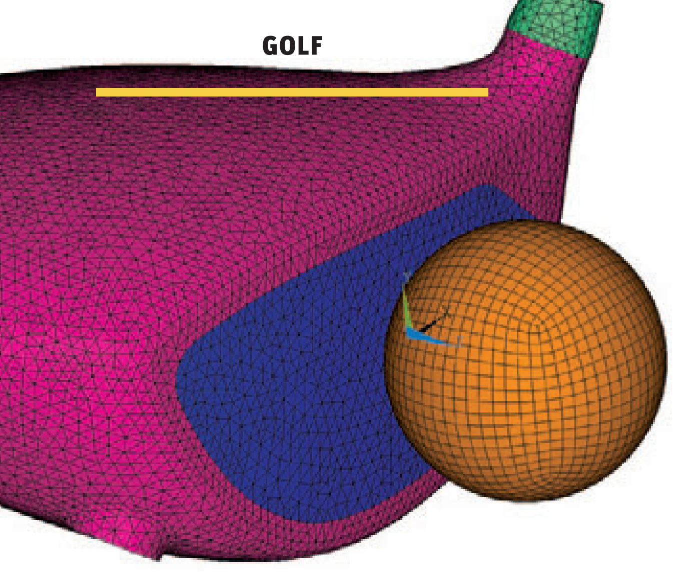
INCREASING DISTANCE

By evaluating a wide range of ball materials and club head geometries, Advanced International Multitech engineers increased the rebound velocity of the ball by about 5 percent. This improvement

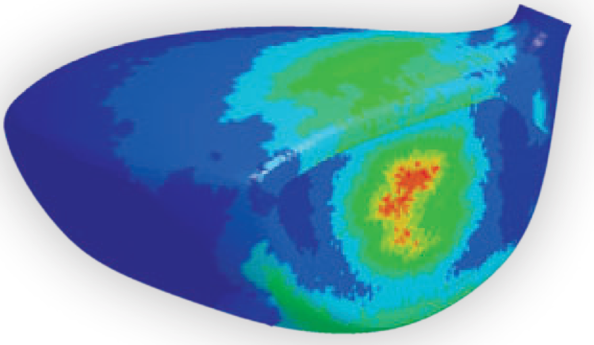
By evaluating a wide range of ball materials and club head geometries, Advanced International Multitech engineers significantly increased the travel distance of its golf ball.



Finite element model of golf ball, with core in green and cover in blue



Simulation of impact of club head with ball in ANSYS LS-DYNA



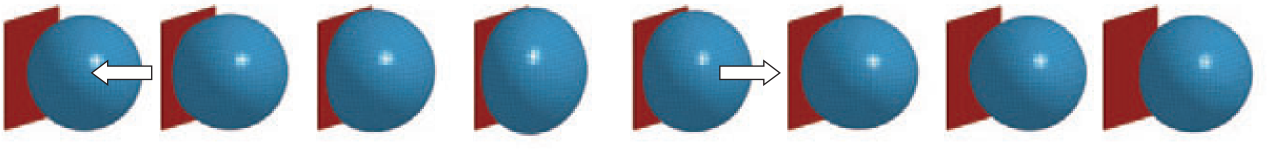
Stress analysis results on club head

Explicit analysis with ANSYS tools has helped the company provide market-leading equipment performance and has played a major role in the company's rapid increase in market share.

directly translates into a significant increase in the distance traveled by the ball for a given club head velocity. Explicit analysis with ANSYS LS-DYNA has helped the company provide market-leading performance for its golf balls and club heads and has played a major role in the company's rapid increase in market share over the past decade.

Advanced International Multitech also uses ANSYS Structural software, which is based upon the implicit method, to optimize the design of bike frames and other composite structural components. Typically, the objective is to minimize the weight of components while providing enough strength to meet safety requirements. The company's engineers use ANSYS Structural to evaluate stresses and deformations of alternative structural component designs by varying part geometry and ply layout; they then iterate to an optimized design that weighs approximately 10 percent less than the previous generation while providing the same level of strength.

Using simulation to gain insight into the way materials respond during complex or severe loading, Advanced International Multitech gained a competitive advantage and improved its products' design.



Golf ball impact analysis with initial velocity of 45 meters per second

CONSTRUCTION

Dry Run

By Bert Blocken, Professor; **Twan van Hooff,** Ph.D. Student; and **Marjon van Harten,** M.Sc. Student, Eindhoven University of Technology Eindhoven, The Netherlands

AFAS stadium in Alkmaar

PHOTO: AZ NV

Simulating wind and rain around a stadium determines the best design for keeping spectators dry.

When developing a sporting venue, designers must consider a large number of factors. Two important overall considerations are spectator comfort/safety and how the stadium itself could affect the play of the sport — and each has many facets. Anyone who has attended a sporting event in the rain will understand the dampening effect that this factor can have on event enjoyment.

In semi-open stadium design, two of the greatest challenges are limiting wind on the field to avoid affecting play and protecting spectators from wind-driven rain. Historically, stadium designers largely ignored these factors because they were unable to determine how their designs would perform until after the venue had been built. Simulation is changing that.

A team from Eindhoven University of Technology has conducted 3-D studies of stadium design using ANSYS Fluent computational fluid dynamics (CFD) software to demonstrate the important effects of the architecture on both wind flow and wind-driven rain. Simulation of 12 different stadium configurations showed the precise areas where wind-driven rain would penetrate the seats in each type of stadium as well as the speed and direction of wind on the playing field. These results can be used to improve the design of future stadiums as well as to diagnose and correct problems with existing stadiums — such as using special paint to protect seats that frequently get wet to reduce maintenance costs.

STADIUM DESIGN AND SIMULATION CHALLENGES

Many stadiums in Europe and elsewhere have an open design in which the roof covers only the stands. In most cases, the roof does not extend farther than the separation between the stands and the field, so wind-driven rain can reach a large area of seating, resulting in spectator

Results can be used to improve the design of future stadiums as well as to diagnose and correct problems with existing stadiums.

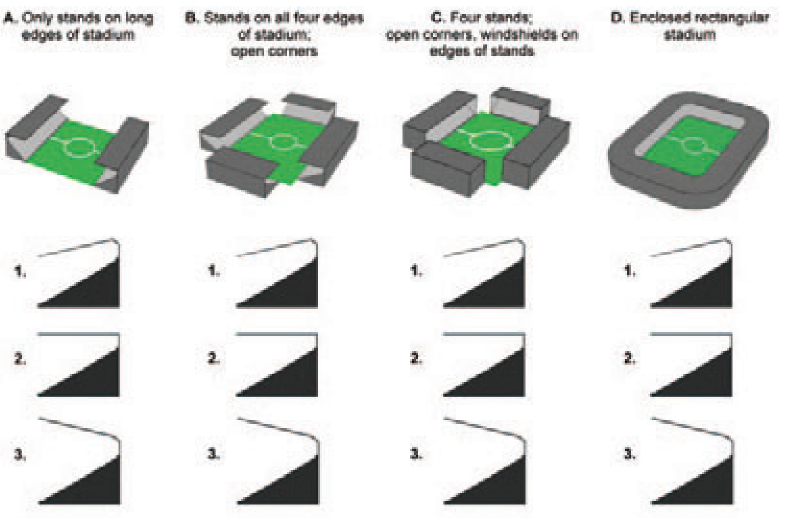
discomfort. To the best of the authors' knowledge, the first simulation of wind-driven rain in a stadium was published in 2008 [1]. This 2-D study was not able to capture the important effects of roof and stadium geometry. Simulation predicted the area of the stands that became wet in seven generic stadium designs. The study showed that roof design can strongly influence which areas stay dry and which become wet.

The new Eindhoven study went a step further by using 3-D CFD to capture 3-D flow patterns and provide a more realistic assessment of the effect of roof geometry on the area wetted by wind-driven rain [2]. One of the biggest challenges was modeling geometry that combines a very large computational domain with the need to model certain areas to a fine level of detail. Another challenge was the need to accurately model the wide range of sizes of potential rain droplets.

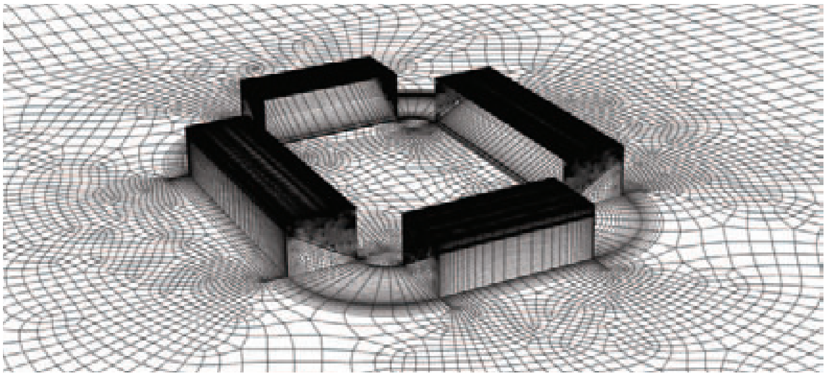
12 DESIGN ALTERNATIVES CONSIDERED

The stadium configurations in this study are based on the AFAS stadium in Alkmaar, The Netherlands, which seats 17,000 people and is the home of the AZ Alkmaar football team. Its exterior dimensions are 176.8 meters long by 138 meters wide by 22.5 meters high; it has a downward-sloped roof with an inclination of 13 degrees. The research team selected this stadium because of its unusual roof, which the architects intuitively chose with the goal of improving shelter from wind-driven rain. The team looked at four different stand arrangements: stands only on long sides of stadium (A), stands on all four sides (B), stands on all four sides with each end of the stands closed off (C), and a closed rectangular stadium (D). Researchers evaluated each stand arrangement with three different roof designs: upward sloped (1), flat (2), and downward sloped (3), yielding 12 different design configurations: A-1, A-2, A-3, B-1 and so on. Configuration D-3 matches the actual AFAS stadium design.

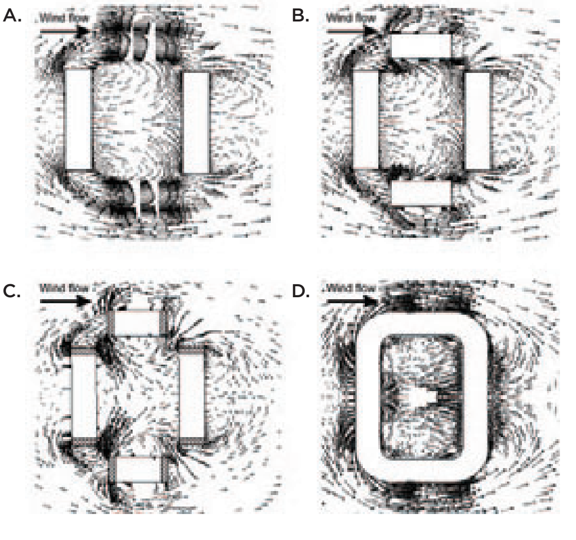
The difference between the largest- (1,100 meters) and smallest- (less than 0.1 meters) length scales in the domain created challenges in producing the computational grid. The Eindhoven team generated the master grid by creating a series of premeshed 2-D cross sections, then using a series of translation and rotation operations. The master grid contained hexahedral and prismatic cells. This approach provides full control of grid quality and resolution. Researchers used higher-resolution mesh at the deck of the stadium, while lower resolution was applied to outside the stadium. They increased the grid resolution in the vicinity of the roof for a more detailed prediction of wind flow. The team deleted designated meshed volumes from the master grid to produce the mesh for each of the 12 different stadium design variants.



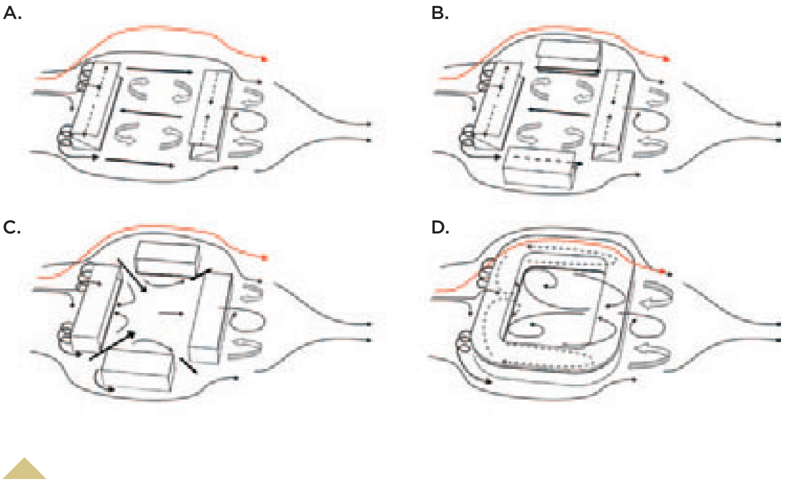
Four different stand configurations were evaluated with three different roof designs, for a total of 12 possibilities.



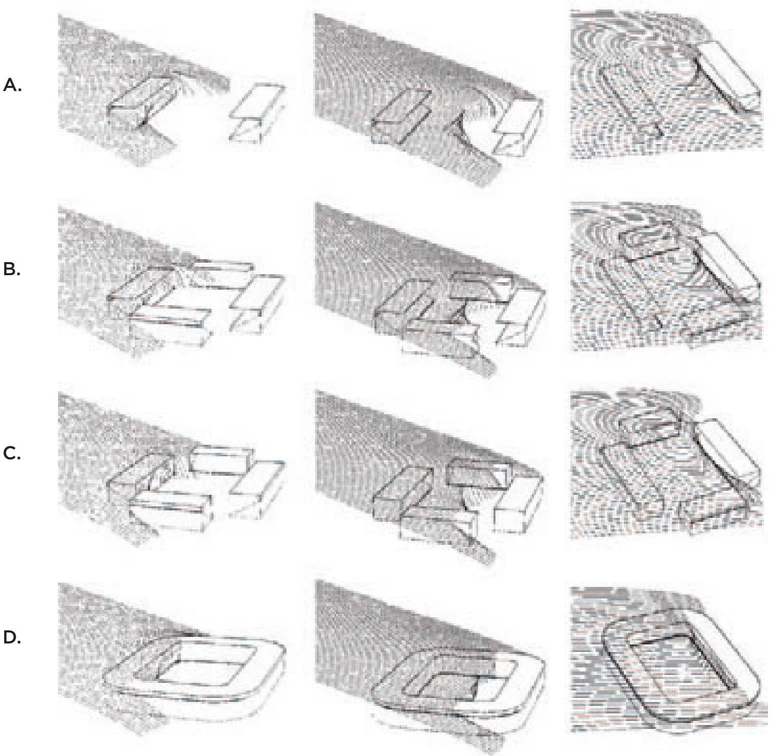
Increased grid resolution in the vicinity of the roof provided a more detailed prediction of wind flow.



2-D wind flow patterns one meter above ground for the four different stand configurations



3-D wind flow for four different stadium stand configurations



Raindrop trajectories for 1 mm raindrops and the same wind speed for four stadium configurations. Each column shows a different injection position for raindrops.

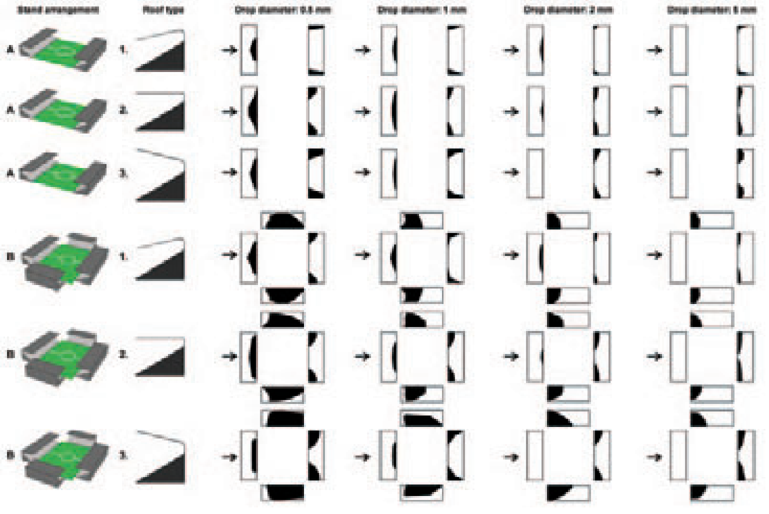
WIND-FLOW PATTERNS

Researchers calculated steady-state wind-flow patterns around the stadiums using the Reynolds-averaged Navier–Stokes (RANS) CFD technique with a realizable k-epsilon turbulence model. They based this choice on earlier extensive validation efforts of wind-driven rain on building facades and over various types of small-scale topography, including hills and valleys [3, 4, 5]. They determined the wind-flow patterns in the field at a height of one meter above ground level (chosen to determine the impact of wind conditions on the game) for each of the four stand arrangements described earlier with a flat roof. The results for configuration A-2 revealed two large counter-rotating vortices between the stands. These vortices are mainly driven by the corner stream shear layer originating at the corners of the upstream stand. Applying fluid flow modeling to configuration B-2 revealed results similar to A-2, except that the velocities in the upstream corners of the stadium are higher. The simulation results show that the stands in the short edges of the stadium have a relatively small influence on the flow because the side flow is not obstructed by these stands and can pass through the area between the stand and the roof. Closing off the sides of the stands in configuration C-2 eliminates the vortices and, instead, shows that streams from two different corners combine to generate two high-velocity jets directed toward the center of the stadium. Configuration D-2 has two vortices, but they are smaller and have a lower velocity than configurations A-2 and B-2 — so the flow pattern inside the stadium is considerably more complex. Additional small vortices exist above the field and underneath the roof.

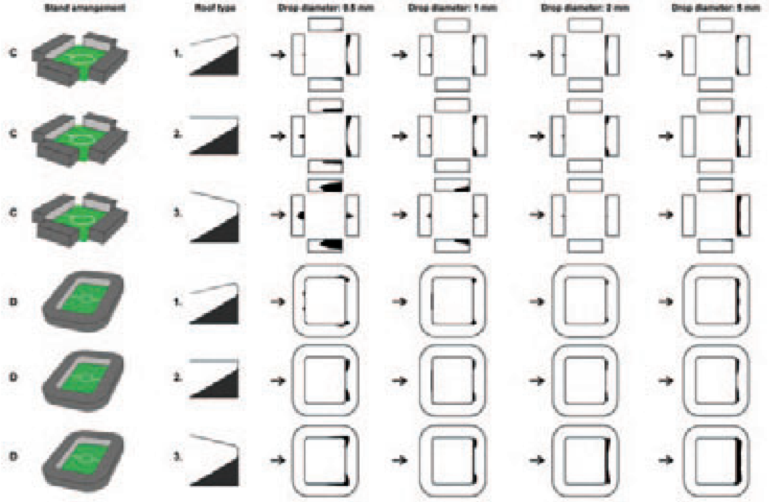
WIND-DRIVEN RAIN PATTERNS

Researchers determined raindrop trajectories by injecting raindrops of different sizes in the calculated wind-flow motion and solving their equations of motion. Raindrops with diameters of 0.5 mm, 1 mm, 2 mm and 5 mm were considered. The raindrops were injected from the top of the domain with the local wind speed as horizontal injection velocity and terminal velocity as vertical injection velocity. Raindrops of 0.5 mm diameter represent the median diameter for drizzle (rainfall intensity of 0.1 mm/h), 1 mm diameter represents the median diameter for a rainfall intensity of 1 mm/h (common Dutch rain), 2 mm diameter represents 10 mm/h (heavy shower), and 5 mm diameter is the largest raindrop in heavy rain showers. The areas of the stadium receiving rain were calculated based on the calculated raindrop trajectories for each stadium configuration. For configurations A and B, the large vortices between the upstream and downstream stand cause part of the upstream stand to be wetted, especially for smaller raindrop diameters that are more sensitive to the wind-flow pattern and can be more easily swept underneath the roof. On the other hand, with configuration C, the wetting of the two

CFD can provide detailed insight into wind-flow patterns and wind-driven rain distribution in a wide range of stadium configurations.



Overview of wetted stand areas for stand arrangements A and B with three different roof types



Overview of wetted stand areas for stand arrangements C and D with three different roof types

stands is limited, especially for the downward-sloping roof. The closed ends of the stands effectively shield the stands from rain, especially for upward-sloping and flat roofs. Configuration D provides only limited wetting at the rows of the stands closest to the field.

Wind flow and wind-driven rain affect spectator comfort and fair competition, but until recently stadium designers have had no way to determine the performance of design alternatives based on these conditions. This study demonstrates that CFD can provide detailed insight into wind-flow patterns and wind-driven rain distribution in a wide range of stadium configurations. Much further work remains to be done, such as applying design optimization to determine the ideal roof angle to maximize the comfort level of the spectators.

References

[1] Persoon, J.; van Hooff, T.; Blocken, B.; Carmeliet, J.; de Wit, M.H. On the Impact of Roof Geometry on Rain Shelter in Football Stadia. *Journal of Wind Engineering and Industrial Aerodynamics.* **2008**, Vol. 96, Issue 8–9, pp. 1274–93.

[2] van Hooff, T.; Blocken, B.; van Harten, M. 3D CFD Simulations of Wind Flow and Wind-Driven Rain Shelter in Sports Stadia: Influence of Stadium Geometry. *Building and Environment.* **2011**, Vol. 46, Issue 1, pp. 22–37

[3] Blocken, B.; Carmeliet, J. Validation of CFD Simulations of Wind-Driven Rain on a Low-Rise Building. *Building and Environment.* **2007**, Vol. 42, Issue 7, pp. 2530–2548.

[4] Blocken, B.; Poesen, J.; Carmeliet, J. Impact of Wind on the Spatial Distribution of Rain over Micro-Scale Topography — Numerical Modelling and Experimental Verification. *Hydrological Processes.* **2006**, Vol. 20, Issue 2, pp. 345–368.

[5] Blocken, B.; Carmeliet, J.; Poesen, J. Numerical Simulation of the Wind-Driven Rainfall Distribution over Small-Scale Topography in Space and Time. *Journal of Hydrology.* **2005**, Vol. 315, Issues 1–4, pp. 252–273.

Germany Rocks on Water

By Nicolas Warzecha, Simulation Engineer (CFD Expert)
Institute for Research and Development of Sports Equipment
Berlin, Germany

Advances in boat equipment design via simulation help to sustain Germany's Olympic excellence.

In the pantheon of Olympic sports, boating competitions have been around since the very beginning. Rowing was to have its debut in the 1896 games in Athens but was delayed until the 1900 games in Paris by bad weather. Canoeing and kayaking were first contested in the Berlin games of 1936. Over the course of modern Olympic history, no country's teams have won more medals in the rowing, canoeing and kayaking events than the German teams. This winning tradition has been upheld in recent years due in part to engineering innovations originating from the Institute for Research and Development of Sports Equipment, which is known by its German acronym, FES.

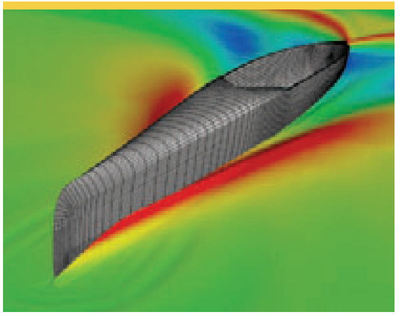
From its first successes — creating more aerodynamic bicycles for the 1988 games in Seoul, South Korea, to more recent efforts — designing more streamlined bobsleds for the 2006 Winter games in Turin, Italy — FES has continued to achieve international success by developing sports equipment in accordance with the latest engineering technology. Numerical simulation has been a part of the company's toolbox for over a decade as FES engineers have used ANSYS CFX to optimize the fluid dynamics for different classes of racing boats. Their work contributed to several medal-winning performances at the 2008 Beijing games, and it is helping the German team prepare for the 2012 London games.

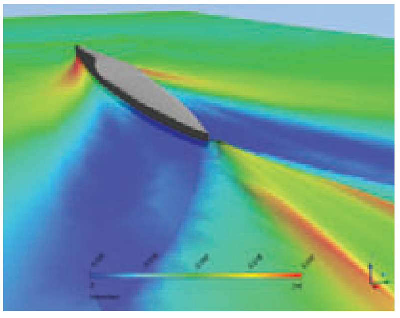
The focus of recent simulation efforts at FES has been on some of the smaller boats, specifically the pairs (two-person) sweep rowing shell and K-1, K-2 and K-4 (one-, two- and four-person) racing kayaks. From both construction and simulation points of view, FES decided that it made sense to analyze these smaller boats; even though they require less material to build, they are more difficult to optimize. For example, the K-1 kayak must be no longer than 17 feet (5.2 meters) with a minimum weight of 26 pounds (12 kilograms). With this relatively small amount of boat surface to modify while taking into account the surrounding air flow, water flow, free surfaces and complex boundary conditions, simulation was a necessary and crucial part of the design process.

FES engineers have used ANSYS software to optimize the fluid dynamics for different classes of racing boats, contributing to several medal-winning performances at the 2008 Beijing games and preparing for the 2012 London games.

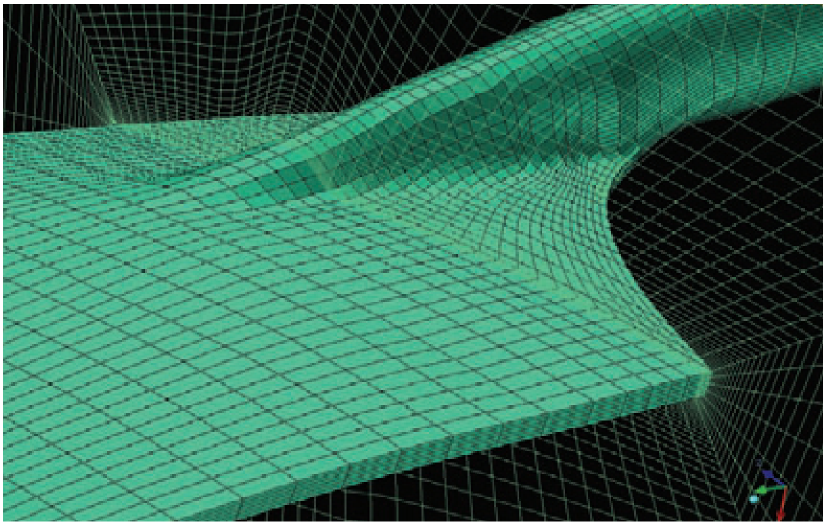
In the case of the pairs rowing shell, drag reduction was at the top of the list for optimization. In a sport timed to the

PHOTO: CAMERA4/THONFELD



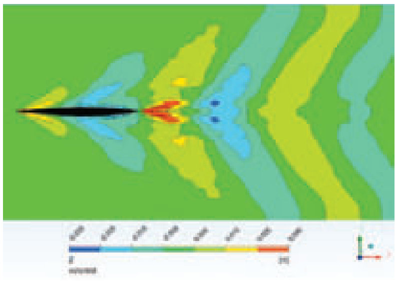


Contours of the free surface at steady state colored by wave height: canoe single (C1) boat (top); kayak single (K1) boat (bottom)



Close-up of surface mesh and surrounding fluid zone mesh for rowing oar blade

Sequence of blade stroke as the blade contacts water: rowing oar (top); kayak paddle (bottom)



Overhead view showing wave pattern produced on water surface by a K1 boat. Colors indicate the height above (yellow and red) or below (blue, light blue) initial undisturbed water (green).

thousandth of a second, even a 1 percent improvement can make a crucial difference in a close race. The main components that cause drag are friction on the wetted surface and the formation of waves during the rowing process. Other parameters that come into play include the location of the center of gravity and the volume of water that the boat displaces. During simulation efforts, it became apparent that the initial assumption of a fixed boat position in the water was not correct, as results deviated significantly from towing-tank experiments on scale models.

To correct this issue, FES engineers used the CFX Expression Language, with its capability to integrate user-defined FORTRAN™ subroutines, to automatically determine the variable floating position of the boat, which was dependent on the actual velocity during the calculation. For every iteration, the new floating position of the boat was calculated and the mesh deformed. Using a structured mesh generated by ANSYS ICEM CFD that contained 3 million cells, engineers could perform extensive transient simulations with reasonable calculation times.

The FES team expanded their efforts through interaction with experts at CFX Berlin, an ANSYS channel partner, which provided support for meshing, simulation, and setting up the high-performance computing (HPC) cluster at FES for both Linux® and Windows® hardware. Transient simulations of moving boats — once prohibitively expensive — could now be accomplished in just two or three days.

Using HPC, the FES team was able to efficiently consider up to 20 different virtual designs per boat class, and from those 20 designs engineers gained enough confidence to build a single prototype for testing. Since flatwater sprint courses have generally the same conditions around the world, the goal was not to exactly reproduce the towing-tank test results with simulation; rather it was to understand the relative difference in performance between hull designs under the same boundary conditions. As a result of these numerical investigations, optimizing hull shapes that reduce the friction on the wetted surface and the formation of waves — and hence the overall drag — should assist the German team in its quest for medals.

Beyond boat hydrodynamics, the other main process that FES has studied is the oar and paddle stroke. Both rowing and kayak strokes consist of a drive phase, in which the boat is propelled,

Leveraging its HPC access, the FES team considered up to 20 different virtual designs per boat class, and from those engineers gained enough confidence to build a single prototype for testing.

and a recovery phase, during which it slows down. The resulting cyclic velocity is responsible for changes in immersion, trim, drag resistance and lift. Though numerous parameters and influencing factors strongly depend on the individual habits of the athletes, some promising first steps have been taken using simulation to optimize the designs of oar and paddle blades.

Throughout all these efforts at boat optimization, the athletes themselves have been closely involved with the development, since changes in equipment can necessitate adjustments in how Olympic team members need to perform to realize the full benefits of the simulation-driven design changes. This collaboration is essential, as top performers can be reluctant to risk changing routines or well-honed rowing and paddling techniques in advance of a showcase event.

Extending the success of German athletes on the water was an excellent incentive for FES engineers in overcoming technical challenges and gaining valuable insights into oar, paddle and boat performance.

BUSINESS IS BOOMING

Multiphysics simulation helps to design a thermal diffuser that converts explosion into long-lasting heat.

By **Jean-Philippe Guillemin**, R&D Engineering, Davey Bickford, Héry, France, and **Stéphane Leconte** Engineering Student, Institut Française de Mécanique Avancée, Clermont-Ferrand, France

Pyrotechnics are used for more than fireworks. Pyrotechnics are employed in products for the defense and aviation, security system, mining, seismic, quarry and construction sectors. Development of pyrotechnic applications making use of controlled explosions presents enormous engineering challenges. A recent application in the security field required using pyrotechnic properties lasting a few milliseconds and generating temperatures greater than 900 C to heat a liquid for 30 minutes to about 100 C. The challenge for Davey Bickford's engineering team was to design a heat exchanger that could do the job at a reasonable cost. This required tracking the heat flux generated by the device through a solid thermal diffuser into the liquid to be heated.

Davey Bickford used the ANSYS Mechanical thermal transient model to determine the heat flux generated by the combustion of the pyrotechnic device in the thermal diffuser. The team then used an ANSYS Fluent computational fluid dynamics (CFD) model to evaluate the volume of liquid that the thermal diffuser could heat. Simulation helped the engineers to quickly demonstrate the feasibility of the application and evaluate the relative performance of alternative design concepts. This method substantially reduced the time required to bring the product to market.

Fusehead of a small pyrotechnic device

UNUSUAL PYROTECHNIC APPLICATION

Typical applications for Davey Bickford include actuators and fire extinguishing systems for aircraft, electronic detonators for mining applications, detonators for seismic exploration, inflators, and explosives for tunneling. Davey Bickford's customers are becoming more and more demanding as they try to achieve higher performance and greater reliability at a lower cost.

This application is somewhat paradoxical because pyrotechnics is typically the last thing that an engineer would consider to generate a relatively low temperature for a long time period. However, pyrotechnics is being used more frequently for this type of application because it generates high temperatures and pressures with a very low electrical or mechanical energy input. The challenge for Davey Bickford engineers was to control the heat transfer from the combustion reaction to the liquid, evaluate several design alternatives within a short time, and confirm the feasibility of the application before making the considerable investment required for detailed design and manufacturing tooling.

Davey Bickford selected ANSYS solutions to design the heat exchanger because the simulation tools cover the full range of physics capabilities required to investigate advanced pyrotechnic devices. The pyrotechnic engineering team worked closely on this particular application with the Institut

Française de Mécanique Avancée (IFMA), which trains engineers in advanced mechanics and industrial engineering; the institute also works extensively with ANSYS solutions.

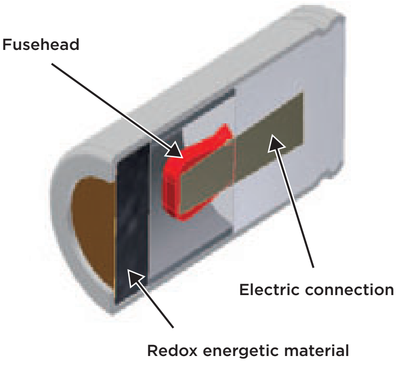
PICKING THE RIGHT MATERIAL

The first step in the project was selecting a pyrotechnic material. This application requires a kinetic combustion reaction that generates high temperatures and low pressures, so the team decided to use a redox material pair. A very small quantity of the selected pair when ignited generates high temperatures for a few milliseconds. The next step was designing the pyrotechnic chain. In this device, electrical initiation of a fusehead sets off combustion of the compressed redox powder. Over 1,000 C is generated with only 1 amp to 2 amps power input.

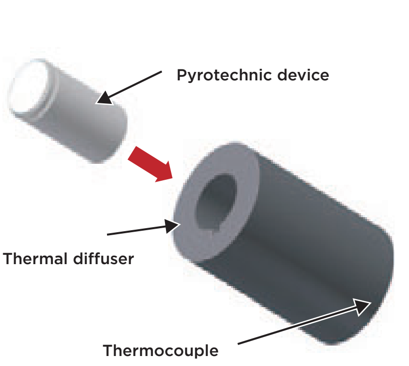
The huge amounts of energy and high temperatures released by the pyrotechnic device were controlled by enclosing it in a thermal diffuser that governs the heat transfer to the liquid volume. The team connected a K-type thermocouple to a multimeter linked to a Labview® station to measure the temperature increase in the surrounding medium as a function of time.

After determining the experimental temperatures for a simple initial concept design, engineers created a virtual prototype to explore the design space and optimize the design using the ANSYS Mechanical transient thermal solution. First, the team imported the initial design geometry created in Autodesk® Inventor® Pro 2009 into ANSYS DesignModeler and then simplified it slightly to remove all features that were not relevant for the virtual prototyping concept, ensuring accurate but fast simulation. They meshed the geometry and defined initial conditions. The initial temperature was set at ambient, and a heat transfer coefficient representing natural convection of 5 W/(m² C) was applied to the 3-D geometry. Engineers modeled the heat flux generated by the pyrotechnic device by mapping the transient heat flux in accordance with experimental results. Then they used the ANSYS thermal transient model to simulate the experiment. The simulation results correlated well with experimental results.

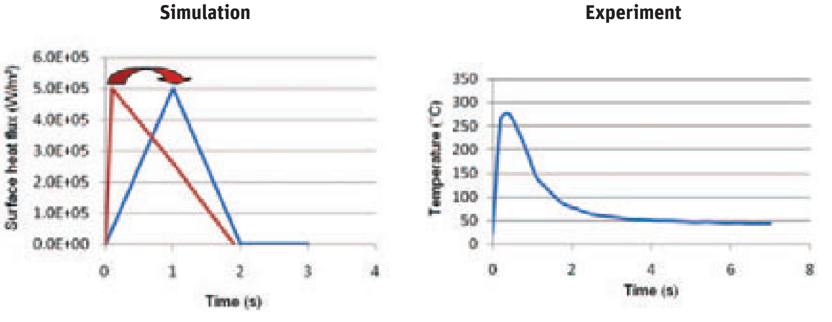
Engineers applied the transient thermal model to determine the temperature distribution around the external envelope



After receiving an electrical signal, the fusehead ignites an energetic redox material.



The pyrotechnic device is enclosed by a thermal diffuser.



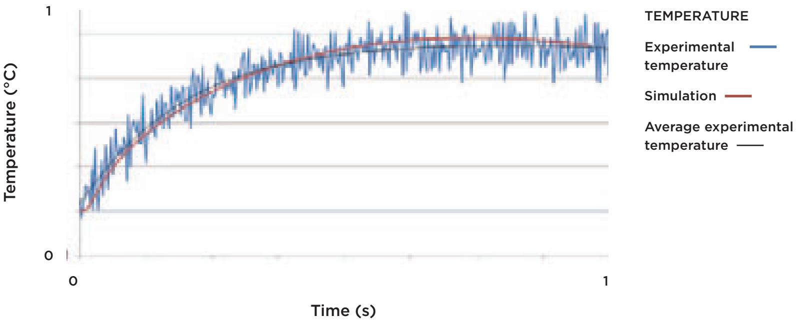
Engineers modeled the heat flux generated by the pyrotechnic device based on experimental results.

of the thermal diffuser. Simulation showed that the initial design of the diffuser produced high temperatures around the bottom of the diffuser and low temperatures toward the top. The customer required much greater uniformity in temperature distribution throughout the liquid, so the Davey Bickford team focused on the thermal diffuser's material properties in an effort to conserve the heat generated in the pyrotechnic reaction and to reduce the rate at which it was transferred to the volume of liquid. Engineers selected four materials with high thermal effusivity and low thermal diffusivity as candidates for use in the thermal diffuser. Thermal effusivity provides a measure of a material's ability to exchange thermal energy with material around it. Thermal diffusivity is the thermal conductivity divided by density and specific heat capacity at a constant pressure.

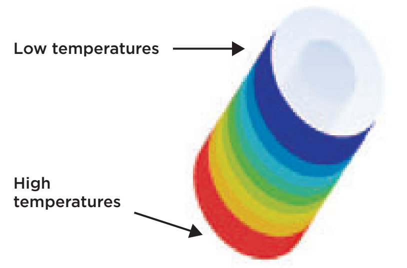
The team evaluated each material using the ANSYS transient thermal model. Materials 1 and 3 produce relatively large temperature variations, so they were quickly eliminated. Material 4 exhibited interesting thermal properties, but because it could not be easily machined, manufacturing costs would be too high. Engineers decided to focus on material 2, which provides a homogeneous temperature profile and is highly machinable.

OPTIMIZING THE DIFFUSER GEOMETRY

The next step was optimizing the geometry of the thermal diffuser. Davey Bickford engineers used design exploration capabilities, including the goal-driven optimization tool provided in ANSYS Workbench, to find the dimensions of the diffuser that would best achieve a uniform temperature distribution over the liquid volume. The first study used the thermal transient model to follow heat transfer through the solid–solid interface between the pyrotechnic device and the thermal diffuser. The optimization tool automatically



There was very good correlation between the numeric model and experimental results.



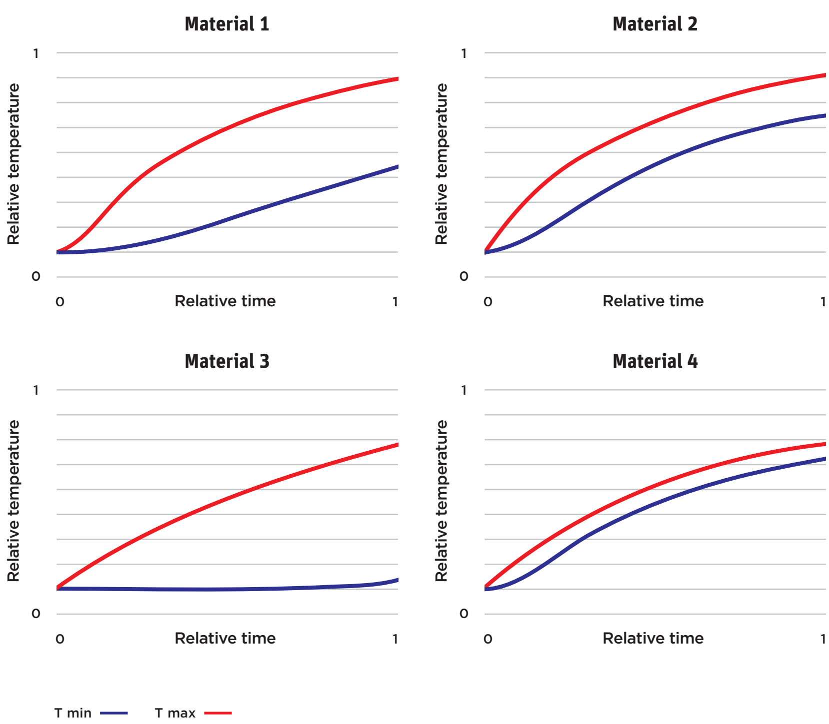
Temperature distribution around the external envelope of the initial thermal diffuser design concept

explored the design space and set the diffuser dimensions to best achieve the goal of a uniform temperature profile.

Engineers then used ANSYS Fluent to evaluate heat transfer between the solid–fluid interfaces and determine the volume of liquid that the thermal diffuser could heat. They modeled the container that held the liquid around the thermal diffuser. They exported results from the transient thermal analysis to the CFD software and set the initial conditions and thermal properties of the liquid. Simulation results showed that the optimized design provided a uniform temperature distribution in the liquid, meeting the customer's requirements.

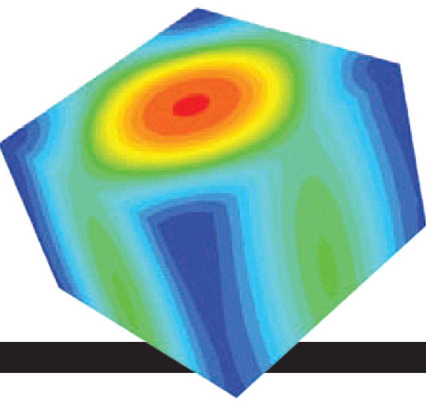
Simulation demonstrated the feasibility of controlling thermal transfer from combustion of an energetic material to a liquid by optimizing the material properties and geometry of the thermal diffuser. Of course, other constraints must be taken into account, and the team will further tune and tweak the geometry. ANSYS solutions allowed engineers to anticipate development risks and enabled them to quickly make a pragmatic decision about the feasibility of producing the product.

In addition, the success achieved through simulation in this project led to developing a new process to determine the heat flux of energetic materials. Transient thermal analysis was used to establish a database of the heat flux functions of energetic material formulations and configurations. This database enables the applications team to quickly propose solutions based on specific customer requirements.



Thermal diffuser maximum (red) and minimum (blue) temperatures as a function of time. The objective was to minimize the difference between the two to heat the largest possible volume of material. Materials 2 and 4 had good thermal properties. Material 2 was selected because of its superior machinability.

Temperature distribution in the liquid was predicted by CFD.



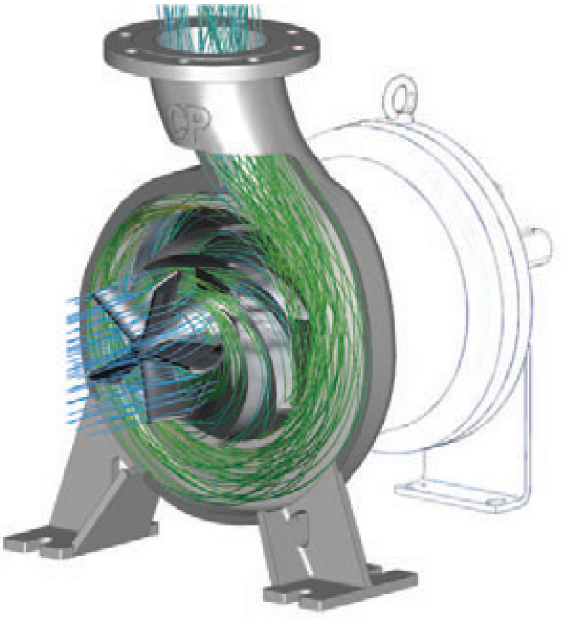
PUMPING UP SUSTAINABILITY

Flow simulation optimizes the energy requirement of pumps.

By Thomas Folsche, Head of Technology, CP Pumpen AG, Zofingen, Switzerland

Process plant operators increasingly strive to reduce power consumption, achieve high efficiency and lower maintenance costs when investing in pumps. To gain an edge in this competitive marketplace, CP Pumpen (CP Pumps) in Switzerland, one of the industry's leading suppliers of premium quality centrifugal pumps, invested in engineering simulation software to improve its products.

The company has been helping its customers achieve sustainable improvements in fluid handling systems for many years. The power costs for operating a pump may comprise as much as 85 percent of its overall lifecycle cost. This provides the potential for huge savings by improving hydraulic performance and increasing overall efficiency, for example, thereby reducing power consumption.



ANSYS CFX simulation provides information on the discharge head, power consumption and efficiency; it also provides the designer with a view of the flow field inside the pump.

RAPID AND COST-EFFECTIVE DEVELOPMENT

Several years ago, CP Pumps needed to modify a legacy product family: a metallic chemical magnetic coupling pump (MKP). After initial attempts with in-house development tools, the team determined that the standard method of product development was far too time consuming and costly. The initial designs could be compared only based on experimental data, requiring prototypes to be individually produced and subsequently tested on the hydraulic test bench. In the search for alternatives, the company learned about ANSYS CFX and ANSYS BladeModeler software. BladeModeler allows CP Pumps engineers to quickly and easily model impeller geometries. This tool enables the user to design both the meridional flow path and the blade shape, including the blade-thickness distribution. Once blade geometry is specified, the software determines the cross-sectional area along the flow path to enable assessment of flow characteristics.

The engineers then can launch a CFX computational fluid dynamics (CFD) simulation from within the ANSYS Workbench platform. CFX calculates the complete three-dimensional flow field, including fluid pressure and velocity, which enables assessment of the impeller discharge head and efficiency. CP Pumps optimized multiple impeller designs for a particular pump size and subsequently tested them on the in-house pump test bench. The engineers developed an experimental testing method that allowed these prototypes to be used with the necessary metal connecting components (such as bearings or magnetic drive) and then tested them in existing spiral housings. Using simulation, the company rapidly and cost effectively created multiple designs, compared them with each other and validated them.

WORTHWHILE INVESTMENT

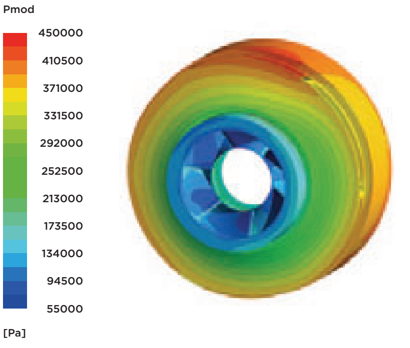
Many of the impellers designed using CFD dramatically improved the pump's hydraulic characteristics, as verified by the pump test rig. Other designs that initially looked promising proved to be inadequate following CFD assessment, so prototype building (and accompanying costs) were avoided. Originally, the impeller was the focus of the new design. However, the development team quickly realized that the interaction between the spiral housing and the impeller was far more important than had been originally thought. It was not sufficient to simulate one part in isolation; instead, the team needed to include the whole unit in the simulation. Based on market research and this positive experience with ANSYS software, CP Pumps decided on systematic use of the simulation software for pump design.

Initially, the team calculated simulations on a steady-state basis since, at that time, the company did not have sufficient resources, including computing power to perform transient analysis. Transient calculations provide more detailed results but are more computationally intensive. To gain the benefits of a transient simulation, the company invested in a high-performance computer (HPC) cluster with multiple processors and a very large working memory along with multiple software licenses. The engineering group at first had to persuade management that this investment was worthwhile; when the first successful pump development left the test bench just a few months later, management was convinced. CP Pumps' hardware, software and personnel outlay for simulation with HPC is considerable. However, if the company develops only four pumps per year with CFD, the investment pays for itself because the pattern doesn't need to be altered for every pump size.

OPTIMIZED IN RECORD TIME

A further advantage of transient calculation is that, in addition to receiving hydraulic data (discharge head, power consumption and efficiency), the user also obtains information with regard to the transient pressure distribution inside the pump. With simulation, the company improved hydraulic performance in all 18 pump sizes of its MKP model; it also minimized the mechanical load on the impeller bearings, leading to improvements in the service life of the pumps. A further advantage of the hydraulic optimization is noise reduction for the pumps. Without CFD, it would have taken the company at least three times as long to develop equivalent products.

Completed pump



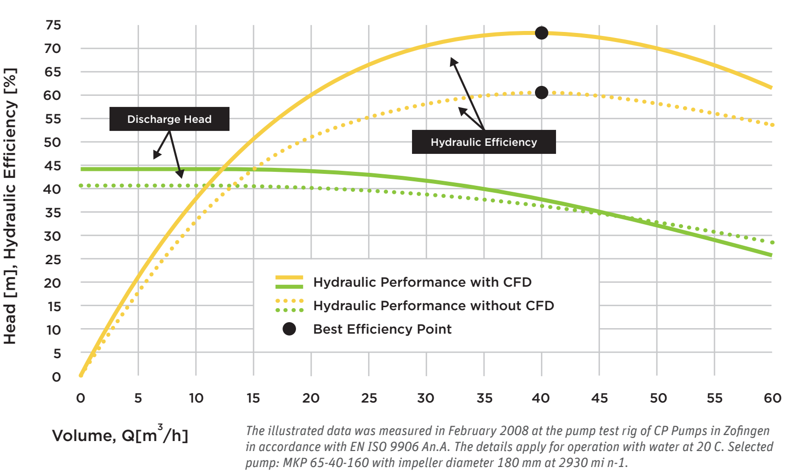
Pressure distribution around impeller

Without CFD, it would have taken the company at least three times as long to develop equivalent products.

IMMEDIATE PAYBACK

Before modification, the pumps' efficiencies were acceptable, but today they are outstanding. In the final analysis, the use of fluid flow simulation has led to efficiency improvements of up to 50 percent. For customers, the pump consumes less power and requires less maintenance. "Last year, we had to replace two large heat exchanger oil pumps. The energy savings was so great that the investment paid for itself within a year," said Urs Wursch, chairman of CP Pumps. As the hydraulic efficiency of a pump increases, energy consumption declines correspondingly. This enables CP Pumps' customers to meet sustainability goals. The hydraulic components optimized with ANSYS CFX ensure high efficiencies and, in combination with special eddy-free can units, power savings of up to thousands of euros per year are possible.

Hydraulic performance improvement gained using CFD for MKP pump 65–40–160



The illustrated data was measured in February 2008 at the pump test rig of CP Pumps in Zofingen in accordance with EN ISO 9906 An.A. The details apply for operation with water at 20 C. Selected pump: MKP 65-40-160 with impeller diameter 180 mm at 2930 mi n-1.

ENGINEERING SIMULATION TAKES FLIGHT

Parker Aerospace uses ANSYS technology to reduce time and costs — as well as risk — in the design of aircraft systems.

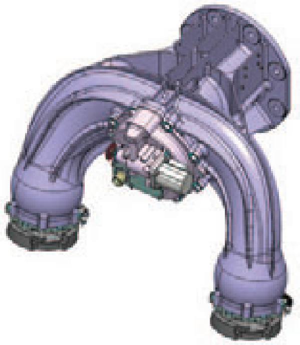
By Carsten Mehring, CFD Technology Lead, Parker Aerospace, Irvine, U.S.A.

As aircraft become more efficient, reliable and safe, they are also becoming more complex. Integrated systems design, architecting and systems control now take the spotlight in any new aircraft development project. Complex systems modeling requires accurate knowledge of subsystem and component performance. Although engineers can obtain component performance predictions using experiment, these test programs can be prohibitively expensive as the operating envelope continues to expand. The range of this operating envelope, characterized by the range of operating conditions (which include inlet pressures, flow rates, altitude and many others), continues to increase as companies extend the use of each aircraft component.

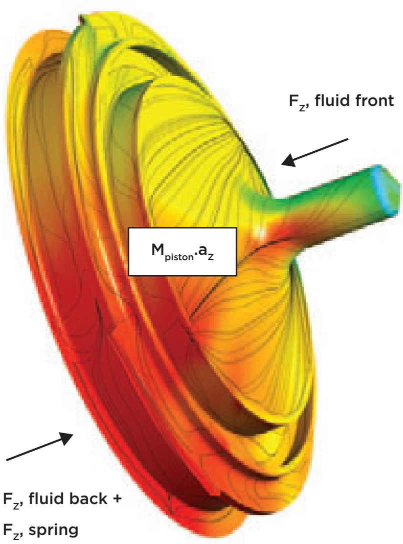
To analyze and evaluate design of components within a system and to support component test programs, the Central Engineering Group at Parker Aerospace uses ANSYS CFX for CFD and ANSYS Icepak for thermal packaging analyses. This helps the company to decrease development time and costs as well as to reduce risk.

Parker Aerospace is a global leader in flight control, hydraulic, fuel, fluid conveyance, thermal management, and engine systems and components used on virtually every commercial and military aircraft and aero-engine in production in the world today. The company's products are found on commercial transports, military fixed-wing planes, general aviation and business aircraft, helicopters, missiles, and unmanned aerial vehicles, as well as in other high-tech applications.

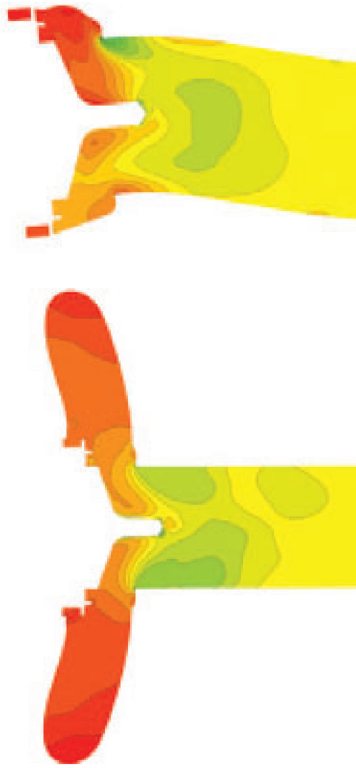
Simulation of Parker refuel coupling predicts pressure distribution at steady-state conditions and main piston dynamics based on transient FSI.



CAD model of refuel coupling



Streamlines and contour lines of static pressure on main piston



Contour lines of static pressure in two cross-sectional planes

Parker Aerospace uses ANSYS software to analyze and evaluate the design of components within a system, thereby reducing development time and cost as well as risk.

The Central Engineering Group at Parker Aerospace provides the company with broad expertise in a variety of engineering disciplines. The mechanical design team is responsible for delivering superior quality, cost- and time-effective analyses for aerospace systems and components. This includes providing premier analytical expertise by leveraging diverse problem-solving capabilities, including CFD and thermal packaging analysis using ANSYS tools.

AIRCRAFT/ENGINE FUELING

Parker's refuel coupling, used as a connection during refueling, might look like a part of a jet pack or Iron Man's suit, but, in reality, it controls the fueling process for one of the latest aircraft under development. This coupling, comprising two independent refuel lines that merge into one main port controlled by its own piston, is crucially important to meet aircraft safety as well as fuel fill and defuel time requirements.

Employing a mesh of approximately 12 million elements for numerous steady-state configurations (including piston settings), engineers used CFX to predict wall pressure distribution within the system to optimize the locations of pressure monitoring points. The system uses these points to control the closing and opening process of the piston in the main feed line. Experimentally measured overall pressure drop across the system agreed with the CFD results (within the limits of uncertainty associated with test data accuracy). Parker's CFD analysts are also deploying fluid–structure interaction (FSI) capabilities to tailor the dynamic behavior of the refuel coupling to specific customer requirements.

While full-blown FSI has its place within Parker's suite of modeling approaches, it does require considerable computational resources. To obtain quick turnaround and to conduct parametric design studies, the team often uses a combination of steady-state CFD analysis and 1-D simulation. If the system characteristic time scales allow, engineers take advantage of Parker's unique expertise in dynamic system modeling using SimuLink®, in which fluid dynamic

Analysts have successfully employed fluid dynamics to predict the performance characteristics of ejector pumps, especially their performance limits.

effects are incorporated into the 1-D simulation via lookup tables generated by steady-state CFD analyses.

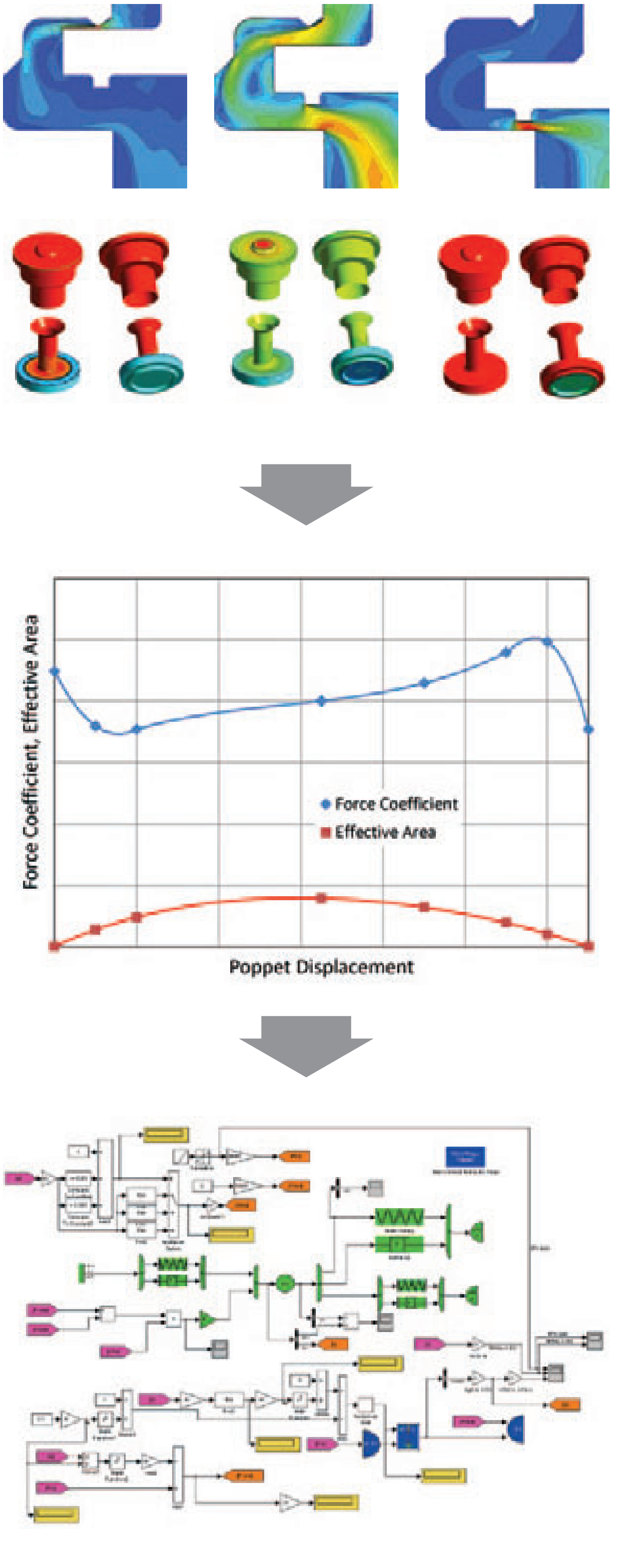
FUEL PUMPING

Aircraft fuel tanks are often housed in the wings within multiple compartments; the tanks contain baffles to prevent sloshing. Accessing this fuel requires the use of specialized pumps to feed the main fuel pump(s). Scavenging pumps, usually ejector pumps, are commonly used to collect fuel from remote corners or the bottom of fuel tanks and to discharge that fuel at the inlet to the main fuel feed pump(s). Fuel scavenged from the bottom of fuel tanks is often contaminated with considerable amounts of water, which was originally dissolved within the fuel or entered the fuel tank through condensation. Another important function of ejector pumps is to disperse that water into fine drops so that the engine can safely consume the fuel, with these finely dispersed water droplets, without any impact on performance.

Traditional ejector pumps do not have any moving parts and are highly reliable if used within their operating range. However, the operating range of fuel ejector pumps is limited by the onset of cavitation. Cavitation limits the pump's operating range, so the ability to identify this range early in the design process is important, even before building development hardware. Analysts at Parker Aerospace have successfully employed computational fluid dynamics to predict performance characteristics of ejector pumps — especially their performance limits — by utilizing the Rayleigh–Plesset cavitation model within CFX. The team is currently working to improve the existing cavitation model via customized scripts. Areas of focus include considering shear-induced cavitation effects and cavitation in multifluid ejector systems.

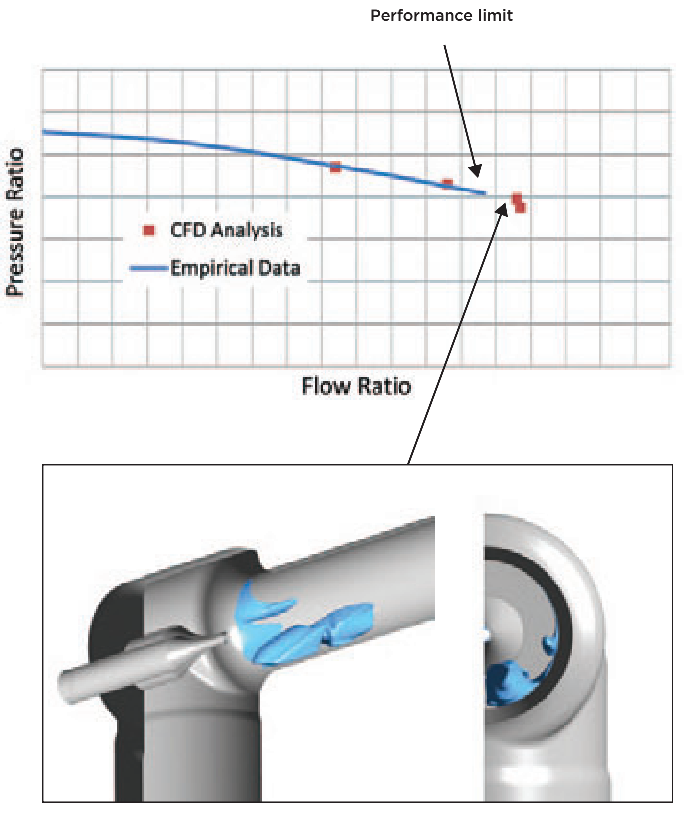
FUEL-TANK INERTING

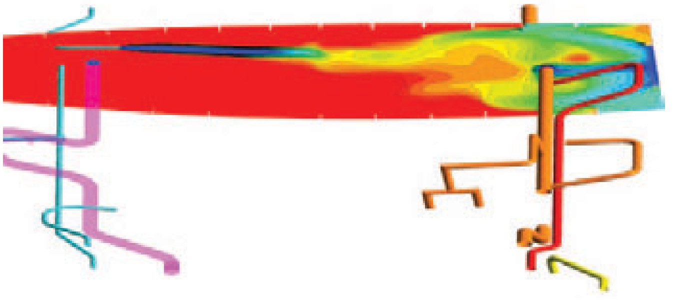
To prevent fuel tank explosion, fuel-tank inerting systems are deployed on commercial (and military) aircraft. One specific type of inerting system delivers nitrogen-rich (oxygen-deprived) air into the ullage (non–fuel-filled) space of the aircraft wing and center fuel tanks to reduce the likelihood of flammable fuel/air mixture formation, thus reducing the chance for ignition and fuel tank explosion. To demonstrate proper operation of the inerting system over the entire aircraft operating envelope, Parker engineers employ a comprehensive lumped parameter model of the tanks, vent lines and air flow within them in



Engineers at Parker combine steady-state CFD analyses and their unique lower-dimension simulation capabilities to analyze system dynamics of hydraulic valves. The example shows a fuel reprime valve. CFD analysis (top) generates lookup tables for force coefficient and effective area as a function of poppet position (center). These tables are later used within a 1-D simulation tool to predict overall system dynamics.

Comparison between jet pump performance prediction using ANSYS CFX and experimental data including onset of cavitation. Upon inception of cavitation, the cavitation region quickly expands as pressure ratio decreases. Iso-surface of vapor-phase volume fraction (=0.5) at the onset of cavitation is shown in blue.





Result from wing tank ullage space CFD analysis showing instantaneous oxygen concentration contour lines in a "spar" plane through the center of a nozzle injecting nitrogen-enriched air into ullage space for inerting purposes.

conjunction with a Monte Carlo method. This method generates statistical flammability exposure times (the time during which a spark would ignite a flame) for a range of unknown operational parameters that are bounded by known distribution functions and for a large number of theoretical flights (taking into account, for example, circumstances such as plane climb rate and weather conditions).

To validate and improve the prescribed lumped parameter model, Parker analysts conduct detailed CFD analyses for a number of characteristic flight sequences. The detailed information obtained from the CFD analyses is then utilized for benchmarking the lower-dimension lumped parameter model. Despite the large amount of computational resources required to conduct the CFD analyses, the cost of those resources is still minute compared to the expenses of a full-scale ground or flight test. While flight data is available at some stages of the development process, early CFD analysis provides an opportunity to deploy design improvements that cannot be determined by the lumped parameter model: for example, location and direction of nitrogen-enriched air (NEA) jets and location of air vents. In addition, detailed CFD analyses of the mixing within the ullage space and mass transfer between the various fuel tank compartments provides the opportunity to improve upon the lumped parameter model and, consequently, to improve predictions of flammability exposure times, all leading to a safer aircraft design.

CONTROL ELECTRONICS

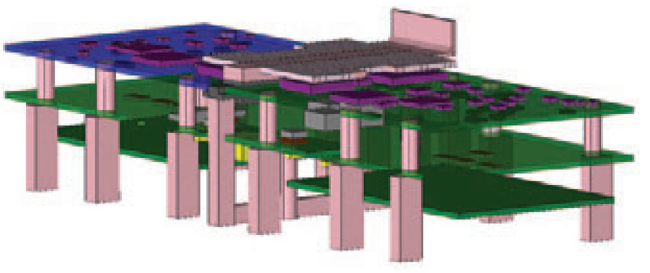
With the development of the more electric aircraft, the number of electronic control units on board increases, while the space available for housing those components does not (a result of efforts to maximize cargo or passenger space). Additionally, increasing usage of composite materials in aircraft design reduces the opportunity to utilize the airframe surrounding electronics boxes as heat sinks. At the same time, energy spent to cool electronics components should be as low as possible to reduce the impact on overall system efficiencies. Because of these factors, electric components run hotter than ever before, and accurate predictive tools are needed to guarantee their proper function even before in-flight testing commences. At Parker, analysts extensively use ANSYS Icepak to fulfill those needs. Using this tool, engineers are able to gather information on temperature levels early in the design process to assure that electronic components are operable both under cold-start conditions (meeting warm-up requirements) and in continuous operation under hot ambient conditions. The analyses conducted at Parker span a broad spectrum, varying in the level of fidelity by which the components are modeled (ranging from very rough models with components lumped together to more detailed approaches verifying that maximum component junction temperatures are not exceeded). Analyses also vary in the type of heat transfer used to dissipate the majority of energy away from the electronics components (forced convection via fans and cooling channels or conduction-dominated heat transfer through PCB boards and casings that are enhanced by board vias and board/casing conductive bracings, for example).

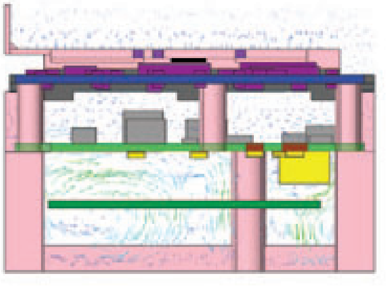
KNOWLEDGE COLLECTION

A wealth of other Parker products have benefited from analyses using ANSYS tools, including fuel system solenoid banks, liquid ring pumps, pneumatic butterfly valves, centrifugal fuel and gear pumps, and electronics cooling plates. Similar to the path FEA analysis took a couple of decades ago, CFD has matured to become an integral part of the product design process at Parker Aerospace, reducing development time and cost while achieving optimum product performance. The wealth of information obtained from CFD analysis provides detailed insight into systems behavior and facilitates generating knowledge and correlation databases, thereby reducing development risk, cost and time.

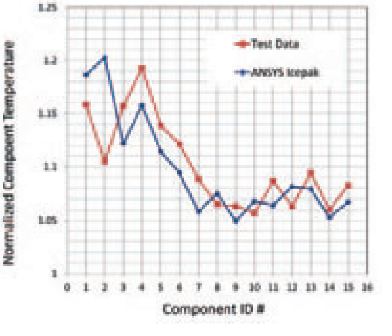
ANSYS Icepak predicts buoyancy-driven flow and PCB board surface temperature distribution within an electronics box at specific ambient and operating conditions. Comparison with test data identified faulty thermocouple measurement on a component.

Multiboard electronics control unit

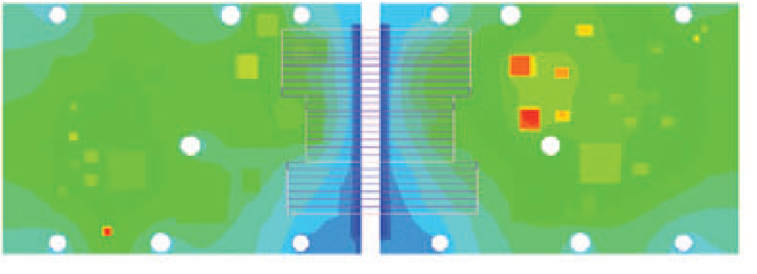




Buoyancy-driven flow between PCB boards



Model benchmark using available test data



Board temperature contour lines on one side of PCB board

The wealth of information obtained from CFD analysis provides detailed insight into systems behavior and facilitates generating knowledge and correlation databases.

WEARING A WIRE

Simulation helps to optimize body-worn wireless devices for an emerging class of applications.

By Bert Buxton, Senior Electrical Engineer
Synapse Product Development, Seattle, U.S.A.

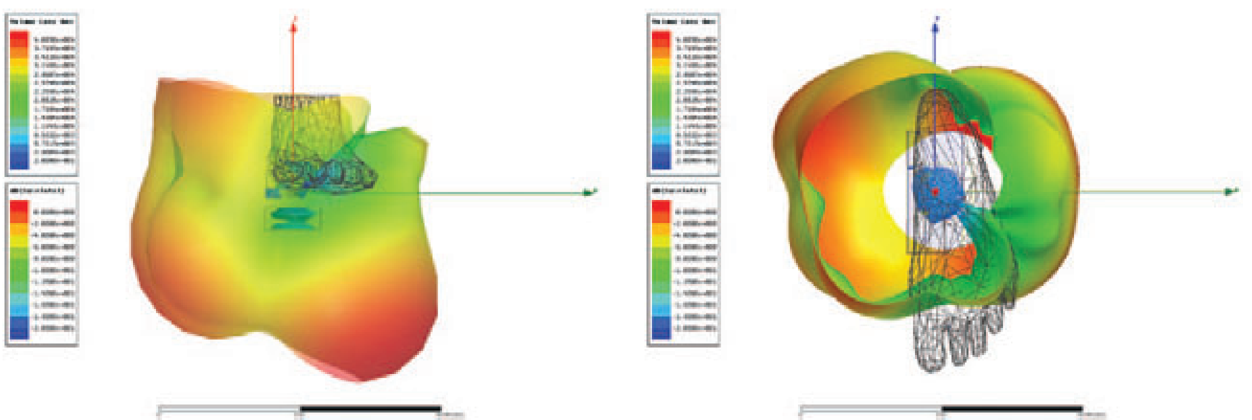
Interest in body-worn wireless devices has grown in recent years because of actual and potential applications in healthcare, sports, law enforcement, entertainment and other areas. For example, the U.S. Department of Defense is working on a wireless device to be worn by soldiers that will allow medics to measure vital signs and collect other medical information from the troops. Body-worn wireless devices have been developed to measure and record an athlete's performance, such as running speed and the number of strides.

Regardless of the application, using a wireless device in close proximity to the human body creates a number of major design challenges. The radiated power of the device must be kept below levels that can create a health hazard. The device's power consumption, size, aspect ratio and weight must be minimized to make it suitable for wearing. Yet the device must be designed to deliver a signal of sufficient power to the right location, with good reception by the target device — despite the fact that the human body may absorb a significant portion of the signal.

MODELING THE SYSTEM

Synapse Product Development solves such difficult engineering challenges from concept through manufacturing for leading consumer electronics and life-sciences companies. One of the company's specialties is developing body-worn wireless devices for a wide range of applications. The design of the antenna is often a major challenge in these devices because the body absorbs so much energy. Synapse uses the ANSYS HFSS 3-D full-wave electromagnetic (EM) simulator and the ANSYS human body model

Synapse uses ANSYS HFSS and the ANSYS human body model to evaluate performance of various antenna designs by modeling the complete system, including the wireless device and antenna and their interactions with the human body.

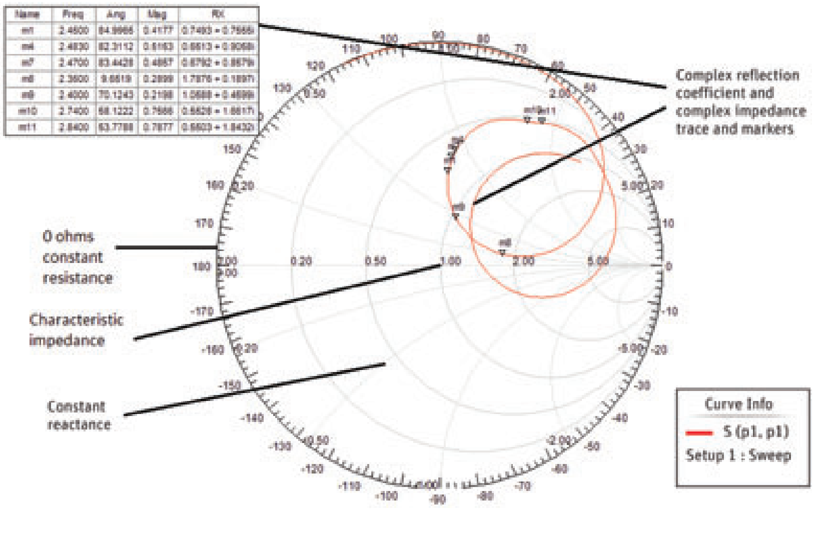


ANSYS HFSS simulation output shows power absorbed by foot and ground.

to evaluate the performance of various antenna designs by modeling the complete system, including the wireless device and antenna and their interactions with the human body. The ability to evaluate designs without building physical prototypes typically helps Synapse engineers to increase the performance of the antenna by a factor of five compared to the original design concept.

Antenna design focuses on transferring power from transmitter to receiver. A dipole antenna is a well-established reference for performance and has the perfect geometry to optimize the power transfer of the antenna. For a 2.45 GHz antenna built with an FR4 printed circuit board, the wavelength is 60 mm, so the total length of the dipole should be 30 mm. This is too long for most body-worn wireless devices. So instead, electrical engineers design a smaller antenna with properties as similar to a dipole as possible. For example, they attempt to match the antenna's radiation resistance to the optimal load impedance of the transceiver. Radiation resistance is that part of an antenna's feedpoint resistance that is caused by the radiation of electromagnetic waves from the antenna.

The complexity of the antenna geometry required for body-worn wireless devices makes it very difficult to create an acceptable design in a reasonable amount of time using the traditional build-and-test design process. Facing this and many other difficult design challenges, Synapse engineers evaluated a number of different simulation products. ANSYS provides a solution for nearly all of their design challenges, including circuit, electromagnetic, mechanical and thermal simulation. ANSYS software enables automatic data transfer to simultaneously optimize the product over multiple disciplines and domains. Synapse's management staff concluded that purchasing all of its simulation tools from a single vendor would deliver great benefit, such as a single-support contact for questions and training.



Smith chart helps engineers to match impedance of antenna and transmitter.

THE DESIGN PROCESS

The design process typically begins with the industrial designer providing a concept that incorporates the electronics and antenna. Synapse electrical engineers then use ANSYS HFSS to optimize the wireless antenna design. The engineer starts the modeling process by importing the geometry of the initial design from a SAT file. The next step is defining the electrical properties of the materials, such as permittivity and dielectric loss tangent, permeability and magnetic loss tangent, bulk electrical conductivity, and magnetic saturation.

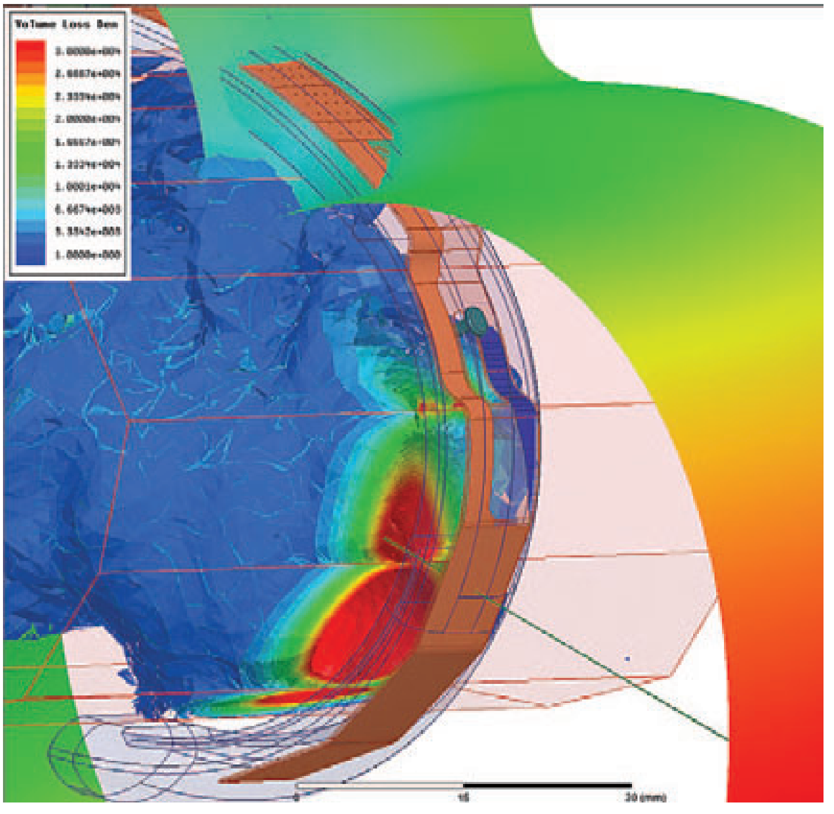
Optimizing the performance of the antenna requires close attention to the way in which the human body affects antenna performance — thus the need for a systems approach to analysis. The ANSYS software's human body model enables users to set the dielectric constant for different parts of the body. Typically, Synapse engineers vary skin thickness from 0.4 mm to 2.6 mm and assign it a dielectric constant of 38. The thickness of the fat layer is chosen to account for all impedance-matched effects, typically half of the wavelength, with a dielectric constant of 5.3. The muscle serves as a termination to the model with a thickness of approximately 20 mm and a dielectric constant of 53.

HFSS automatically specifies the field behavior on object interfaces and defines a geometrically conforming tetrahedral mesh. Adaptive meshing refines the mesh automatically in regions in which field accuracy needs to be improved. The software computes the full electromagnetic field pattern inside the solution domain. The next step is computing the generalized S-matrix from fields calculated in the solution volume. The resulting S-matrix allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals, reducing the full 3-D electromagnetic behavior of a structure to a set of high-frequency circuit parameters.

The HFSS simulation shows the power absorbed by the body and the gain of the antenna in the form of a color map incorporating both the body and surrounding airspace. In the typical case, simulation results show that the areas of the body closer to the antenna absorb more power. In the case of a device worn in a shoe, for example, the results will identify the amount of power absorbed by the ground as well, which sometimes turns out to be even larger than the energy absorbed by the foot. Based on these results, electrical engineers provide feedback to the industrial designers and system engineers, including information about the geometry of the antenna as well as how close and where on the body it can be positioned.

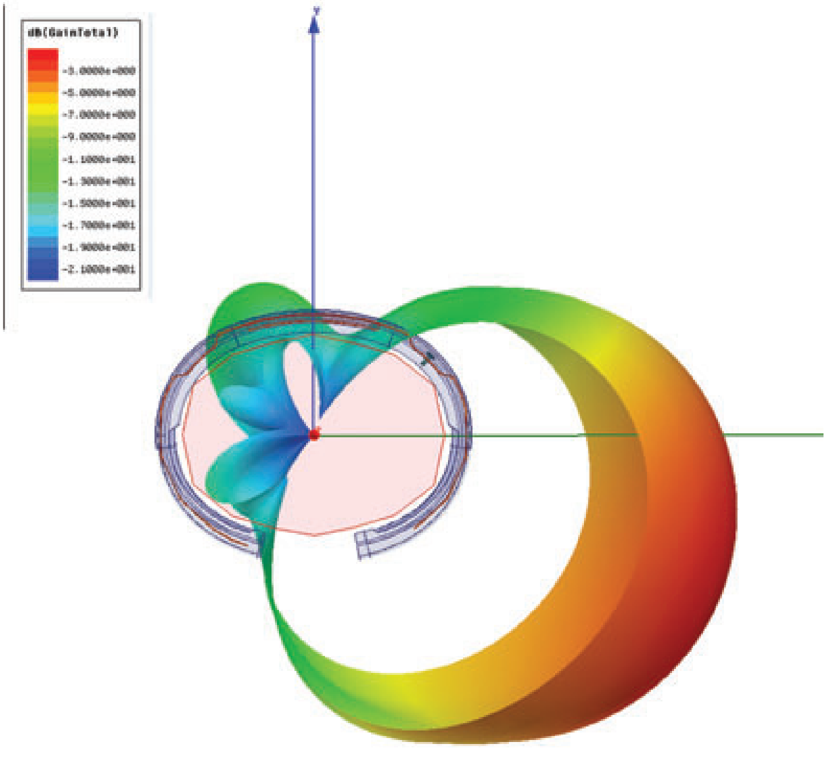
INCREASING RANGE WHILE SAVING TIME

The antenna performance information provided by simulation plays an important role in the system design of a body-worn wireless product. The antenna gain results are critical in link analysis, which



Power absorption of a product worn specifically on the wrist

Guided by simulation, electrical engineers typically can increase the range of the product by a factor of five while saving an estimated three months of development time.



3-D gain of product worn on the wrist

determines the range and throughput. The antenna gain also helps to determine how much transmit power is required, which, in turn, impacts battery life. In the typical case in which more than one device is worn on the body, the antennas of all devices are optimized simultaneously to align the gain between them and to minimize battery power consumption.

In addition, simulation is used to make the antenna smaller to meet industrial design and mechanical design objectives while achieving the required level of performance. As the size of the antenna is reduced, it works over a narrower bandwidth of frequencies. Simulation predicts not only in-band performance but also out-of-band performance, and it helps to avoid radiating at frequencies that would interfere with other devices. Guided by simulation, electrical engineers typically can increase the range of the product by a factor of five, relative to the initial concept, while saving an estimated three months out of a traditional 12-month development cycle.

TO LIVE FOREVER

Fatigue simulation can increase product life, reduce costs and decrease risk.

By Bence Gerber, Lead Product Manager, ANSYS, Inc.

Fatigue failure is insidious. It might be happening to your product right now, but you don't know it yet. Cyclic loadings cause changes to the surface of a part that can be seen only at a microscopic level. After long-term repetitive compression and tension, a visible crack appears, and then the part quickly fails.

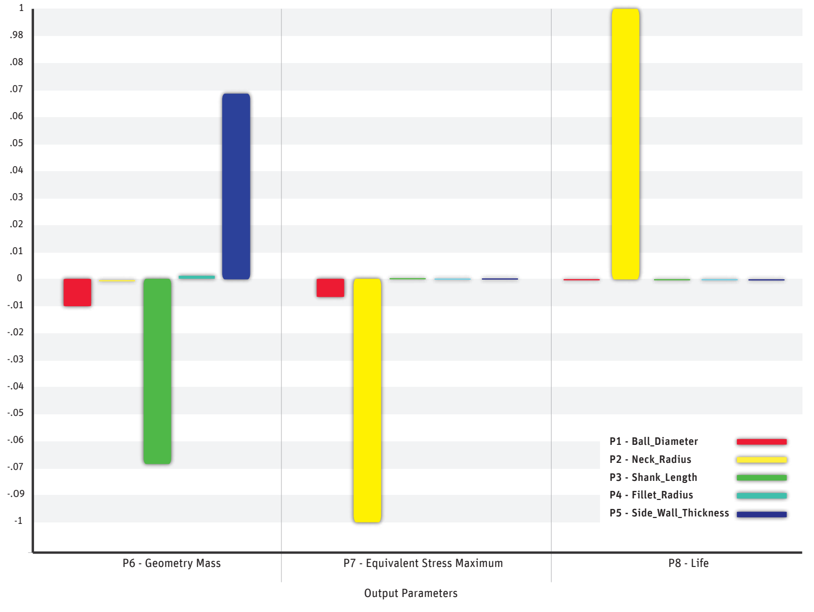
Fatigue is the failure of a part subjected to repeated loading and unloading at stress values lower than the part material's yield strength. Simple tension tests identify only failure caused by loadings that exceed the yield strength; without physical testing, such as a shaker test, the result is false confidence in a part's durability. However, physical testing is time consuming, expensive, and often impractical or impossible for large and/or complex products. Whether or not a product will fail is a function of the shape of the product, quality of the surface, materials used, and severity of the loading history.

Failure from repetitive stress-based phenomena has been recognized since the industrial revolution, but the term "fatigue" was not coined until the mid-19th century. During most of the 20th century, engineers performed fatigue calculations by hand or with a spreadsheet, making it difficult to gain a complete understanding of how structures fail from fatigue. Results of tests for large numbers of loading cycles (10^4 to 10^8) are often characterized by a semi-logarithmic graph with stress on the y-axis and life in cycles based on a logarithmic scale along the x-axis, referred to as an S–N curve or Wöhler graph. These charts are the basis for material models used to determine when material failure will occur.

Now, by combining two computational methods, design teams can calculate the time to failure for predetermined loadings with reasonable accuracy. Using finite element methods, a product's response to loadings can be simulated, and the resultant maximum stresses and strains determined. These results can be combined with one or more loading histories, which can be predicted using mathematical functions to represent the loading and unloading or based on test data that was gleaned, for example, from a test car equipped with a number of strain gauges and accelerometers. Using material data (S–N curves) collected from experiments and stored as part of a material library, it is easy to determine the number of loading cycles at which point the part fails from fatigue. This represents product life.

A unique and powerful fatigue process is enabled by combining the best-in-class structural simulation tool, ANSYS Mechanical, with ANSYS nCode DesignLife, a leading durability software that works within the integrated ANSYS Workbench environment. Workbench uses parameters to evaluate design options as well as design exploration for

Now, design teams can calculate the time to failure for predetermined loadings with reasonable accuracy.



Fatigue life optimization based on neck radius for a trailer hitch

full optimization. This fatigue process automates the steps in evaluating product life; it reduces errors when compared with the manual calculation method. Within the Workbench project, an expert can define a workflow process that can be used repeatedly by designers who are not fully trained fatigue specialists. Because the process can be stored, it can be reused long after the initial design if a redesign is necessary.

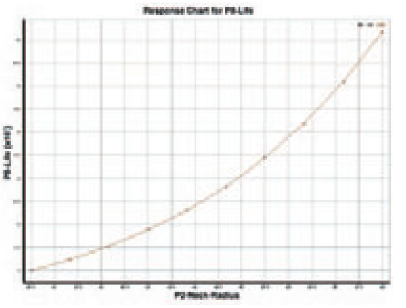
Simulation of fatigue allows product designers to make informed decisions in evaluating trade-offs, such as using different geometries and materials, to achieve desired product life. The traditional way to eliminate product

Bicycle crank fatigue failure from cyclic loading

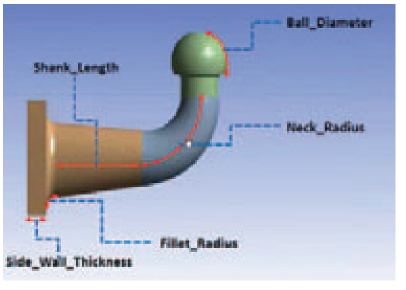
failure is through the use of conservative engineering. In today's economy, with growing pressures to reduce manufacturing costs, improve performance and efficiency, and decrease risks associated with product failure, optimization through simulation provides a competitive advantage; it may soon become a necessary tool for survival.

Simulation not only assists with the evaluation of numerous design options, it provides insight into the physical phenomena taking place inside a product during its lifecycle. When attempting to understand fatigue failure, testing provides only a yes-or-no result. If the product does not fail, testing does not provide any indication of how close it is to failure and the potential for some minor additional load to cause failure. If in fact the product does fail during testing, it is impossible to determine if failure would have occurred with significantly less loading or with a shorter loading cycle at higher loads.

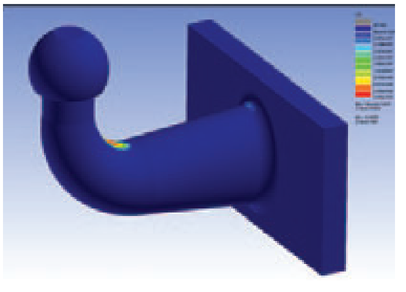
The aerospace industry was among the first to adopt fatigue simulation,



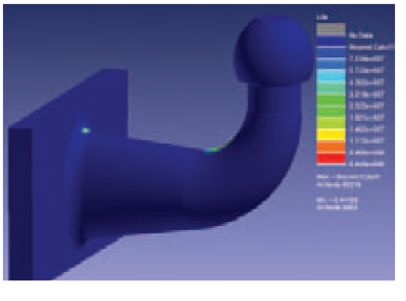
Sensitivity of life to optimization parameters for trailer hitch



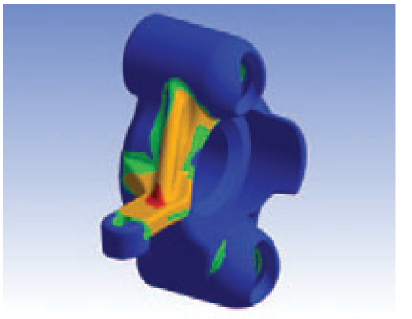
Design parameters used for optimization



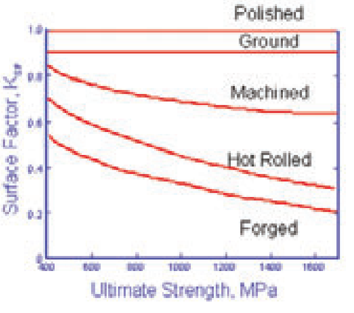
Life contour of trailer hitch identifying shortest life spot



Optimized fatigue life of trailer hitch



Safety factor contour for a given fatigue life of gear



Influence of surface quality on fatigue properties

because aircraft undergo high cycle loadings from vibrations and pressurization of the fuselage. Jet engines and landing gear also are exposed to repeated loads. A key factor in this industry is weight reduction, as designs that are too conservative (and, therefore, too heavy) are more costly to operate. In the automotive industry, manufacturers are pressured to increase fuel efficiency while decreasing weight. In the energy industry, environmental concerns are fueling the installation of wind turbines, which undergo repeated cyclic loading while rotating. Simulating fatigue can assist a wide range of organizations in addressing these concerns.

Failure from fatigue can have dire consequences for most companies. Monetary costs can be huge, including legal liabilities; maintenance, redesign and warranty expenses; and a damaged brand. Thanks to modern simulation technologies, product developers can greatly reduce the probability of failure while enabling the creation of more competitive, more profitable and more sustainable products. ∧

INTRODUCING APDL MATH COMMANDS

A new set of commands in ANSYS Mechanical APDL provides access to powerful matrix manipulation features.

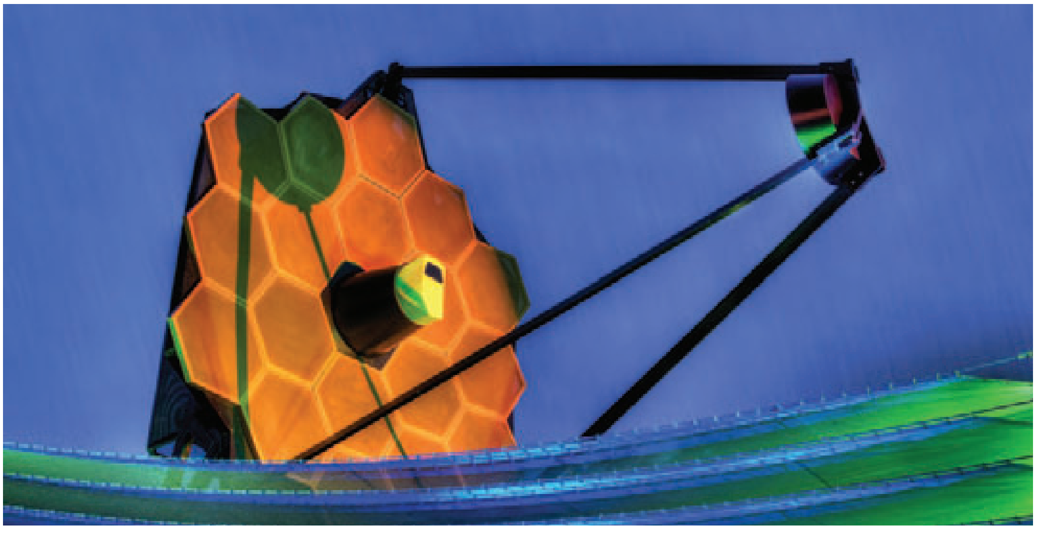
By Lester M. Cohen, Chief Engineer, Structural Analysis & Design Group; **Michael Eisenhower**, Structural Engineer; and **Vladimir Kradinov**, Structural Engineer, Harvard-Smithsonian Center for Astrophysics, Smithsonian Astrophysical Observatory, Cambridge, U.S.A.

Fady Mamoud, Software Developer–Simulation Expert, and **Claire Le Baron**, Numerical Modeling Research Engineer, Michelin, Clermont-Ferrand, France

Dave Conover, Chief Technologist, and **Frederic Thevenon**, Lead Software Developer, ANSYS, Inc.

The ANSYS Parametric Design Language (APDL) allows you to script and control the different steps of a solution process, from pre-processing to post-processing. A new set of commands called APDL Math provides the ability to access and manipulate the large sparse matrices involved in the solve step itself. Accessing this data delivers more possibilities than ever to customize the analysis process. This command set is based on tools for manipulating large mathematical matrices and vectors that provide access to standard linear algebra operations, access to the powerful sparse linear solvers of ANSYS Mechanical APDL (MAPDL), and the ability to solve eigenproblems.

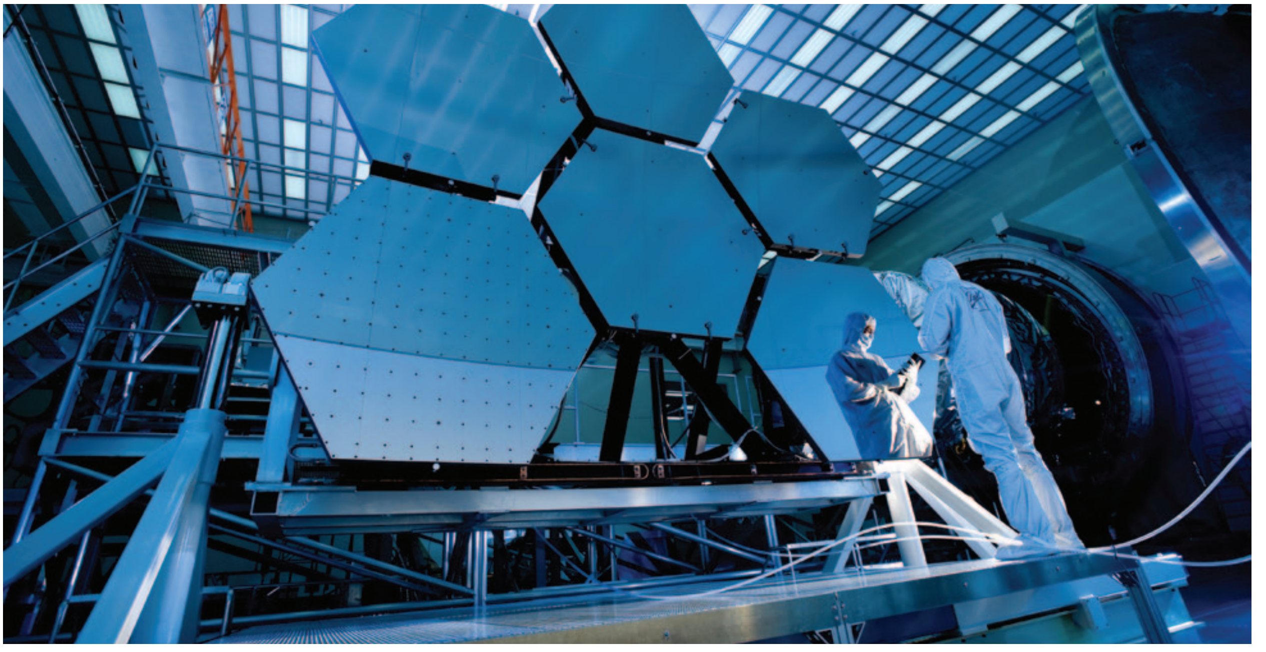
APDL Math allows you to access matrices and vectors on .FULL, .EMAT, .MODE and .SUB files, as well as other sources, so you can read them in, manipulate them, and write them back out or solve them directly. This functionality augments vector and matrix operations (such as *VOPER and *MOPER) in the standard APDL scripting environment. Both dense matrices and sparse matrices can be manipulated using APDL Math. Prior to the implementation of APDL Math, you needed to use FORTRAN code to obtain access to the data in these files.



The James Webb space telescope

PHOTO: BOBBY BRADLEY

PHOTO: NASA/MSFC/DAVID HIGGINBOTHAM/EMMETT GIVEN



Six of the 18 mirror segments ready for optical test in a cryogenic environment at NASA Marshall Space Flight Center

On average, the ANSYS sensitivity capability has decreased by approximately six times the amount of time to perform work on the telescope.

PROGRAMMING ADVANCED ALGORITHMS

APDL Math commands open a new field of application: the ability to program complex algorithms directly in MAPDL input files so you can manipulate sparse and dense matrices to solve linear systems and eigenproblems, for example:

APDL Math Commands Examples

```
*SMAT,K,D,FULL,file.full,STIFF      ! Obtain the stiffness matrix from an existing
                                    ! MAPDL full file

*LSENGINE,MySolver,BCS,K            ! Factor this matrix using the BCS sparse solver
*LSFACTOR,MySolver
```

THE JAMES WEBB SPACE TELESCOPE: SENSITIVITY ANALYSIS USING APDL MATH

The James Webb Space Telescope (JWST) is an orbiting infrared observatory that will complement and extend the Hubble Space Telescope's discoveries. The Smithsonian Astrophysical Observatory (SAO) is involved in research and development work on this next-generation telescope, which has a launch date of 2018.

Once the telescope is in orbit, it will observe distant objects in space for as long as two weeks at a time. During that period, the telescope's temperature may change a small amount, and even this minute change will affect optical performance (wavefront error). To verify that performance is within specifications, the NASA Johnson Space Center (JSC) will test the JWST optical telescope element (the eye that gathers light for the instruments) and science instrument payload in a huge vacuum chamber that will bring the telescope temperature down to spaceflight conditions of approximately 30 K. During that test, JSC will make optical measurements of the telescope to compare them with numerical model predictions. To accurately make those measurements, researchers must install many thermal sensors on the telescope so that its temperature state can be understood. Those sensors need to be placed on the parts that have the highest wavefront error sensitivity to small temperature changes; therefore, to place the sensors properly, researchers need to understand the part-by-part sensitivity.

One of the challenges researchers face is to understand the sensitivity of the mechanical behavior of this very complex, large assembly (more than 65 million equations) with respect to a very large number of input material parameters, each of which vary in a set of several different values.

The structure that supports the 18 mirror segments has more than 3,000 graphite-composite and pallet components. To determine the optical wavefront sensitivity of each of these parts would take more than 3,000 CPU hours in excess of four calendar months using the normal brute-force method of changing the thermal strain behavior of each and every part separately.

Taking full benefit of the very local definition of all these parameters — less than 1 percent of the global number of elements are influenced by a variation of one parameter — and starting from a single complete solve on the whole model, researchers defined a set of APDL Math commands to accelerate this sensitivity analysis, using an iterative solve method.

With APDL Math commands, you can create specific customized processes in MAPDL that are adapted to your organization's real needs.

Overview of APDL Math Commands Sequence

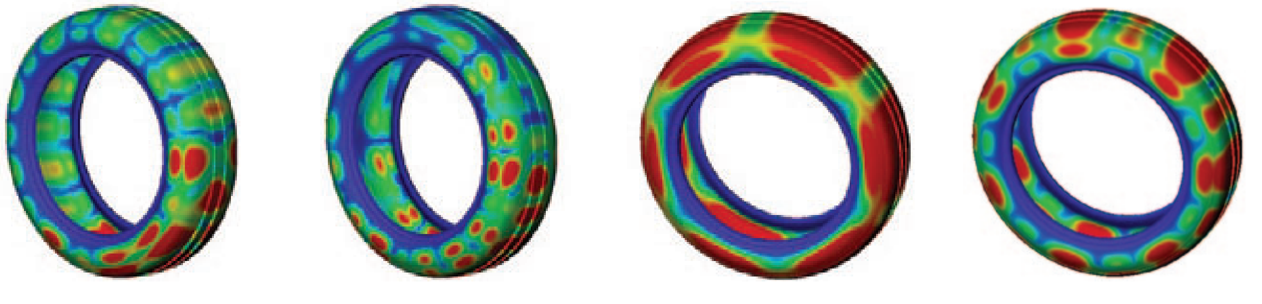
`…` `WRFULL,1` `SOLVE`	! Set up initial state: $K_0U_0 = F_0$! Create the full file but do not solve the ! equations
`*SMAT,K0,D,IMPORT,FULL,file0.full,STIFF` `*VEC,F0,D,IMPORT,FULL,file0.full,RHS`	! Get K_0, F_0 from the full file
`…` `*LSENGINE,MyBcs,BCS,K0` `*LSFACTOR,MyBcs` `*LSBAC,MyBcs,F0,X0`	! Solve the initial state: $K_0U_0 = F_0$! using the BCS sparse solver
`…` `! Parameter variation`	! Introduce small variations in K and/or F: ! $K_1 = K_0 + \Delta K$, $F_1 = F_0 + \Delta F$
`…` `WRFULL,1` `SOLVE`	! Form the matrices K_1, F_1
`*SMAT,K1,D,IMPORT,FULL,file1.full,STIFF` `*VEC,F1,D,IMPORT,FULL,file1.full,RHS`	! Import the new matrix and vector
`*ITENGINE,PCG,MyPcg,MyBcs,K1,F1,X1`	! Use a PCG algorithm, based on the ! factorization of K_0 and the updated ! matrices K_1, F_1 to quickly get the ! updated solution U_1.

On average the new ANSYS sensitivity capability has decreased by approximately six times the amount of time to perform this work. Using this customized process, it is possible to accomplish this analysis in a realistic amount of time.

With APDL Math commands, you can create very specific customized processes in MAPDL that are adapted to your organization's real needs.

OTHER APPLICATIONS

APDL Math allows the rapid implementation of a number of different algorithms, and the commands have been used in implementing specific algorithms.



Tire vibration

Michelin's teams of experts rely on APDL Math technology as part of its simulation effort.

- In rotordynamics, APDL Math was used to develop an algorithm to very quickly directly solve for the critical speeds by resolving a unique and customized eigenproblem. The algorithm replaced the classical loop needed to generate the Campbell diagram.
- In modal analyses, the derivatives of the eigenvalues with respect to a design parameter variation is an item of interest for sensitivity studies. The formula to compute this derivative is known, and is easy to implement with the APDL Math set of commands.
- To identify material properties from a set of experimental data to build an accurate model in MAPDL, APDL Math was employed to perform the interpolation of these quantities to automate this parameter identification step.
- By exporting MADPL substructures (SUB file) to NASTRAN DMIG files and vice versa, customers have been able to work in environments using both codes.

ENHANCE COMMUNICATION WITH THIRD-PARTY TOOLS

APDL Math commands deliver new ways to communicate with third-party tools, such as Matlab®, by offering the ability to import and export data in various sets of file formats (Matrix Market, Harwell Boeing, etc.).

MICHELIN REDUCES TIRE NOISE

Michelin®, a world leader in tires for all types of vehicles, is committed to conducting business in a sustainable and responsible manner, in accordance with its Michelin® Performance and Responsibility program. The program also guides the company's product strategy to satisfy market requirements by offering innovative, high-technology products that comply with standards, regulations and specific uses. To continue to reduce the environmental footprint of tires and to plan for future standards and different levels of labeling (tire classifications based on fuel consumption, wet grip and noise) around the globe, Michelin is redoubling its efforts to significantly reduce the noise that tires produce.

Simulation is the best means for achieving the noise-reduction objective. It can be used to identify and understand the mechanisms that cause noise, leading to development of increasingly innovative products. Michelin's teams of experts rely on APDL Math technology as part of this simulation effort. The technology couples generic digital analysis methods with Michelin's own internal expertise and systems. The combination makes it easier to obtain high-quality data rapidly and provides more accurate noise-reduction diagnostics.

COUPLING THE MICHELIN IN-HOUSE SOLVER WITH THE MAPDL SOLVER

Michelin has very specific, accurate knowledge of the numerical simulation of tire models. The company needs to retain this sensitive, proprietary information, so the knowledge cannot be incorporated into a commercial code. On the other hand, Michelin engineers would like to use the power of the HPC solvers and algorithms developed inside MAPDL, since the MAPDL solver contains a highly optimized algorithm to perform harmonic analyses, allowing the solver to sweep over frequency ranges in a reduced amount of time.

To address these competing requirements, Michelin engineers set up a hybrid solve process that combines an FEM data provider (the Michelin in-house solver) to assemble the system of equations to solve along with the pure numerical solver contained in MAPDL. Communication between the two codes is based on APDL Math features and is performed in the same process. APDL Math commands also offer the ability for Michelin engineers to use the MAPDL eigensolvers on their own FEM problems.

This approach allows Michelin to focus on enhancing its knowledge of tire modeling and behavior, and it lets ANSYS technology solve the resulting equations.

SUMMARY

APDL Math extends the scripting language environment of ANSYS Mechanical APDL to give you access to powerful matrix manipulation routines, including fast and efficient solvers.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA U.S.A. 15317

Send address corrections to
AdvantageAddressChange@ansys.com