SPOTLIGHT ON
THE BUSINESS VALUE
OF ENGINEERING SIMULATION
Technology changes everything

Sometimes the change is minor. And sometimes it rocks an industry.
Transform your business with ANSYS and HP.

ALTERNATIVE THINKING ABOUT CAE

Partnering with the industry leaders in CAE hardware and software will ensure that you do it all faster, better, and more competitively.

www.hp.com/go/CAE
www.ansys.com

Simplify HPC. Share the knowledge.

An online community enabling IT professionals and engineers to share knowledge, gain information and capitalize on the power of High Performance Computing. Join the innovators and visionaries—be part of the momentum.

www.ClusterConnection.com
Who Will Survive?

Engineering simulation may help determine which companies make it through the current economy and strengthen their competitiveness when markets rebound.

Engineering simulation is an indispensable tool in efficient product development processes at a growing number of companies. Engineers use the technology early in the cycle to evaluate concepts, compare alternatives, identify problems and optimize designs. Upfront analysis avoids the slowdowns and expenses of late-stage problem-solving with frantic design fixes and trial-and-error testing. These are among the powerful capabilities that enable companies to reduce costs, shorten time to market, improve quality and create innovative designs.

The ramifications of such benefits can be tremendous in terms of top-line revenue growth and bottom-line savings, and the increased profitability that companies reap is the staggering business value of engineering simulation, which can provide the impetus for executive-level decisions to invest in the technology. This is the theme of this issue’s Spotlight section.

Coverage of this topic is especially timely, given the importance of this business value to companies around the world contending with continuing economic distress, financial uncertainty and volatile markets. Indeed, the competitive advantage provided by smart use of engineering simulation can be a deciding factor in determining which companies survive the current economic chaos. As the articles in this section show, there is no cookie-cutter approach to engineering simulation. Because of the wide range of corporate priorities and product portfolios, the ways in which technology is implemented and business values are obtained are unique for each company.

For example, mobile electronics and transportation system supplier Delphi Electronics and Safety Systems develops more-robust, reliable products and greatly reduces validation failures in prototype testing through a comprehensive program to train engineers at distributed sites around the world in upfront simulation, logging in over 11,000 hours of usage on a single product from ANSYS in a typical year. Tier-one mechatronic system supplier Brose Group had one simulation engineer in the 1990s and now has 45, with CAE usage growing by 50 percent annually. That organization applies engineering simulation in developing higher-quality, lower-cost automotive door and closure systems using tools such as ANSYS multiphysics technology.

Gas turbine component supplier Power Systems Manufacturing helps power-generation companies avoid expensive downtime using ANSYS Workbench based technology to optimize the design of compressor blades. Nozzle manufacturer Spraying Systems Co. strengthens its relationships with customers and provides an additional source of revenue by using software from ANSYS to study and suggest improvements in the designs of customers’ gas conditioning solutions that utilize nozzles in complex pollution control systems.

Clearly, these companies recognize that they’re not just analyzing parts but are building customer relationships, creating value-added services, growing revenue streams and boosting their competitiveness. That’s the true business value of engineering simulation, which NAFEMS Chief Executive Tim Morris aptly describes as a strategic weapon. In his article “Championing Simulation,” he notes that best-in-class companies are often those that make the greatest use of the technology. Investments in simulation make companies more competitive, allowing businesses to emerge from the current recession stronger — and probably with fewer competitors.

John Krouse, Senior Editor and Industry Analyst
SPOTLIGHT ON THE BUSINESS VALUE OF ENGINEERING SIMULATION

OVERVIEW
Profiting from the Investment in Smart Engineering Simulation

AUTOMOTIVE/ELECTRONICS
Leveraging Upfront Simulation in a Global Enterprise
Corporate initiative at Delphi focuses on the benefits of simulation as an integral part of early product design at sites around the world.

TURBOMACHINERY
Designing for Quality
Simulation paired with optimization helps eliminate compressor blade failure.

ADVOCACY
Championing Simulation
NAFEMS champions CAE awareness, delivers education and sets simulation standards.

ENVIRONMENTAL
Extending the Bounds of Customer Service
Spray nozzle manufacturer expands value-added services by using simulation to develop and validate gas conditioning solutions in complex pollution control systems.

AUTOMOTIVE
Opening New Doors
Brose uses simulation to drive product quality, reduce testing and minimize costs.

SIMULATION@WORK

BUILT ENVIRONMENT
ANSYS Sets the Stage
Simulation was used to design the floating stage set used in the latest Bond movie.

ENERGY
Harnessing the Power of Ocean Waves
Engineers use structural and hydrodynamic analysis to ensure that wave-powered electrical generation machines produce maximum energy output and operate effectively for decades.

MARINE
Designing for Strength, Speed and Luxury
Simulation software from ANSYS helps a yacht designer deliver the optimal combination of luxury and performance.

ELECTRONICS
Seeing the Future of Channel Design
NVIDIA uses VeriEye and QuickEye as an extension to traditional SPICE-level simulation approach to design high-performance graphics solutions.
28 ELECTRONICS
Hot Topics: High-Capacity Hard Disks
Samsung uses simulation to improve thermo-fluidic performance of hard disk drives.

30 AUTOMOTIVE
Fatigued by Stress Limitations
The combination of fe-safe and ANSYS software helps Cummins improve life prediction accuracy.

32 ELECTRICAL
Managing Heat with Multiphysics
Multiphysics simulation helps a global company design better electrical products.

34 AEROSPACE
Up, Up and Away
Simulation-driven innovation delivers a new ejection seat design for a military aircraft in less than 14 months.

36 MARINE
Propelling a More Efficient Fleet
Rolls-Royce uses simulation for propeller design to reduce marine fuel consumption.

DEPARTMENTS

38 ANALYSIS TOOLS
Staying Cool with ANSYS Icepak
Thermal management solution predicts air flow and heat transfer in electronic designs so engineers can protect heat-sensitive components.

40 Analyzing Vibration with Acoustic–Structural Coupling
FSI techniques using acoustic elements efficiently compute natural frequencies, harmonic response and other vibration effects in structures containing fluids.

44 PARTNERS
Integrating CAE Tools: a Package Deal
Moldex3D and ANSYS Mechanical team up to simulate microchip encapsulation.

47 ACADEMIC
Sailing Past a Billion
Racing yacht design researchers push flow simulation past a meshing milestone.

50 TIPS AND TRICKS
Optimizing Options
Technologies converge in ANSYS Workbench for parametric fluid structure interaction analysis.

52 Analyzing Nonlinear Contact
Convenient tools help analyze problems in which the contacting area between touching parts changes during the load history.
WEB EXCLUSIVES
These additional articles are available exclusively on www.ansys.com/exclusives/209.

ANALYSIS TOOLS
The Immersed Boundary Approach to Fluid Flow Simulation
The add-on immersed boundary module, jointly developed by ANSYS and Cascade Technologies, is a preliminary design analysis tool that dramatically reduces the amount of time needed for fluid flow simulations and provides fast results by directly addressing the challenges associated with this meshing step.

ENERGY
Tracking Down Vibrations Fast with FSI
Offshore oil and gas operations can lose significant revenue from even a few days of downtime, so they must efficiently study and rectify equipment failures that could shut down any part of the platform. With high-speed iterations between mechanical and fluids software, Denmark-based Lloyd’s Register CDS used fluid structure interaction to quickly pinpoint the cause of damaging vibrations and to assess new designs.

AUTOMOTIVE
Designing Giants for Tough Work
In the mining industry, the engineering challenge is to design lightweight trucks strong enough to withstand harsh operating conditions. Liebherr engineers rely on ANSYS structural simulation technology to develop giant diesel electric trucks designed to withstand hostile conditions while providing maximum load capacity.

ENVIRONMENTAL
Keeping New Orleans Flooding at Bay
In flood-prone coastal Louisiana, structural analysis of massive platforms holding storm drainage equipment helped to predict and address vibration problems. Mechanical Solutions, Inc. performed the simulation, the results of which were actually validated during a subsequent Category 2 hurricane.

SPORTS/AUTOMOTIVE
Overtaking Race Car Design
Italian design firm Fioravanti has proposed an innovative Formula 1 car that could bring more excitement to the race. Using aerodynamic simulation, the company designed a car that would allow drivers to more frequently overtake the other racers — safely.

MARINE
Designing Safe and Reliable Ships
Delta Marine Engineering uses upfront simulation to minimize risk and maximize performance well before launch day. The Turkish shipbuilder identifies and corrects troublesome vibration problems to comply with international standards as well as to ensure the safety of the crew and a long life for the ship.

PARTNERS
Enabling Detailed Chemistry
CHEMKIN-CFD is an advanced chemistry simulation technology that efficiently and robustly couples accurate chemical kinetics to flow simulations. Its developer, Reaction Design, provides the software free to ANSYS FLUENT users who are looking to improve the accuracy of their analyses.

For ANSYS, Inc. sales information, call 1.866.267.9724, or visit www.ansys.com.
For address changes, contact AdvantageAddressChange@ansys.com.
To subscribe to ANSYS Advantage, go to www.ansys.com/subscribe.
Email the editorial staff at ansys-advantage@ansys.com.

Executive Editor
Fran Hemler
Managing Editor
Chris Reeves
Senior Editor and Industry Analyst
John Knouse
Art Director
Dan Hart
Editors
Erik Ferguson
Shane Motechens
Mark Ravenstahl
Ad Sales Manager
Helen Renshaw
Editorial Advisor
Kelly Wall
Designer
Miller Creative Group
Circulation Manager
Sharon Everts

About the Spotlight
Upfront engineering simulation may help determine which companies make it through the current economy and strengthen their competitiveness when markets rebound. The spotlight begins on page 5.

Neither ANSYS, Inc. nor the senior editor nor Miller Creative Group guarantees or warrants accuracy or completeness of the material contained in this publication.

ANSYS, ANSYS Workbench, Ansoft Designer, CFX, AUTODYN, FLUENT, GAMBIT, POLYFLOW, Airpak, DesignSpace, FIDAP, Flotran, Iceboard, Icechip, Icomax, Icospark, FlowWizard, FLOWLAB, Go/Turbo, MaxSim, Neoxim, Q3D Extractor, Maxwell, Simplorer, Mechanical, Professional, Structural, DesignModeler, TGrid, A*Environment, ASYS, AQWA, AutoRealGas, Blademodeler, DesignXplorer, Drop Test, ED, Engineering, Knowledge Manager, Emag, Fatigue, Isoprop, Icewave, Mesh Morpher, Paraview, TAS, TASSISTRESS, TASFET, TurboGrid, Vista, VT Accelerator, CADDIE, CoolSim, Simwheel, Turbo Package Analyzer, RFmerit, RFport, HFSS, Full-Wave SPICE, VerifyEye, QuickEye, Optometrics, TPA, AnsoftLinks, ePhysics, Simulation Driven Product Development, Smart Engineering Simulation and any and all ANSYS, Inc. brand, product, service, and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2009 ANSYS, Inc. All rights reserved.
To survive and profit in the current demanding business environment, organizations must engineer high-quality and innovative products, design and manufacture them for the lowest possible cost, and then beat their competitors to the global marketplace. To meet that challenge, many of the most innovative companies in the world use engineering simulation to develop products that dominate the marketplace.

There is no single solution for every organization, although one basic principle is universal: Engineering simulation is performed up front in the product development process at companies large and small because Simulation Driven Product Development provides quantifiable value. Tools used to test many alternatives for product designs before prototyping provide engineering teams with the ability to get products to market quicker. It also allows these teams to rapidly iterate on designs so they can determine the best and most innovative alternatives. In addition, reducing the number of expensive prototypes provides direct cost-savings benefits.

As simulation technology continues to expand in depth and breadth, companies are able to simulate a greater range of their product requirements. Multiphysics simulation delivers results that approach real-world conditions, supplying an even more reliable basis for making product design decisions. However, these larger solutions may come at the price of computational time. High-performance computing continues to address this problem, making it possible to solve complex problems on a laptop, which, several years ago, would have been impossible using an entire room of processors.

Capturing and reusing engineering data generated from numerous design iterations and increasing types of simulation help to maintain product design efficiency and protect intellectual property. Managing engineering knowledge is vital for individual users, small teams and enterprise-wide implementations.

Engineering teams and individual designers can be even more productive when using simulation technology with an adaptive architecture, which gives them the capability to iterate with their existing CAD files, incorporate other software tools, explore what-if scenarios and develop reusable models all within the same working environment. Not only does this speed up the product development process, it allows team members to apply their experience and skills in the most efficient manner and generates more opportunities to develop the innovative products required for continued profitability.

The world’s most successful companies turn to simulation solutions from ANSYS, whose products are used by 16 of the top 20 most innovative companies in the world today, according to a BusinessWeek report prepared by the Boston Consulting Group and by 97 of the top 100 industrial companies on the FORTUNE Global 500 list. Today, more than ever, these companies have invested in Smart Engineering Simulation to increase the efficiency of their processes, improve the accuracy of their virtual prototypes, and capture and reuse simulation processes and data. ANSYS delivers solutions to help organizations profit in today’s turbulent economic environment.

Spotlight on the Business Value of Engineering Simulation

Profiting from the Investment in Smart Engineering Simulation

www.ansys.com
Most high-technology companies now realize the potential benefits of simulating the performance of their products using tools such as finite element analysis (FEA). They also clearly know that performing analysis early in the design cycle has the potential to identify and solve design problems much more efficiently and cost effectively compared to handling them later. One of the leading companies in employing upfront analysis throughout the product engineering organization is Delphi Electronics & Safety Systems — a major division of Delphi Corporation specializing in mobile electronics and transportation systems for the automotive and consumer product industries.

Beginning in the late 1990s, Delphi Electronics & Safety embarked on a program to take full advantage of FEA in the product development process. Along with other companies, Delphi Electronics & Safety had been using FEA in a more limited way as a troubleshooting tool often later in development. The new initiative intended to employ finite element analysis as an integral part of the product development process — especially focusing on the use of simulation up front in the design cycle.

To achieve this goal, Delphi put into place a comprehensive program to train design engineers in the use of FEA in the early stages of the design process. This program began by classifying engineers according to their skill levels in use of FEA or other commercially available analysis tools.

Steady-State Thermal Analysis

The most common use of the ANSYS Workbench tool at Delphi is by design engineers engaged in product development. Analysis types include steady-state thermal, free vibration and linear static stress analysis. More advanced types of analysis, including those involving material or geometric nonlinearity, transient loading, fluid flow and multiphysics, are performed by full-time analysts using products from ANSYS or other commercially available analysis tools.

Delphi produces a number of products including those for use in the automotive and consumer product sectors that must meet stringent thermal requirements. A steady-state thermal analysis is, in many cases, the first step in ensuring that the final product will meet the thermal requirements of the customer. Based on usage data collected annually, the ANSYS DesignSpace tool is widely used to perform this type of analysis. Shown in the accompanying figure is an example of the results of a steady-state thermal analysis of an integrated circuit package used in a transmission controller unit. Once the steady-state performance is established, transient and system-level analyses are performed to completely characterize the system.
the program incorporated use of FEA into the Delphi Electronics & Safety product development plan. Safeguards such as peer reviews, engineering fundamentals training and mentoring were implemented to ensure proper use of FEA. Furthermore, Delphi Electronics & Safety has restricted use of this technology to engineers and scientists with a minimum of a bachelor’s degree. Training in the use of the structural mechanics simulation software — in this case, ANSYS DesignSpace that uses the ANSYS Workbench platform — is a prerequisite at Delphi Electronics & Safety. Occasional users such as product engineers utilize the software to perform linear and static analyses. More advanced analyses involving nonlinearity or transient loading are referred to full-time analysts.

Today, the company has incorporated the use of structural mechanics simulation into the Delphi Product Development Process (PDP) as a requirement. The PDP begins with the concept stage and proceeds to the validation stage when prototypes are built and tested, and finally the program is handed off to manufacturing. This has led to developing much more robust and reliable products as well as greatly reducing or eliminating validation failures. This process is enforced by a Design Failure Modes and Effects Analysis (DFMEA) plan represented by a spreadsheet of possible failure modes for a product and the required analyses to show that the product is immune to the specific failures.

A large number of engineers around the world in the overall Delphi organization use these tools, including the full suite of software from ANSYS within the ANSYS Workbench interface, to perform thermal, stress, vibration and other general analysis in the course of product development. In 2007, the number of ANSYS DesignSpace users exceeded 200, and approximately 30 percent were Delphi Electronics & Safety engineers. Delphi Electronics & Safety users globally logged 11,151 hours of usage on the software, or 34 percent of the total for all sites internationally. The licenses for many CAE tools — including ANSYS products such as ANSYS DesignSpace — are supported from servers in Michigan, in the United States, allowing engineering management to stay up to date on the use of these tools.

By adopting a comprehensive approach for implementing FEA across the worldwide organization, Delphi has effectively incorporated an extremely powerful technology into the product development process. The initiative to focus on upfront analysis in particular has resulted in outstanding business value for Delphi in terms of improved designs developed very efficiently. The use of the ANSYS Workbench platform has certainly facilitated this process by providing the ability to perform a variety of analysis types of different complexities in the same familiar environment. Perhaps the best indicator of the effectiveness of this software in a business context is management support for its widespread use by such large numbers of Delphi engineers around the world.

Finding Natural Vibration Modes

ANSYS DesignSpace software is often used for determination of the natural modes of vibration of a system. Many of Delphi’s products are used by the automotive industry, and the first step in establishing that a product can be used in a vehicle is to ensure that the first few modes of vibration of the product are beyond the minimum values that can be excited by the vehicle’s operation. The accompanying figure shows the first mode of vibration for a bracket used to support an airbag control unit.

Using the ANSYS DesignSpace tool to perform modal analysis, product engineers are able to determine if any changes to the initial design are needed to improve the vibration characteristics of the system. The design then proceeds to the next stage, in which harmonic and power spectral density (PSD) analyses are performed and any required changes are made.
Designing for Quality
Simulation paired with optimization helps eliminate compressor blade failure.

Power Systems Manufacturing (PSM) is a global provider of aftermarket gas turbine components in the industrial power generation industry. The company’s product line includes stationary and rotating airfoil components, low-emission combustion systems, and advanced components for GE Frames 6, 7 and 9 and Siemens 501F-class machines. In one specific redesign case, an original equipment manufacturer (OEM) first-stage compressor blade design for a popular large gas turbine used for generating electricity was not reaching its expected life.

Field failure scenarios included blade breakage resulting in considerable downstream damage. Companies running these engines then had to shut them down for lengthy periods while making repairs, losing the revenue from the electricity that the engines would normally generate. “Several of our customers were running into the same problem with this compressor blade and asked us if we could improve and fix the issue,” said Page Strohl, former lead structures engineer for PSM. “Any new design that we developed had to fit into exactly the same envelope and to have the same aerodynamics as the original blades.”

PSM designers and engineers worked together to model and simulate the original blade design, taking into account the aerodynamic and centrifugal loads. Using simulation, PSM located the highest static stresses in the blade. In addition, PSM engineers determined that the blade design had several vibratory modes that were excited during engine operation that when coupled with the static stresses could result in failure. PSM’s analyses predicted
that the blade would fail in the exact locations at which failures occurred in the field.

Using the ANSYS DesignXplorer tool to study a range of design variations, Strohl defined the blade parameters he wanted to vary, assigned acceptable parameter ranges, and identified the variables he wanted to optimize, which in this case were blade natural frequencies and peak steady stresses at several locations on the blade. Based on the peak stress locations, design space definitions were created. ANSYS DesignXplorer software redefined the CAD model and generated the series of design variations needed to carry out experiments for the entire range of parameters. Strohl then used the ANSYS Workbench environment to mesh each of these designs, solve the models, capture the results and perform the statistical analyses needed to identify the optimal design. Once Strohl identified an optimal design, the aerodynamics team made minor design modifications in order to ensure that the design performed aerodynamically as required.

The resulting design reduced all peak steady stresses. The vibrational issues were eliminated as well with these design modifications. The blades are performing as expected and have been in operation since May 2008. Additional sets are now on order.

Strohl concluded, “Key to these improvements was the ability of the ANSYS Workbench platform to interface with our CAD system, allowing us to quickly prepare new geometries for analysis and to control the CAD system to explore a design space. We were able to quickly iterate to a design that was optimal, all while maintaining the same aerodynamic properties as the original design.”

---

**Transitioning into the ANSYS Workbench Environment**


**You’d been a long-time user of the traditional ANSYS Mechanical interface. Did you have any hesitance transitioning to the ANSYS Workbench platform?**

I found it hard to put down something that I was already comfortable and proficient with in order to start using something completely new. It wasn’t until I attended an “Intro to ANSYS Workbench” training that I attempted to really utilize it.

When this compressor blade analysis came up and we were faced with a time crunch, I knew I could save time by taking advantage of the connectivity to Pro/ENGINEER® and the fact that the actual component settings stay with the model as it is passed from the CAD environment to ANSYS Workbench. By using this approach, the long lead tasks became the CAD modeling and the aerodynamic analysis efforts, not the structural analysis-specific ones.

**Has using ANSYS Workbench changed how you approach some of your projects?**

I find that I look for projects that can take advantage of the ease-of-use benefits that ANSYS Workbench offers. PSM recently started a major redesign of another compressor airfoil, and we jumped right into ANSYS Workbench and ANSYS DesignXplorer for it. I created the baseline model and showed the aerodynamicist how to duplicate it, regenerate a modified Pro/ENGINEER model, then solve and check the results. I actually gave my work away.

**How did this new process work?**

It was very easy to perform the simulation — a great benefit of ANSYS Workbench. If the geometry had problems reading in or didn’t pass all of the built-in checks, I would slide my chair over to the aerodynamicist’s workstation and take a minute or so and fix things. It was really a benefit in that I was able to do other work while we were in the iterative phase of the design. I, like many, am workload-challenged these days, and ANSYS Workbench helped to relieve me of that particular task.

From a larger project perspective, it gave the aero guy a good look at all the items that needed to be reviewed outside of his aero world. Several of my coworkers joke with me that my job consists purely of clicking the mouse button one or two times for an analysis job; they say that the hard ones are when I have to click the mouse three times. The real trick in this case was setting up the original baseline model. Once I did that correctly, it was a piece of cake to have someone else turn the analysis around with ANSYS Workbench.
NAFEMS, founded more than a quarter-century ago, is an impartial best-practice champion of computer-aided engineering (CAE) standards. A nonprofit organization headquartered in the United Kingdom, NAFEMS provides information to secure the best returns on investment in CAE software, to develop and enhance simulation capabilities, and to ensure the safest and most effective use of the software. About 940 companies around the world, from large multinational corporations to small engineering consultancy firms, are members of NAFEMS, and this number is growing. Although the largest proportion of members is involved in finite element analysis, the computational fluid dynamics (CFD) group is expanding rapidly. Members belong to almost every industry sector.

ANSYS Advantage staff interviewed Tim Morris, chief executive of NAFEMS, on his viewpoint about trends in CAE and the value of engineering simulation.

**Is the manufacturing industry taking full advantage of engineering simulation technology?**

The power and scope of simulation technology has increased dramatically in the past 10 years. Simulation should now be at the heart of the design process, driving it, not merely validating it in the latter stages. To do this requires change in how simulation technology is deployed in many organizations. Simulation engineers need to be better integrated as a fully involved part of the product development strategy from the very beginning. They also need to understand the commercial imperatives driving development. Product development managers need to better understand what engineering simulation can offer, in areas such as shaping external design appearance, instead of leaving this to marketing designers. By embedding engineering simulation in the product development strategy, technical products that meet all market needs can be realized, and engineering simulation can deliver best value by increasingly compressing development processes in order to reduce time to market.

Engineering simulation is a strategic weapon inside companies today, especially for nimble organizations that have a philosophy of core adoption and deployment.
because it leads to competitive advantage. Financial and commercial pressures from an ever-more competitive market have led to companies’ increasing reliance on engineering simulation to cut costs and reduce design and development cycles.

How important is a multiphysics approach to the development of engineering simulation?

By relying on engineering simulation as the primary tool to develop new products or processes, engineers and designers often want to simulate as near to the real world as possible. A multiphysics approach is inevitably going to be more accurate at simulating the real world than one that uses only CFD or FEA, for example. As high-performance computing (HPC) continues to improve, we will see more- and more-realistic multiphysics simulations in engineering. As engineering simulation becomes more powerful and additional companies come to rely on simulation to develop products and processes, being able to employ multiphysics will become more and more critical.

What are the current challenges facing NAFEMS?

As an organization, our broad challenge is to continue to sharpen our focus on the commercial application of engineering simulation. For example, we are actively seeking ways to bring about the kind of mutual understanding among simulation engineers, product development and business teams that is necessary to put engineering simulation at the very heart of product development strategies. Another example is to address the lack of CAE standards.

In college education, we need to establish a set of learning outcomes for simulation engineers, and knowledge capture from more-experienced engineers is essential for best practices.

Although bigger companies usually have rigorous engineering simulation processes, small and medium enterprise companies may not. NAFEMS would like engineers to understand the reliability of their CAE analyses. We aspire to establish grades of competencies for good simulation based on experience for these engineers.

What is the significance of CAE in the current economic climate?

Simulation ultimately helps companies to save money. It is all about making the design process more effective and efficient. Simulation empowers engineers and designers to envision and develop better designs — in which better might mean lighter, cheaper or stronger. While companies might need to cut back on manufacturing in these times of recession, the smarter companies will not cut back too much on design or research but, rather, will use the opportunity to improve both products and design processes.

Continued investment in simulation will continue to bring rewards in terms of making companies more competitive and should allow these businesses to emerge from the recession in a stronger state, and quite probably with fewer competitors. What we do know, from independent research that we have been involved with, is that the best-in-class companies are often those that make the greatest use of simulation.

Where is engineering simulation heading in the next 10 to 20 years?

The developments that HPC makes possible are very exciting and could transform the complexity of physics that can be simulated to the point that it may be possible to simulate right down to the molecular level. One day, ambient intelligent environments, ultra-high-bandwidth networks, pervasive wireless communications, knowledge-based engineering, networked immersive virtual environments and powerful games engines will transform multiphysics CAE for product design, creation, validation and manufacturing.

In the future, mesh-insensitive iso-geometric pre-processing techniques will become more common. We will see the gaming industry and Hollywood-style post-processing and visualization being pulled into CAE more and more. More stochastic simulation as opposed to deterministic predictions will be performed because, as more computing power becomes available, it will be possible to study a range of analyses rather than worst-case/best-case simulations that are the trend today.

Simulation data management is an up-and-coming issue in our industry. Good standards are required with petabyte-sized files that may soon become common. Security of data and information is crucial with enterprise-wide projects and collaborations across the world.

FEA, CFD and other related technologies are still very much in their infancy. Engineers in the future may look back and be amused at how crude and unreliable the methods of today are when compared with the technology that is yet to come. The technology itself continues to be developed at an ever-increasing rate, but the complexity of the applications that industry would like to tackle continues to exceed the available capabilities.
Spray nozzle manufacturer expands value-added services by using simulation to develop and validate gas conditioning solutions in complex pollution control systems.

By Rudolf Schick, Vice President, Spray Analysis and Research Services, Spraying Systems Co., Illinois, U.S.A.

Founded in 1937 in a small converted garage, Spraying Systems Co. is now a worldwide leader in spray technology, producing a wide range of spray nozzles, automated spray systems, specialized fabricated products and accessories. Customers that use this technology are in hundreds of industries, including steel, paper, food, chemical, petrochemical, pharmaceutical and metal fabrication. Spraying Systems Co., always on the lookout for ways to improve product development and expand the value-added services it provides to customers, began using ANSYS FLUENT fluid dynamics software a few years ago in analyzing nozzle behavior. The company quickly discovered that the simulation software is also ideally suited for studying the design of its customers’ gas conditioning solutions that utilize nozzles in complex pollution control systems.

Installed in industrial furnaces, burners, refineries, processing facilities, power plants and other sites, gas conditioning systems remove toxins such as nitrous oxide (NO_2) and sulfur dioxide (SO_2) from exhaust gases prior to release into the atmosphere. Spraying Systems Co. manufactures systems that use spray technology to cool and otherwise treat the gas before it enters specialized pollution control equipment. For example, spraying water into an exhaust stream cools the gas from 1,430 degrees F to 620 degrees F — a temperature that ensures optimal performance from downstream pollution control equipment. If the spray does not enter the gas stream at the correct angle, or if too much water is injected, droplets will not fully evaporate. The resulting acid-laden mist can impinge on duct walls, equipment and other parts of the structure, causing erosion and damage.

Sizing and positioning nozzles in these gas conditioning systems to avoid impingement and other spray problems are demanding tasks. Engineers must calculate numerous variables such as flow temperature, gas velocity and exhaust pollutants. Moreover, they must determine effective spray patterns for complex ductwork — especially when retrofitting older exhaust systems that have existing flanges, recesses and twisting geometries. Because there is no standard computational development method available to solve such difficult problems, companies often spend months of time and perhaps millions of dollars in prototype tests and late-stage troubleshooting. Worse yet, some of these poorly designed systems are put into service. Plant owners then may face costly penalties for failure to comply with emission control regulations, as well as expensive downtime for system redesign.

A far better approach to gas conditioning system design uses computational fluid dynamics (CFD).
In one recent case, engineers performed such work on a gas conditioning system retrofit project for a refinery exhaust tower. Due to physical constraints of the retrofit, the design called for the quench system to be installed and controlled from a fixed height and single side panel of the square tower. The custom solution called for the installation of three FloMax® FM25A air atomizing nozzles from Spraying Systems Co. Numerical calculations accurately determined the resulting nozzle pressure, liquid flow rate, atomization air flow rate and drop size for the nozzles.

To analyze the performance of the proposed system, the 3-D geometry of the exhaust duct system was generated in CAD based on information provided by the customer. Spraying Systems Co. then imported this geometry into the pre-processing tool from ANSYS to create a mesh using elements small enough to provide a high level of detail for the study. To understand the behavior of the spray, engineers analyzed the model to determine droplet velocities and trajectories, exhaust pressure and temperature distributions and overall spray concentration throughout the duct tower.

The CFD simulation indicated significant problems with the proposed gas conditioning system, including strong swirling flow, several regions of low pressure, uneven velocity and temperature profiles, and near-zero flow in some local zones. These conditions would likely result in incomplete droplet evaporation along with impingement of droplets on internal walls, the duct structure and downstream equipment. Using CFD, engineers performed iterative simulations to study and modify the proposed nozzle insertion depth, rotation angle and insertion angle configuration. Various nozzle configurations were evaluated based on the baseline gas flow through the existing tower.

These iterations enabled the engineers to develop a more effective design that optimized gas flow and improved the evenness of the gas velocity. The resulting flow uniformity helped ensure better temperature distribution and droplet evaporation. The final duct design improved evaporation by over 10 percent with impingement of droplets on walls and other parts of the structure virtually eliminated. In addition, temperature profiles at the duct exit were controlled to within 7.7 percent of nominal temperature requirements — a considerable improvement over the 42 percent variation found in the initial design.

Spraying Systems Co. now routinely uses this simulation approach to validate many of its customers’ proposed retrofit designs and to design new gas conditioning systems. The method provides a unique value-added service, strengthens the company’s relationships with customers and provides an additional source of revenue. Simulation enables engineers to efficiently complete several additional projects each month, ensuring successful performance of installed equipment.

The company continues to investigate opportunities to leverage engineering simulation in enhancing value-added services to customers. Recently, they deployed ANSYS Mechanical software to integrate more structural analysis into the development cycle for determination of stress and deformation of nozzle mounting lances. Also, use of the ANSYS DesignXplorer tool enables engineers to study alternative designs quickly and to help zero in on optimal solutions.

In this manner, simulation has become a key component in the company’s design capabilities, and its value has been proven many times over. In addition to providing customers and regulatory agencies with documentation of system performance, simulation has become a powerful new communication tool for the company. Spraying Systems Co. has a 3-D projection studio that allows customers to stand virtually inside the application to see and experience the technical details of the proposed solution. In this respect, simulation is as much a sales tool as a design tool, enabling the company to increase business significantly in this growing market.
Opening New Doors

Brose uses simulation to drive product quality, reduce testing and minimize costs.

The Brose Group, headquartered in Coburg, Germany, is a tier-one supplier specializing in developing and manufacturing mechatronic systems and electric drives for body and interior of automobiles: components such as door systems, seat adjusters, closure systems and electric drives in particular — to more than 40 automobile manufacturers and suppliers worldwide. The company has 52 locations worldwide and a global staff of more than 14,000 people.

ANSYS Advantage staff recently interviewed Sandro Wartzack, a simulation and knowledge-based engineering manager at Brose. Dr. Wartzack completed one of the first Ph.D.s in Germany for the integration of finite element analysis (FEA) methods in combination with knowledge-based engineering into design processes.

Can you describe the simulation strategy at Brose?

At Brose, we consider how our computer-aided engineering (CAE) usage impacts each area in product development. We try to optimize our CAE approach to maximize product quality, reduce testing and minimize cost. Brose had one simulation engineer using software from ANSYS in the 1990s. Today Brose has around 45 simulation engineers across the company. When integrating simulation into our development efforts, we often find that savings or improvements in our product designs trickle on to client savings, which can be significant.

Our door system design effort is a great example: By using simulation, we have been able to reduce one specific door system’s mass by approximately 50 percent compared with its design 10 years ago. Because the new designs use less material, original equipment manufacturers (OEMs) that install these components in their vehicles ultimately produce a lighter and, therefore, more fuel-efficient and cheaper automobile. These cost savings get passed on to the car vendor and buyer too.

How does the ANSYS vision for multiphysics tools, virtual prototyping and a single simulation environment like ANSYS Workbench fit in at Brose?

We have a lot of CAE specialists around the world who need to work closely together. The multiphysics products
“A typical door assembly from the Brose Door Systems Business Unit consists of at least 20 to 30 components. By automating portions of our FEA processes within the ANSYS Workbench environment, our engineers have shortened their simulation times, for both static and transient analyses of these complex designs, by as much as a factor of five.”

— Sandro Wartzack
The Brose Group

and ANSYS Workbench environment allow us to do this. Being able to perform structural and fluid simulations in one environment is a great benefit, and the continuous improvements that ANSYS makes to its solvers — really the foundation level for these technologies — can only improve things.

We typically take more than 1,000 hours for testing of new components today, but we want to reduce prototype testing, replacing it with more virtual testing. I believe that development processes tightly integrated with simulation are the future. For example, if we were devising a new plastic part with a certain fiber orientation, ideally the engineers would feed designs into an automated production-like analysis environment for optimization and virtual testing.

How has the growth in the high-performance computing sector affected your simulation approach?

We invariably target a typical nonlinear simulation cycle of 12 hours to 20 hours per simulation so that we can run structural simulation cases to get overnight results. Our simulation model sizes and complexities increase every week, and the need for powerful hardware is a never-ending process. By using this approach, our CAE engineer productivity is much higher now than it was 10 years ago. We typically had one or two processors available per engineer a decade ago, whereas we now have a supercomputing cluster.

We incentivize our CAE engineers financially to find innovative ways to streamline their CAE design processes, targeting changes that might, for example, reduce a typical 250-hour CAE design process down to 80 hours or less. This allows us to introduce capture and use of our best practices and experience to our benefit.

Where do you see Brose technology going in the next 20 years?

With regard to environmental issues, we are looking at more lightweight products to minimize carbon dioxide emissions. I also see acceleration in continuing trends for automotive safety feature improvements. More and more, we need to produce products that are automatically “presafe” electronically. From my viewpoint in the automotive door and seat system sector, I see customization and implementation of modular components as a focus for the automotive industry of the future. For Brose, this means moving to develop “plug and play” modular door concepts, with separate outer panels and colors to match different day-to-day situations.

Where do you feel engineering simulation will move over that time?

We are literally growing our CAE usage by 50 percent every year now, because it is so central to our business. The increased usage of simulation at Brose today has resulted in the development of products with lower mass and, consequently, lower costs. In 10 years’ time, I could foresee that we will do significantly less physical prototyping but, rather, focus on full and accurate single-environment multiphysics simulations and virtual prototyping.

I also think the area of visualization is very important. We are already seeing 3-D co-operative visualization work with virtual meeting rooms. The ability to use simulation results to convey design and development choices together with worldwide distributed teams is going to be vital.
Simulation was used to design the floating stage set used in the latest Bond movie.

By Gerhard Lener, ZT Lener, Feldkirch, Austria

An important scene in the latest James Bond movie *Quantum of Solace* takes place in and around the Bregenz Festival’s stunning open-air floating stage. This European stage, originally constructed for the opera *Tosca*, was built at a cost of nearly $8 million and features a huge eye, 31 meters (101 feet) high by 48 meters (157 feet) wide with an independently moving 9-meter-diameter eyeball. The structural design of the stage was validated to ensure that it could safely withstand environmental loads, loads caused by moving various elements of the stage and loading during assembly of the stage.

Finite element analysis predicted the stresses and deformations in the structure at various wind speeds, at different positions of the moving elements and in various stages of construction. Linear analysis was used to check the structure for serviceability, then nonlinear analysis was performed to ensure that the structure could withstand even higher loads without failing catastrophically. ZT Lener used the broad set of analysis capabilities in ANSYS Mechanical software to analyze the *Tosca* stage because it enabled evaluation of the structure from every possible standpoint — all within a single simulation environment.

*Opera Stage also a Movie Set*

A key *Quantum of Solace* sequence shot at the Bregenz stage occurs during a production of the opera *Tosca*. The eye portion of the stage changes throughout the performance of the opera to become a projection screen, an opening door, an execution platform and a ledge from which a stunt-fall into the lake is performed.

Bregenz is the capital of Vorarlberg, the westernmost state of Austria, located near the border with Germany and Switzerland. Every two years, the Bregenz Festival constructs a new floating stage on Lake Constance for presenting a single opera. The latest floating stage was built in 2007; its amphitheater has about 7,000 seats, and, over two years, approximately 320,000 people will have seen *Tosca*. These stage sets always represent complex engineering constructions that have to simultaneously fulfill artistic and strength requirements. Because no stage set is similar to a
previous one, each becomes a new challenge for the entire team, which has to find solutions in many engineering disciplines.

Stage Presents Structural Challenges

The biggest challenge in the Tosca stage design was providing the strength needed to safely move the eye while staying within the weight limits of the foundation. The moving parts of the stage weigh about 250 metric tons, while the entire stage and foundation weighs only 463 metric tons. Another important limitation is that each component must be moved to the stage by a crane that can handle only about 12 tons. The small bridge between the stage and the land is limited to a mere 1 ton per square meter. This means that any larger components must be moved in smaller pieces and assembled on the stage itself.

Further challenges resulted from the components’ materials: The eye and eyeball are made of a composite construction with a steel frame and a wood outer surface. The composite construction increased the complexity of the analysis, since connecting the steel and wood in a shear plane provides additional stiffness beyond the sum of the properties of the two materials.

Broad Range of Analysis Tools

The stage design began as a 100-to-1 scale model, provided by the stage designer, that was used to create a 3-D Pro/ENGINEER® computer-aided design (CAD) model that guided the structural design. ANSYS Mechanical software was the ideal tool for simulating the floating stage because it provides a very wide range of tools — including linear and nonlinear analysis — addressing materials ranging from metal to rubber, a wide range of solvers and the ANSYS Workbench environment, which provides bidirectional communications with most CAD systems. In performing structural analysis for the last six floating stages for the Bregenz Festival, the design team has encountered a very wide range of structural analysis problems, and technology from ANSYS has been able to handle every one.

ZT Lener employed the traditional version of ANSYS Mechanical software for the main structure because it allowed use of a script to generate input files that automated the process of analyzing the structure at intermediate positions. Engineers modeled the eye’s steel supports with beam elements and its wooden surface using shell elements. Powered by two hydraulic cylinders, the eye rotates 90 degrees between the vertical and the horizontal position. Many structural members receive their highest loading in intermediate positions of rotation, so it was necessary to analyze the

The eyeball portion of the stage is far more than a static background: The iris and pupil were engineered to rotate and fold out via hydraulics, creating a horizontal performance space. The iris also serves as a screen for special visual effects and a door that opens to reveal yet another scene. Copyright Bregenz Festival/Karl Forster.
structure at many different positions to be sure that no structural member is overstressed. The script moved the mechanism through a range of positions and tracked the highest stresses and deformations on each area of the model throughout the entire range. Under wind loading, the eye structure deforms as much as 127 millimeters (5 inches).

Analyzing Structural Details

The team modeled the triangular brackets that connect the eye to the rotating shaft, critical parts of the steel support structure, in the ANSYS Workbench environment. ANSYS Workbench makes it very easy to bring the CAD model into the analysis environment. Based on the analysis results of the triangular support, the team changed the wall thicknesses and positions of the stiffeners on the brackets to reduce deformations and stresses to safe levels.

Because of the transportation limitations already mentioned, it was critical to model the structure at various stages of construction. For example, the steel structure of the eye is much weaker before the wooden shell is assembled, but, on the other hand, it also experiences less wind pressure. Simulation verified that the stage could perform all movements at normal speed at a wind speed up to 50 kilometers per hour (kmh). There is a range of wind speeds above 50 kmh at which the stage can be moved — but at a slower speed. At wind speeds above this level, the stage needs to be moved to a specific position, where it is best able to resist wind loading, and held there.

Evaluating Ultimate Limits of Structure

The ultimate limits of the stage structures that take plastic elastic capacity into account were also evaluated with the loads multiplied by safety factors ranging from 1.3 to 1.5. The structure had to be designed to withstand these design loads and with elastic deformations under characteristic loads (safety factor 1.0). Dynamic analysis on various parts of the structure ensured that it would not resonate nor cause vibrations that might interfere with a performance or damage the structure. The mode shapes and frequencies of the eyeball were calculated to ensure that it would not resonate when several people moved on it at the same time. When the decision was made to film the James Bond movie on the stage, 1,500 kg (3,306 pounds) of lights had to be installed in the upper corner of the eye structure. This required a separate simulation, which indicated that the structure needed to be strengthened.

In constructing the floating stages for the Bregenz Festival, there is obviously no opportunity for building prototypes or making design changes along the way. Since the opening date of the festival is set long in advance, unlike many building projects, the completion date for the stage cannot be changed. The safety of the singers, the stage crew and the audience depends upon getting the design right the very first time. The use of ANSYS Mechanical software, whose accuracy has been proven on a very wide range of analysis tasks, gave the entire project team confidence in the analysis results.

CADFEM, an ANSYS channel partner in Germany, supported Lener in his use of software from ANSYS.
Harnessing the Power of Ocean Waves

Engineers use structural and hydrodynamic analysis to ensure that wave-powered electrical generation machines produce maximum energy output and operate effectively for decades.

By George Smith, Managing Director, and Tamas Bodai, Analyst Engineer, Green Ocean Energy Ltd, Aberdeen, Scotland

With rising fuel costs and environmental concerns, governments around the world are focusing on clean, safe and sustainable alternative energy sources for power generation. One of the most unique and promising of these concepts is harnessing the energy of the earth's oceans by converting the relentless force of waves into electricity.

The idea has captured the imagination for centuries, but until now the business justification has not been sufficient to move such projects forward. Significant engineering hurdles must be overcome to develop efficient, reliable and economical wave-powered electrical generation systems that could be deployed on a mass-production basis. Green Ocean Energy is meeting these challenges with the help of simulation technology from ANSYS: ANSYS AQWA software for hydrodynamic analysis of the wave action and ANSYS DesignSpace software for structural analysis.

The engineering team uses this advanced technology in the development of the Ocean Treader, a floating device designed to be moored five kilometers offshore in open ocean water with relatively high wave activity. A second device called the Wave Treader mounts on the base of offshore structures such as wind turbines or tidal turbines.

Machines developed by Green Ocean Energy produce 500 KW of electricity from on-board generators powered by wave action that raises and lowers floating arms, which sit atop buoyant spars. The Ocean Treader (top) is moored to an anchor while the Wave Treader (bottom) mounts on the base of offshore structures such as wind turbines or tidal turbines.
turbines or tidal turbines. Both machines share a similar design, with two 20-meter steel arms floating on a set of sponsons (components that make the machine buoyant) made of glass-reinforced composite plastic. As wave action moves the floating arms up and down, hydraulic cylinders spin generators that produce electricity sent back to shore via underwater cables. Each machine is designed to produce 500 KW of electricity — enough to power 125 homes — so a farm of 30 such devices would have a rating of 15 MW.

One of the primary challenges the engineers faced was reaching a balance between structural strength and weight restrictions. With an expected 25-year design life, the machines must withstand rough waters of the North Atlantic, where waves can reach over 9 meters in height in gale-force winds. Conversely, structural members must be lightweight to keep production costs within budget and to allow for sufficient floatation.

Software from ANSYS played a key role in meeting these objectives. The design team used the ANSYS AQWA product to determine how the structure would respond to a particular wave action. First, the team created a hydrodynamic model of the submerged part of the structure based on the geometry of components together with their density and inertia. Next, they entered wave data profiles, including wave height and frequency, obtained from empirical measurements in the particular body of water.

From these inputs the ANSYS AQWA application generated a variety of hydrodynamic parameters including:

- Diffraction force accounting for the deformation of waves as they impact the structure
- Froude–Krylov force derived from the pressure field of waves against the structure
- Hydrodynamic damping due to radiation of waves induced by structure motions and the associated energy dissipation
- Added mass of the structure as a result of the surrounding water set in motion by the oscillating body
- Hydrostatic stiffness and buoyancy

Hydrodynamic parameters were entered into a proprietary code developed by Green Ocean Energy for computing the kinematic response and resulting power output of the machine. Customized plots of the power output for a range of sizes of the major components, such as the length of the arms and shape of the sponsons, enabled engineers to determine optimal design parameters for the major structural members.

The structural analysis model to compute stress distribution and deformation of components was efficiently achieved through tight integration with Autodesk® Inventor®, which enabled part geometry to be automatically transferred from CAD to ANSYS Workbench using the Geometry Interface for ANSYS DesignSpace structural analysis software was used to determine stress distribution in the Ocean Treader arm (top) and deformation of a spreader beam structure that lifts the machine safely into the water (bottom).
Inventor/MDT. The analytical meshing was greatly simplified through the use of surface-to-surface contact element features that automatically detect contact points of touching parts. The use of multiple parts allowed different material properties to be assigned — including the anisotropic nature of the glass-reinforced composite plastic parts.

After an initial simulation cycle was completed using ANSYS DesignSpace software, direct associativity with the CAD system enabled engineers to readily change the design and quickly perform subsequent simulations on the new part geometry without having to re-apply loads or boundary conditions. Green Ocean Energy engineers performed successive iterations to eliminate stress concentrations by adding or trimming material where needed.

ANSYS DesignSpace software was instrumental in minimizing the weight of the entire structure while helping to ensure that each part could withstand the range of expected wave forces over time. The technology from ANSYS was crucial in achieving the sensitive balance of mass, moment of inertia and center of gravity so that the floating arms would react optimally to wave action. Currently, engineers are using this procedure to develop Ocean Treader scale-model prototypes — which are undergoing wave-tank trials — and several organizations are expressing strong interest in both the Ocean Treader and Wave Treader machines. To fill future orders once tests verify power output and structural integrity, full-scale production models of the machines will be developed using these same tools from ANSYS, with detailed full analysis of final designs to be performed with ANSYS Mechanical software.

In this complex development process — in which so many variables must be considered — standard hydrodynamic calculations alone would be too slow and not detailed enough to provide sufficient insight into the behavior of the machines subjected to severe environmental conditions. Moreover, because prototypes cost over $3 million each and take months to construct, numerous rounds of hardware test-and-redesign cycles are impractical. To meet the rigorous technical requirements, product delivery deadlines and business objectives for both the Ocean Treader and Wave Treader machines, Green Ocean Energy finds the virtual prototyping capabilities of the advanced tools from ANSYS a critical element in getting the product to market in a cost-effective and timely manner.

ANSYS AQWA hydrodynamic software determines how the Wave Treader reacts to wave action by computing hydrodynamic parameters such as diffraction, Froude–Krylov force, radiation damping and added mass.
Designing for Strength, Speed and Luxury

Simulation software from ANSYS helps a yacht designer deliver the optimal combination of luxury and performance.

By Chad Caron, Naval Architect, Delta Marine Industries Inc., Washington, U.S.A.

Purchasers of 100-foot-plus megayachts have come to expect the ability to customize the interior design to a level that matches their wildest dreams. Award-winning yacht builder Delta Marine has become one of the world’s leading builders of megayachts—in part through expertise in designing carbon fiber structures that enable virtually any interior configuration while providing high levels of strength, durability and performance. However, giving interior designers the freedom to place walls or partitions wherever they wish creates structural design challenges by increasing the complexity of the load paths.

Graphite composites provide the ideal material for megayacht design because they are stiffer and stronger than metals per unit of weight, making it possible to build a lighter and stronger boat. Composites enable more flexible designs because their physical properties can be tailored to a very high degree. In the past, interior design was constrained by structural considerations; more recently, advancements in composites have provided designers with far more flexibility. Today, for example, the location of pillars can be more readily accommodated by structural engineers to suit the interior designers’ vision.

Traditional design methods, such as handbook formulas and rules of thumb, are not adequate to achieve an optimized structural assessment for these new types of interiors. This means that analysis typically needs to be performed on a global basis, which in turn requires very powerful software and hardware.

The horizontal structural elements in a megayacht are the decks. In Delta’s latest megayacht, the three decks are made of two-inch-thick composite sandwich construction. The vertical structural elements consist of free-standing pillars that are used to support beams and also vertical beams that are attached to the superstructure plating (the mullions). Delta selected ANSYS Mechanical technology as its structural design software.
software seven years ago because the yacht maker believed that the software’s composite analysis capabilities were well ahead of competitors. At the time ANSYS Mechanical was the only finite element software Delta could find with a composite shell element. As composites simulation technology has progressed, according to Delta, the ANSYS Mechanical package has maintained an advantage in composite design capabilities.

The Delta Marine team models the major shapes of the yacht in Rhinoceros®. The Rhinoceros model is exported to a neutral file format and imported into ANSYS Mechanical software to provide the geometry for the model. The naval architect uses composite shell elements to model the laminate stack layer by layer and uses solid elements for foundation parts that are cast in resin and in scantlings (frame solid elements for foundation parts that laminate stack layer by layer and uses the model. The naval architect uses ware to provide the geometry for imported into ANSYS Mechanical soft-

exported to a neutral file format and have a core that is structurally significant. The prediction of vessel vibration frequencies is dependent on the total weight distribution for the yacht. The interior design has the potential effect of increasing overall weight through the substantial use of hardwood and stone, especially common in today’s yachts. Delta has developed parametric approaches to estimate interior weights using targets on a per-square-foot basis for various materials. The outfit weight along with the other structural and mechanical weight components coupled with the hydrodynamic added mass of the water directly affect the vibration frequencies and mode shapes the yacht will exhibit. Accurately predicting these frequencies and mode shapes is critical to successful design.

Delta designed its new mega-yacht in two stages: first for strength and then to resist vibration. Today’s yacht buyers are interested in a luxurious interior and a high cruising speed, but it is critical to optimize the structural elements to deliver the required strength while avoiding any extra weight that would reduce the speed of the boat. The designer uses ANSYS Mechanical technology to evaluate global and local stresses on a layer-by-layer basis. Most other finite element analysis packages merely average the loads over the stack.

ANSYS Mechanical tells the designers exactly where the load is going, down to the individual composite layer. This simplifies the design of the mullions, beams and pillars. The ability to distribute the loads among the different layers also helps to tune the laminate stack. Delta uses ANSYS reports to detail and defend structural decisions that the regulatory body rule books cannot cover adequately.

Even after a structure has been designed to support the design loads, the yacht may still vibrate. The Delta designer performs modal analysis to investigate its primary modes of vibration using the same ANSYS Mechanical model. The technology determines the natural frequencies of the mass matrix. The analysis results in a recent project showed that the first mode of vibration was a racking mode, which meant that the superstructure of the boat vibrated horizontally, with the decks decoupling from each other like a deck of cards sliding back and forth.

The engineers addressed the concern by adding a racking frame, a structure that spans two decks and resists horizontal motion. Delta modeled the racking frame using 0/90 and +45/-45 biaxial laminate and unidirectional carbon fiber. The analysis results identified the stresses in the area that experiences the highest bending moment.

ANSYS Mechanical simulation makes it possible to determine exactly how loads distribute through this complex structure, so that engineers can tailor the properties of structural elements to provide strength and stiffness exactly where it is needed. These capabilities free the designers to put walls and partitions wherever they want and to keep the weight to a minimum level. As a result, the new boat delivers the optimal combination of luxury and performance. ■
Seeing the Future of Channel Design

NVIDIA uses VerifEye and QuickEye as an extension to traditional SPICE-level simulation approach to design high-performance graphics solutions.

By Chris Herrick, Technical Lead, Ansoft LLC

It’s hard to imagine, but there was once a day when slick high-resolution graphics were not the norm. With increasing monitor sizes as well as ever-sharper digital media and spectacular gaming technology, computer graphics have progressed in an amazing way. NVIDIA® is a world leader in visual computing technologies and the inventor of the GPU, a high-performance processor that generates breathtaking, interactive graphics on workstations, personal computers, game consoles and mobile devices. NVIDIA serves the entertainment and consumer market with its GeForce® graphics products, the professional design and visualization market with its Quadro® graphics products, and the high-performance computing market with its Tesla™ computing solutions products.

One of the hardest challenges when designing these high-performance graphics solutions is ensuring that the communication link is clear between the pixel generation and pixel display. That means the signal, representing a zero or one, originating at one part of the system needs to propagate undistorted to another area so it may be detected without errors.

As data link speeds increase, so do the problems that affect signal quality. Every part of the physical routing channel has some influence on the propagating electromagnetic fields and, thus, on the detected waveform. The channel could be assembled from many combinations of elements including packages, transmission lines, cables, connectors and vias. A discontinuity or impedance
mismatch to the propagating signal could occur at any point along the transmission path.

A common tool for the signal integrity engineer is circuit simulation. By modeling the channel virtually, engineers are able to predict waveforms not only at the receiver but for each section of their modeled channel. This level of detail allows engineers to verify signal detection as well as to determine the contribution of signal distortion for each section of the channel. To improve or optimize a system, the sections of channel that produce the greatest signal distortion can be identified, and intelligent changes can be made. These changes could include geometric variations, elimination or addition of components, or material selection.

In order to construct the virtual channel, NVIDIA chose the Ansoft Designer tool as its simulation environment. Ansoft Designer allows the engineer to assemble each piece of the channel as a black box model. These models may comprise measured data, simple circuits, SPICE components or dynamic links into any of Ansoft’s circuit extraction tools. These individual models may be rearranged, bypassed or parametrically varied, providing the engineer with the ability to test all possible configurations. This high-level schematic approach also allows the design to be easily shared among different groups, which then can quickly see what is being modeled and provide input into the design.

Under the hood of Ansoft Designer software lies the powerful circuit simulator product Nexxim. Nexxim technology is a high-capacity, high-accuracy engine for linear network analysis, transient analysis, harmonic balance, fast convolution and statistical methods. With this array of different simulators, NVIDIA is able to look at channel performance from many different perspectives, all from within the same environment.

The traditional signal integrity simulation methodology is to perform a transient simulation. Using this approach, the driver is toggled for a length of time, and the voltage is monitored at receiver. A common way of viewing this received data is to overlay the voltage versus time for each bit period, or unit interval. The resulting graph is called an eye diagram, and it can very clearly show whether the received data is able to be detected error free.

Transient analysis is the most accurate means of determining signal waveforms, but it is limited by the scope of the problem. To fully analyze all the variations in a channel with nonlinear devices can take days to weeks. Trying to achieve low bit error rates (BERs) poses another challenge. Using transient analysis, simulating enough bits to satisfy a BER of $10^{-12}$ could take years. In order to satisfy these engineering challenges, Nexxim technology has incorporated two specialized solvers, namely QuickEye and VerifEye.

According to Ting Ku, director of signal integrity at NVIDIA, “The obvious reason for statistical transition is related to simulation coverage. Given there is a finite amount of time and machine resources, the statistical approach gives engineers systematic coverage without

The top image demonstrates the receiver eye on a channel where the data cannot be recovered, the bottom is a clean eye diagram where the data is recoverable.
running an astronomical number of simulation corners. One other good benefit of the statistical approach is in dealing with design corner definition by projecting what the final production yield would be. Using the statistical methodology allows engineers to make judgment calls between cost and production yield."

While both QuickEye and VeriEye methods offer significant speedup over transient analysis, each offers a different solution to the problem at hand. QuickEye is a fast convolution-based method that allows the user to explicitly define a bit pattern that is sent and to view the resultant waveforms. VeriEye is a purely statistical-based approach that characterizes BER of a channel down to $10^{-16}$.

Both of these methods begin analysis the same way by first computing the transfer function of the channel. This computed channel response is assumed to be linear-time invariant. For QuickEye, the channel response is then convolved with a user-specified bit sequence to obtain a time vs. voltage waveform. For VeriEye, a cumulative distribution function is derived from the step response convolution-based method that allows the user to explicitly define a bit pattern that is sent and to view the resultant waveforms. VeriEye is a purely statistical-based approach that characterizes BER of a channel down to $10^{-16}$.

While it is critical to fully characterize the entire passive channel, the scope of the analysis does not stop there. To compensate for frequency-dependent effects of the channel, such as inter-symbol interference (ISI), NVIDIA uses silicon-based compensation. Additionally, there may be other influences on the signal in the form of jitter that must be accounted for. This jitter may be seen at both the driver and the receiver.

As part of its investigation into silicon-based channel compensation, NVIDIA can use either QuickEye or VeriEye to evaluate feed forward equalization (FFE) or decision feedback equalization. If the silicon has already been characterized, the number of equalization taps and their respective weights can be added to either the driver or receiver on the channel. During early stages of design, the Nexxim tool can be used to automatically calculate the ideal weights necessary to invert the effects of the channel on a bit stream.

Jitter characterization, and its inclusion in simulation, is another area critical to NVIDIA. Without including all sources of noise, accurate BER simulations would be impossible. Random jitter (RJ) and duty cycle distortion (DCD) can also be added to each driver. ANSYS staff learned from this partnership with NVIDIA that the inclusion of periodic jitter (PJ) and sinusoidal jitter (SJ) would be useful features, so ANSYS has since enhanced the Nexxim tool to include these features. Deterministic jitter (DJ), based on ISI, will inherently be modeled by the channel’s transfer function. At the receiver, the source jitter will accumulate with the DJ of the channel to create a new jitter distribution. This jitter in combination with the jitter defined at the receiver, either RJ or a user-defined distribution, will account for the total jitter (TJ) of the channel. Reducing TJ is the main objective when designing a channel for low BER.

With ever-increasing bit rates and channel complexity, the landscape of signal integrity analysis is changing drastically. Transient analysis can no longer be relied on as the sole means of channel simulation, especially when trying to achieve extremely low BER. This challenge has been met head on at NVIDIA by adopting QuickEye and VeriEye analysis into their design process.
Make your ANSYS applications fly...
Panasas high performance storage makes ANSYS software go faster!

Panasas and ANSYS take Computer Aided Engineering (CAE) to an elevated level:

- more than 2X computational efficiency and job turnaround times
- more than 2X additional license utilization for the same investment

innovate faster | innovate better | innovate at lower cost

Start flying now...download ANSYS FLUENT 12 Performance Study
www.panasas.com/jsys/ansys

Speedier Product Design with Sun

Accelerate simulations by up to 2x

Sun Business Ready High Performance Computing Solutions employ the latest technology and are tested in the lab to deliver:

> Up to twice as many simulations per day for a wide array of applications
> Significantly reduced power, cooling and space requirements compared to previous generation systems

Learn how your organization can benefit today at sun.com/hpcsolutions

© 2009 Sun Microsystems, Inc. All rights reserved. All logos and trademarks are property of their respective owners.
Hot Topics: High-Capacity Hard Disks

Samsung uses simulation to improve thermo-fluidic performance of hard disk drives.

By Haesung Kwon, Senior Staff Engineer, Samsung Information Systems America, California, U.S.A.

Recently, the capacity of hard disk drives (HDD) has reached the phenomenal level of more than one terabyte per single drive. Robust mechanical design played a key role in this achievement, since drive development challenges today are not related to just a single physics but to multiphysics. In the past, most mechanical-originated failure modes were identified using only a good understanding of HDD dynamics. But as the tracks on the disk become more tightly packed to achieve higher capacity for the drive, nanometer-level positioning of read and write elements is very important in response to the external and internal vibration of the HDD. These vibrations are often caused by air flow and heat transfer.

To achieve nanometer-level precision, faster seek and access time is needed. This requires higher current, which, in turn, leads to temperature rise in the voice coil motor (or actuator), usually simply called the coil. The coil moves the actuator arm holding the read and write heads, and the arms, heads and coil together are called the head stack assembly (HSA). Temperature rise in the coil can cause undesirable mechanical performance. This temperature rise is strongly dependent on the location of the HSA, and convective heat transfer can affect the temperature rise when the HSA is in different positions. Using both ANSYS CFX and ANSYS Mechanical software, engineers at Samsung made an engineering discovery that allowed them to improve thermo-fluidic performance of the HDD. The finding also provided insight into the design of high-performance HDDs.

As always in simulation-driven product design, simulation during the early stage of HDD development is an important contributor to a successful time to market. The range of simulation available for the HDD industry includes both basic and advanced features of ANSYS Mechanical and ANSYS CFX software. Samsung uses software from ANSYS because of its expandability to multiphysics capabilities. For instance, flow-induced vibration has been used to understand and predict the HSA’s dynamic...
performance for different configurations. The model can be extended in the future to run thermally induced vibration or shock in a systematic manner with relatively little effort.

The simulation models for the HDD contain solid and fluid regions. Samsung engineers used the fluid region to solve for flow and heat transfer. They used the solid domain to determine heat transfer only and, in this study, without considering radiative heat transfer. Temperature rise in the solid regions, particularly the coil and actuator arms, is of great interest because these regions are strongly tied to the reliability of the entire drive.

Using ANSYS CFX capabilities, the Samsung team calculated temperature, velocity and pressure with two different models: with the HSA positioned at the inner radius of disk (ID) and at the outer radius of disk (OD). Intuitively, the engineers knew that different outcomes for flow-induced vibration would occur at each location. However, the team had seldom explored simulations of temperature discrepancies because of the large model required to encompass the two different physics — flow and heat transfer.

The model contained five arms and four disks. The fluids simulation determined that a high flow rate passes through the coil when the HSA arm is positioned in the ID. This creates a high convection coefficient on the surface of the voice coil motor, resulting in a relatively low temperature rise. The arms actually block air flow from passing to other areas of the disk, and the air flow tends to move toward the voice coil. However, with the arm on the OD, a path for air to flow in the circumferential direction is opened up, so that less air flow travels through the coil and a low convective heat transfer coefficient is generated. This observation is the critical reason that temperature rise is quite different in the two cases.

Temperature rise within different parts of the HSA were also investigated using structural mechanics simulation. The bobbin has the highest temperature swing compared to other parts since it is located closest to the coil heat source. The temperature rise for all five arms is almost the same. Samsung engineers performed an analysis to determine if there was uniformity of convective heat transfer from each part compared to the total convective heat transfer. They determined that more heat is transferred through the inner arms located between two disks. The results reveal that the heat transfer path of conduction and the convection modes are highly dependent upon the location of the HSA. This understanding is critical to the thermal packaging design of the HDD.
Fatigued by Stress Limitations

The combination of fe-safe and ANSYS software helps Cummins improve life prediction accuracy.

By Jeff Jones, Technical Advisor, Cummins Inc., Indiana, U.S.A.

In developing cutting-edge design solutions, diesel engine manufacturer Cummins Inc. uses a deterministic approach for predicting product life, one that considers complex materials and loading. Its current solution incorporates technology from two proven leaders — but the path to this approach was not a straight line.

A recognized technology leader in the global diesel engine market, Cummins faces increasingly stringent design requirements as it develops cutting-edge solutions. The company’s roots are planted in soil nourished by innovation. For example, the firm was among the first to see the commercial potential of diesel engine technology. Even before the advent of commercial software tools, Cummins’ engineers developed internal software for thermal, structural and design applications to ensure that its engine designs were cost effective, reliable and durable. Today, Cummins is no longer just an engine business but a global power leader with more than $11 billion (U.S.) in annual sales.

In the late 1970s, Cummins continued its pioneering efforts, becoming one of the first companies to embrace commercial tools for finite element analysis. It standardized on the mechanical analysis solver from ANSYS because of the technology’s flexibility and performance. The relationship between the two companies has expanded since then: Cummins has been an active member of the ANSYS Advisory Board for more than a decade.

However, there was reluctance at Cummins to replace its internally developed fatigue analysis software because of rigorous internal requirements for depth and range of fatigue theories along with the need to handle proprietary materials and loads. In 2002, the company turned to Safe Technology Limited, which offered fe-safe™ for fatigue and durability analysis. The partnership that existed between ANSYS and Safe Technology ensured efficient and effective interfacing between fe-safe and simulation tools from ANSYS.

To verify that fe-safe offered accurate life prediction capability, Cummins executed a sophisticated test plan to compare fe-safe results to internal fatigue analysis software. The test plan included four finite element models:

- Simple 2-D plane stress uniaxial model
- Moderate 3-D biaxial stress model
- Fully featured engine block
- Fully featured engine head

Cummins engineers subjected each model to a number of different and appropriate loading scenarios. By using a range of models, it was possible to gain fundamental insights into the technology and to compare predictions against field data.

The baseline internal fatigue software was based on an advanced Goodman approach: one that is stress-based, in which damage prediction is based on stresses.
Stress-based fatigue analyses are severely limited when it comes to low-cycle fatigue problems. Low-cycle fatigue typically considers approximately $10^5$ duty cycles, while high-cycle fatigue is appropriate for more than about $10^9$ cycles. Cummins needed a unified approach for predicting product life for gray iron components that would be viable for both low-cycle and high-cycle fatigue. The company also wanted to determine if the strain-based methods employed by fe-safe, such as the Smith–Watson–Topper (SWT) algorithm with Neuber correction, were more suitable for meeting this low- and high-cycle requirement for gray cast-iron fatigue prediction.

For load cases dominated by tensile stresses, the Goodman-based internal software provided results that were consistent with the SWT approach. However, in the biaxial case dominated by compressive stress, the internal software predicted much more damage than fe-safe, implying that a stress-based approach in this case may result in an overly conservative design.

Complex real-world models of a cylinder block and head, considering standard proprietary loading conditions, produced more noticeable differences. Stress situations for two complex load cases were compared at more than 20 locations for which considerable test experience existed. In nearly all cases, fe-safe results were in line with expectations, while the Goodman-based approach predicted less damage at several locations (thus over-predicting product life). A closer review at three critical locations revealed that for cases with high mean, low alternating stresses, fe-safe provided results that agreed very well with test and field experience.

Cummins engineers made the following observations when considering the test results:

- While internal software over-predicted product life in some cases and under-predicted it in others, fe-safe results used in conjunction with the mechanical analysis solver from ANSYS correlated very well with Cummins’ industry experience.
- Even with modifications, older stress-based approaches for predicting fatigue have limitations in comparison with modern strain-based methods.
- Use of the mechanical solver from ANSYS in combination with fe-safe offers opportunities to further increase reliability and reduce costs.
- A better understanding of fatigue facilitates design innovation.

A noted contributing factor to the successful outcome of the testing was the tight integration between fe-safe and ANSYS Mechanical software. Using fe-safe, the ANSYS results (.rst) file is read, material properties and fatigue cycle (combinations of load steps) are specified, and the fatigue damage is calculated and written back to an ANSYS results file for display in ANSYS Mechanical software. Fatigue results may be plotted as contours of log-life (log-cycles to failure) or factors of strength (such as design margin). Another important factor was the ability to use comprehensive and user-configurable libraries, facilitating use of internal proprietary materials data with minimal effort.

With the development of the integrative ANSYS Workbench platform, all structural analysis and flow modeling tools at Cummins are being brought into one environment, further enhancing productivity.

Today at Cummins, nearly every engine component is analyzed using the ANSYS Mechanical product. fe-safe is used to perform fatigue analysis for many components, such as cylinder blocks, cylinder heads, pistons, connecting rods and main bearing caps. In engine cylinder heads with high assembly stresses, significant compressive stresses, and peculiar behaviors of gray cast iron, fe-safe software plays a vital role in helping to develop reliable, cost-effective designs. Advanced fatigue analysis using fe-safe with ANSYS Mechanical software helps to get the design right the first time, and it reduces development costs.

**Reference**

www.safetechnology.com
Managing Heat with Multiphysics

Multiphysics simulation helps a global company design better electrical products.

By Arunvel Thangamani and Sanjeeva Reddy, Schneider Electric, R&D, Global Technology Centre, Bangalore, India

Multiphysics simulation is used in electrical industries to predict product performance and failure conditions and to perform optimization. Product testing is very costly, and repeated trials are not part of the preferred product development process, wherein products are optimized early in the design process using simulation. In addition, simulation assists product designers in these industries to meet standards required by bodies such as Underwriters Laboratories (UL) and the International Electrotechnical Commission. Thermoelectric simulations play a vital role in product development in product areas such as final distribution enclosures, industrial plugs and sockets, protection and control of low-voltage power circuits, electrical network management, energy-efficiency devices, automation and control devices, power electronics cooling, and millivolt switching devices.

Schneider Electric is a global specialist in energy management, with operations in more than 100 countries. The company focuses on making energy safe, reliable and efficient. The organization’s Global Technology Centre (GTC) in Bangalore, India, has 460 employees working on product development and resource enhancement, and its resulting innovative products and technologies are available in markets across the globe. To reduce costs and gain time in their product development process, the GTC uses ANSYS Icepak, ANSYS Workbench, ANSYS Multiphysics and ANSYS FLUENT technologies. Researchers there used ANSYS Multiphysics to perform thermo-electric simulations on two Schneider Electric product assemblies — a circuit breaker terminal and an automatic transfer switch — to determine the temperature generated due to Joule heating as well as to define conductor and insulator specifications to effectively manage heat. The study extended into analyzing the effects of convective cooling by varying the convective film coefficient, and the results from the ANSYS Multiphysics simulations were compared with test results.

Miniature circuit breaker (MCB) terminals are subassemblies of the Schneider compact circuit breaker series. The terminal connects the circuit breaking device with external circuitry. An automatic transfer switch is an electromechanical switching device used widely in Schneider Electric low-voltage products. For both the terminal and switch, the CAD model was developed in Pro/ENGINEER® and imported directly into the ANSYS Multiphysics environment. Engineers meshed the geometry with direct coupled-field elements, reduced costs and gain time in their product development process,
which automatically account for the bi-directional coupling between electric Joule heating and temperature in either steady-state or transient analyses. These elements accept both thermal and electric boundary conditions and excitation. Electric boundary conditions were used to prescribe the net DC current passing through individual solid conductors. Thermal boundary conditions consisted of thermal convection coefficients defined on surfaces exposed to 25-degree Celsius (C) air.

The team modeled two MCB terminals: a 63-amp terminal and a 25-amp terminal. The design for the 63-amp terminal was modified in thickness to reduce the temperature by 9 degrees C. Correlations between simulation and experimental lab results were impressive, with only a plus or minus 2-degree C deviation. For the 25-amp terminal, temperature results confirmed the safety of the product and also corresponded well with experimental results.

Similarly, using simulation, design improvements in insulation material were applied to the transfer switch terminal model that resulted in a temperature reduction of 6 degrees C in the assembly. Once again, the simulation results compared well with the experimental results, with a plus or minus 4-degree C deviation.

The R&D experts then studied the current flow path and the variation of the convection coefficient with the maximum temperatures obtained. These simulation results gave researchers confidence that the same modeling approach could be applied to all products in these families. Electrical conductor thickness and insulation material selection can be optimized using simulation to reduce the need for prototyping at the beginning of the design cycle and to save valuable development time and costs.

“The Global Technology Centre uses multiphysics technologies from ANSYS to reduce costs and gain time in their product development process.”
Up, Up and Away

Simulation-driven innovation delivers a new ejection seat design for a military aircraft in less than 14 months.

By Park O. Cover, Jr., Senior Mechanical Engineer, Concurrent Technologies Corporation, Pennsylvania, U.S.A.

The military’s advanced-concept ejection seat, ACES II®, is one of the most successful aircrew escape systems in U.S. Air Force history and is credited with saving more than 450 lives since it was introduced in 1976. With more than 8,000 seats delivered to date, the ACES II is currently used on F-15, F-16, B-1B, B-2, A-10, F-117 and F-22 aircraft. Using the strengths of the ACES II as a foundation, Goodrich Aircraft Interiors and Concurrent Technologies Corporation (CTC), both in the United States, developed the next-generation ACES S seat for the F-35 Joint Strike Fighter (JSF). The new seat was optimized to enhance safety for aircrew, to reduce maintenance downtime, to reduce weight and to integrate with the F-35 cockpit. However, the biggest challenge was developing and delivering a brand new seat structure in less than 14 months.

The parametric link between the ANSYS Workbench platform and Pro/ENGINEER® Wildfire® software was a critical factor in successfully developing a design that met all the requirements while maintaining the aggressive schedule. Engineers at CTC were able to quickly update simulations for multiple design iterations. This concurrent design and analysis approach enabled the team to optimize the seat for both function and weight from the earliest developmental stage.

Analysis of the seat was split into three phases. The first analysis phase was conceptual design development. During this time, engineers designed the seat structure to meet functional requirements, while simulation was used to verify that the structure was sound and weight was optimized. Functional, structural and safety requirements were derived from the performance-based specification supplied by aircraft manufacturer Lockheed Martin for the JSF ejection seat. To reduce maintenance downtime, a modular seat structure was developed to allow the seat to be easily removed from the aircraft. The modular seat consists of the seat back, seat bucket, parachute, survival kit and aircraft interface module. Assembly costs and part count were reduced by designing the new seat to use a few machined components instead of many sheet metal components.

Engineers evaluated designs for tough load requirements, such as ejection from an aircraft traveling at 750 mph, parachute load and crash loads.

The first simulation phase evaluated individual components of the preliminary seat design. Equivalent stress plots of various stages of the bucket design evolution demonstrated how, during ejection, the occupant’s legs are forced apart by the windblast. The structure had to be optimized to contain this splitting force, or else the
AEROSPACE

The engineering team analyzed the structure for ejection and crash loads within the ANSYS Workbench framework. Once the simulation was set up, design iterations were quickly evaluated for all the applicable load cases simply by updating the geometry from the CAD system.

During the second analysis phase, the CTC team built a system model of the seat structure. Analyzing the seat structure as a whole gave the most representative view of how the actual seat structure would behave and eliminated compromises associated with analyzing individual seat subsystems or modules. To prepare the system model for analysis, the team imported the CAD geometry into the ANSYS DesignModeler tool where defeaturing operations, such as elimination of rivet holes, were performed. In addition, a few components were converted to mid-plane surface models using the software’s automatic mid-plane feature.

The CTC team assigned material properties, defined boundary conditions and applied loads to the system model. Contact regions were characterized for each riveted face on the seat. This allowed contact reaction forces to be used to determine the number of rivets required at each joint. Point masses were used to represent nonstructural seat subsystems, such as the parachute and survival kits.

The model was meshed using a hex-dominant mesh control and a 0.125-inch global element size. A single linear static structural analysis of the seat model was solved in less than 30 minutes using the direct solver within the mechanical software available through the ANSYS Workbench platform. The quick analysis turnaround time allowed the engineering team to quickly evaluate various what-if design scenarios.

Actual loads on the seat are very dynamic in nature, and the seat will experience these loads only one time during deployment. Because the system model used a static simulation approach without nonlinear material properties, the simulation revealed small areas of stress concentration that exceeded the allowable ultimate strength of the material. Engineers scrutinized these high-stress zones using submodels that allowed material yielding during the third phase of the analysis.

To produce the submodel, the team first cut out the area of interest using the ANSYS DesignModeler tool. The CTC team developed a submodeling subroutine using the commands object in the mechanical simulation area of ANSYS Workbench. The subroutine interpolated the system model displacements onto the submodels’ cut boundaries. The submodel results typically showed that some permanent deformation occurred, but the ultimate strength of the material was not exceeded. Furthermore, the submodel provided more accurate stress results due to the finer mesh. Roughly 30 high-stress areas were evaluated using this technique to ensure that the structure would not fail when loaded in extreme conditions. These results proved that the ultimate load requirements were met.

After 10 months of development, five prototype seats were built for test purposes, and the first ejection test of the ACES 5 F-35 JSF seat occurred after 14 months. The seat performed flawlessly the first time out. This extraordinary outcome is the result of a great deal of teamwork between Goodrich and CTC and would have been unattainable without using engineering simulation software.
Rolls-Royce is a name associated worldwide with quality — and not limited just to automobiles. Through subsidiary Rolls-Royce Marine, its equipment is installed on 20,000 commercial and naval vessels around the world, and its comprehensive range of products includes gas turbine and diesel engines, nuclear propulsion systems, steering gears, stabilizers, thrusters, water jets, winches, cranes, rudders and main propeller systems. The company is a key part of the Rolls-Royce Group, with 7,000 employees serving 2,000 customers. The Rolls-Royce Hydrodynamic Research Center in Sweden is Rolls-Royce Marine’s center of excellence in hydrodynamic propeller and waterjet design research. The center has combined the best of computational fluid dynamics (CFD) simulation and physical experiments to help the company to develop the Kamewa CP-A, its latest controllable-pitch propeller (CPP).

A CPP is a special type of propeller with blades that can be rotated around their long axis to change their pitch. Changing the pitch makes it possible to provide high levels of efficiency and maneuverability for any speed and load condition. Stopping distance can be cut in half compared with a conventional fixed-pitch propeller. Traditionally, propeller development has been driven by a combination of physical experiments and potential flow analyses. Physical experiments have the advantage of being grounded in physical reality but involve the expense and time of building and testing a prototype. In addition, potential flow analysis is restricted in that it does not account for the full geometry of the propeller. CFD, on the other hand, can incorporate the full geometrical complexity of a propeller’s operation. CFD can also provide much more detailed results than either physical experiments or potential flow analysis, such as flow velocities and pressure at every point in the problem domain, as well as the inclusion of viscous effects. The challenge at Rolls-Royce Marine was to incorporate the full complexity of flow around the propeller into the CFD model in order to accurately match physical experiments.

Engineers developed the CFD models using a hexcore volume mesh generated using TGrid pre-processing software from ANSYS. The hexcore mesh, which maintains cell surfaces perpendicular to the main flow in the core fluid region, kept the number of cells to a reasonable level. Rolls-Royce Marine used the full multi-grid initialization method together with the pressure-based coupled solver in ANSYS FLUENT software, which has proven to be both robust and fast for many applications. The engineering team simulated turbulence using the renormalization group (RNG) k-ε model,
since it was considered to be stable and, thus, a conservative approach.

The engineering team began by analyzing propeller calculations for open-water operating conditions. These calculations considered the operation of the propeller in a uniform flow field without looking at the influence of the ship’s hull. They then used the rotating reference frame method to simulate the rotating propeller. With this method, the team solved the flow equations in the rotating frame of the propeller blade. Integration of the calculated pressures and shear stresses on the propeller blades yielded thrust and torque, and the propeller’s efficiency was then calculated using these values.

Design development then moved into a detailed study of the interaction between the propeller and ship appendages. The Rolls-Royce Marine team simulated the complete ship hull in order to calculate the effect of the wake field on the propeller design. Engineers used a sliding mesh model to simulate the operation of the propeller in the flow field under the influence of the ship hull. The sliding mesh model is a transient approach that calculates the flow field as one grid region rotates (or translates) relative to another. Historically, Rolls-Royce Marine designed the propeller as the last step in designing the ship, so there was no opportunity to improve the hull design to optimize the propulsion system. The ability to simulate the interaction of the propeller and hull has now made it possible to address such a concern.

CFD simulations allowed Rolls-Royce Marine to evaluate a wide range of alternative hub geometries. The simulations also helped the engineering team reach a higher level of knowledge by providing far more information than physical tests could. For example, CFD made it feasible to easily determine the boundary layer in any prospective design. Generally, as the boundary layer gets thinner, the design becomes more efficient. By performing simulations of a number of different designs quickly, the team concluded that they could reduce the boundary layer and improve efficiency by modifying the hub contour.

Rolls-Royce Marine engineers were next concerned about the possibility of cavitation on the propeller hub caused by the boat’s wake. Cavitation is the formation of vapor cavities in a liquid due to a localized reduction in fluid pressure below certain critical values. The vapor cavities collapse violently as they move to regions of higher pressure and generate pressure waves that cause localized stress on and damage to nearby components. ANSYS FLUENT results identified low-pressure areas in which cavitation could occur on the Kamewa CP-A hub. Changing the geometry increased the pressure above the critical level and eliminated cavitation, which made it possible to increase the load at the blade root to further improve efficiency.

According to a 2003 study from the University of Delaware, international commercial and military shipping fleets consume approximately 289 million metric tons of petroleum per year, which is more than twice the consumption of the entire population of Germany[1]. The ANSYS FLUENT simulations run on the modified propeller geometry predicted that the efficiency would increase by 1 percent to 1.5 percent, and physical experiments confirmed that this was, in fact, the case. This seemingly small improvement, however, has the potential to reduce fuel costs by several billion dollars if applied across the board to the world’s commercial shipping fleets. It also has the opportunity to significantly reduce energy consumption and emissions of greenhouse gases.

Reference
Staying Cool with ANSYS Icepak

Thermal management solution predicts air flow and heat transfer in electronic designs so engineers can protect heat-sensitive components.

By Stephen Scampoli, Lead Product Manager, ANSYS, Inc.

ANSYS Icepak technology is aimed at one of the most significant challenges facing engineers designing electronic assemblies: dissipating thermal energy from electronic components to prevent premature component failure due to overheating. This fully interactive software is used to evaluate the thermal management of electronic systems in a wide range of applications, including simulation of air flow in enclosures, analysis of temperature distributions in chip and board-level packages, and detailed thermal modeling of complex systems such as telecommunications equipment and consumer electronics. By predicting air flow and heat transfer at the component, board or system level, the software improves design performance, reduces the need for physical prototypes and shortens time to market in the highly competitive electronics industry.

Based on powerful computational fluid dynamics (CFD) simulation, ANSYS Icepak technology has a specialized user interface that speaks the language of electronics design engineers. Models are created by simply dragging and dropping icons of familiar predefined elements including cabinets, fans, circuit boards, racks, vents, openings, plates, walls, ducts, heat sources, resistances and heat sinks. These “smart objects” capture geometric information, material properties and boundary conditions — all of which can be fully parametric so a user can easily enter values to precisely match application requirements or to study what-if scenarios. The software also includes extensive libraries for standard materials, packages and electronic components such as fans — including fan geometry and operating curves.

As a further modeling aid, the software can import both electronic CAD (ECAD) and mechanical CAD (MCAD) data from a variety of sources. Geometry from ECAD and MCAD data sources can be combined with smart objects to quickly and efficiently create models of electronic assemblies. For instance, a system analysis results of a fan-cooled processor heat sink attached to a printed circuit board.

ANSYS Icepak software predicts the temperature profile in a computer graphics card.
model of a computer enclosure could easily be generated by combining MCAD data for the enclosure, ECAD data for the printed circuit boards (PCBs) and electronic packages, and smart objects for other components. In addition, the ANSYS Icepak solution includes many macros to automate the creation of geometry, including different types of packages, heat sinks, thermoelectric coolers and industry-standard test configurations.

Another productivity feature is the ability to automatically generate highly accurate body-conformal meshes that represent the true shape of components rather than a rough stair-step approximation. Meshing algorithms can generate both multi-block and unstructured hex-dominant meshes. Algorithms also distribute the mesh appropriately to resolve the fluid boundary layer. While the meshing process is fully automated, users can customize the meshing parameters to refine the mesh and optimize the trade-off between computational cost and solution accuracy. By grouping objects into assemblies, the mesh count can be further optimized by meshing each assembly separately and automatically combining them before running the solution. This meshing flexibility results in the fastest solution times possible without compromising accuracy.

ANSYS Icepak uses the state-of-the-art ANSYS FLUENT computational fluid dynamics solver for the thermal and fluid flow calculations. The CFD solver solves the fluid flow and includes all modes of heat transfer — conduction, convection and radiation — for both steady-state and transient thermal-flow simulations. The solver also provides complete mesh flexibility, and this allows the user to solve even the most complex electronic assemblies using unstructured meshes, providing robust and extremely fast solution times.

Once the solution is complete, ANSYS Icepak software provides a number of different methods for visualizing and interpreting results. Visualization of velocity vectors, temperature contours, fluid particle traces, iso-surfaces, cut-planes and two-dimensional XY plots of results data are all available. The software also offers customized reports that allow users to identify trends in the simulation along with the ability to report fan and blower operating points. Reports including images can be created in HTML format for distributing the results data.

ANSYS Icepak tools can interface with other products in the software portfolio from ANSYS to allow comprehensive multiphysics simulation of electronic components. One option is the ability to import a power distribution map from SIwave; this simulation software from Ansoft extracts frequency-dependent electronic circuit models of signal and power distribution networks from device layout databases for modeling integrated circuit packages and printed circuit boards. Based on the results from an SIwave simulation, users can import the DC power distribution profile of printed circuit board layers into ANSYS Icepak software for a thermal analysis of the board. The coupling between the two packages allows users to predict both internal temperatures and accurate component junction temperatures for printed circuit boards and packages.

ANSYS Icepak software can export temperature data from a thermal simulation to a structural mechanics model to calculate thermal stresses of electronic components. With the demands of today’s high-performance electronic devices, electronic components are becoming more complex and using more exotic materials. These newer materials have widely varying thermal and mechanical properties and are being subjected to higher temperatures during both manufacturing and usage. These varying material properties and temperatures can result in significant thermal stresses, which can bring about fatigue-based failure of the components. ANSYS Icepak software, together with ANSYS Mechanical technology, allows users to evaluate both the thermal and mechanical aspects of the design.

ANSYS Icepak technology in conjunction with SIwave and ANSYS Mechanical products provides a full portfolio of software to meet the simulation requirements of the electronics design engineer. ANSYS continues to be a leader in providing solutions to the electronics industry — solutions that provide the high-fidelity electrical, thermal and structural simulations required to meet the challenges of today’s product development demands.
Analyzing Vibration with Acoustic-Structural Coupling

FSI techniques using acoustic elements efficiently compute natural frequencies, harmonic response and other vibration effects in structures containing fluids.

By Marold Moosrainer, Head of Consulting, CADFEM GmbH, Munich, Germany

When designing equipment such as vessels, tanks, agitators, hydraulic piping systems, hydraulic turbines, transformers and sensors, engineers often must take into account a contained fluid. Presence of such fluids may add mass, stiffness and damping, which change the structural mechanics of the system. Also, the fluid may act as an excitation mechanism such as occurs in water hammer, which is the shock wave that occurs in a piping system when a water valve is abruptly shut off.

To fully study structural vibration in these types of applications, engineers must model the coupling mechanisms for fluid structure interaction (FSI). For these detailed studies, software from ANSYS has an outstanding breadth and depth of capabilities for structural and fluid analysis. Models are getting more and more realistic, and FSI continues to be one of the largest segments of multiphysics simulation. FSI simulations are usually performed using the ANSYS multi-field solver, which employs implicit sequential coupling to calculate interactions between fluid and structural solutions.

As an alternative to these types of FSI analyses of fluid-filled structures, engineers may want to consider an approach based on the use of ANSYS FLUID30 elements available in the ANSYS Mechanical and ANSYS Advantage print 10/12/09 3:10 PM Page 40

How Fluids Influence Structural Vibration

The presence of a fluid can significantly change the vibration characteristics of the containing structure. To determine the extent of this effect, engineers must model all relevant dynamics, especially the fluid-structure coupling that represents the interaction between these two domains. Models based on the ANSYS FLUID30 element must, therefore, account for factors such as mass, stiffness and damping, that the fluid adds to the overall system.

Mass. The additional mass of a heavy, rather incompressible fluid such as water in a vessel usually must be included in the analysis model, such as the thin-walled box in the example. Note that the first vibration mode for the partially filled container is about a third lower than that of the dry container. There is no easy rule of thumb in analytically determining the required added mass because results depend so much on frequency and mode shape. A fully coupled FSI simulation is required to answer this question.

The influence of the additional mass of water on vibration modes of a rectangular tank is shown in models of a partially filled tank (left), an uncoupled structural dry mode of the empty tank at 33 Hz (middle), and a coupled wet mode at 10 Hz taking into account FSI using ANSYS SHELL181/FLUID30 coupling (right).
Stiffness. Lightweight, rather compressible fluids such as gases do not add appreciable mass, but they can add stiffness to a closed air-filled structure. To imagine the “stiffness of air,” think of an air spring (gas shock absorber) or a bicycle air pump that you close with your thumb. In the analysis of a rectangular piston, for example, the added stiffness of air included in the model doubled the natural frequency, from 40 Hz to 80 Hz. Further, instead of one mode, analysis of the air-filled cylinder indicated many resonances from acoustic cavity.

Damping. In an unbounded fluid domain (ANSYS FLUID30 combined with FLUID130 for the external absorbent boundary layer), structural vibration may lead to pressure waves that propagate through the entire fluid system. In these cases, the energy spent on these compressive longitudinal acoustic waves is dissipated in an effect known as “radiation damping.” Particularly large plate-like structures in heavy fluids may encounter considerable added damping that must not be neglected, but no fluid viscosity is required for analyzing such cases.

Multiphysics products. These elements have their origin in acoustic applications; typically, they are used for simulating sound radiation. Their elasto-acoustic and hydro-elastic capabilities, however, are very helpful in solving FSI vibration problems by providing straightforward fluid-structural coupling in a given range of vibration analyses in which:

- The fluid is quiescent or at least moderately slow.
- Vibration amplitudes are small (linear theory).
- The influence of fluid viscosity or shear layers is negligible, meaning an ideal gas assumption.

ANSYS acoustic elements accomplish the required fluid-structural coupling because they have four degrees of freedom (DOF): one for the sound pressure and three optional displacement DOFs. Thus, a consistent matrix coupling is set up between structural and fluid elements in which strongly coupled physics cause no convergence or performance problems. Additionally, in analyzing sensor applications, for example, engineers can use ANSYS FLUID30 elements to attach the piezoelectric part of the multiphysics problem via matrix coupling. In this way, three strongly coupled physical domains can be solved simultaneously: piezoelectric, structural and fluid.

Coupling structural elements to acoustic elements in this manner allows for transient analysis and, even more important, for modal and harmonic analysis in the frequency domain. Consequently, for the latter, simulation of the desired stationary peak response within one single frequency step is performed very efficiently. This is conveniently done without having to account for lengthy initial transients (particularly for weakly damped structures) required in most time-domain solutions coupling structural and fluid domains by a load vector.
Why Shallow Containers Slosh

To avoid severe load instabilities, engineers often face strict design requirements to control sloshing of liquid in moving containers, such as tanker trucks or rockets. In these applications, designers usually insert interior baffle plates or similar structures to impede the flow of liquid. Other applications include harbor design or study of long-wavelength tsunami waves. In all these cases, simulation plays a key role in predicting sloshing and evaluating ways to solve the problem.

An example of sloshing is the carrying of a dog bowl full of water, in which the liquid has the tendency to slop from side to side and often spill. Simulation reveals that this behavior occurs because the first sloshing mode of the bowl is roughly at 2 Hz — the typical human step frequency that excites this undesired resonance. Repeating the analysis for a glass of water reveals a first sloshing mode of 4 Hz, which shows why water glasses are much less prone to spilling than bowls. In these simulations, the structural walls have been assumed to be rigid. Using the hydro-elastic coupling capabilities of software from ANSYS, however, engineers also can study sloshing in elastic vessels such as reactor containment structures. Note that sloshing analysis with ANSYS FLUID30 coupling is restricted to small amplitudes, and that full-fledged finite element and FSI analysis must be applied for simulating very large vibration amplitudes or fluid–surface motion.

Another advantage of using acoustic elements is their ability to quickly solve for fluid–pressure fluctuations. Since fluid and structural vibration systems have different temporal and spatial scales, properly discretizing wavelengths in time and space for these structural and fluid domains often requires extensive amounts of CPU time and resources. In contrast, acoustic elements account for pressure fluctuations and fluid properties much more efficiently when structural behavior such as resonant frequencies, mode shapes and peak vibration amplitudes must be calculated. Recent ANSYS solver improvements have significantly sped up this task, solving the resulting unsymmetric coupled system of equations in a fraction of the time previously required.

The hydraulic turbine study is an excellent example of a contained fluid application that can be analyzed quickly and easily by a structural engineer using ANSYS acoustic FLUID30 elements. When coupled to the housing and the rotor, the elements include two important features. First, they account for the vibrating mass of water that strongly affects the vibration frequencies and vibration deformation of the structure. Secondly, the elements allow the engineer to easily model the excitation mechanism: the fluctuating pressure field of the water induced by the rotating rotor. Analyzing this coupled system by modal and harmonic response analysis with acoustic elements is considerably easier than with conventional finite element and fluid dynamics FSI methods, which require extensive effort to perform with transient analysis in both domains.

Today, users can complete modal analysis of FLUID30-based coupled systems within hours, even for large assemblies with millions of DOFs that otherwise would take days to perform using conventional FSI methods. Solution speed for huge models can be further improved by using special component mode synthesis (CMS) methods, Krylov subspace–based order reduction methods and symmetric formulations of the originally unsymmetric FSI system matrix.

Golden retriever Alex assists in a demonstration of sloshing. Analysis indicates that the first sloshing mode is 1.6 Hz, the frequency at which the bowl is prone to resonance and spills its contents.
Tired of reacting-flow simulations that take too long?

Coupling detailed chemistry to your reacting flow simulations leads to better predictions and faster development times. But, it can be difficult to employ a detailed mechanism given the practicality of simulation run-time. CHEMKIN-CFD adds fast, accurate chemical kinetics to ANSYS FLUENT simulations. “The combination of ANSYS FLUENT and CHEMKIN-CFD allowed me to accurately model my CVD process using the full chemistry mechanism,” says Anthony Dipp, Process Manager of Tokyo Electron America.” So why wait? Download your FREE copy today at www.reactiondesign.com/CHEMKIN-CFD.html
Integrating CAE Tools: a Package Deal

Moldex3D and ANSYS Mechanical team up to simulate microchip encapsulation.

By Anthony Yang, Director, Technical Research Division, CoreTech System Co., Ltd., HsinChu, Taiwan

Integrated circuits (ICs), also called microchips, are at the core of the modern electronic device. Packaging — the final stage of IC fabrication — is the essential technology that distributes electrical signals from a silicon chip onto the printed circuit board and provides protection against environmental stresses. Customer demand for highly sophisticated and ever-smaller electronic products has made IC packaging a challenging procedure and, thus, critical to system performance. Moreover, this trend is driving the technology toward higher packaging densities with thinner and smaller profiles, which makes the encapsulation process much more complicated and unpredictable.

Despite recent trends in copper internal heat spreaders, a significant fraction of ICs are encapsulated by plastic, a process that is usually accomplished by transfer molding. A typical transfer molding begins with loading a leadframe into the mold, placing and preheating the preforms, and then closing the mold. The molten epoxy molding compound (EMC) is then forced into the mold system by the transfer ram. The packages are ejected once the packing and curing processes have been completed. Common defects in the transfer molding process arise from the improper selection of processing conditions, molding material, leadframe layout or mold design. These defects include short shot, air trap, wire sweep and paddle shift.

Among these defects, wire sweep and paddle shift in particular are caused by fluid structure interaction problems. Gold wires are common IC package components used for transferring the electronic signals and power between the die and the leadframe contacts. During the encapsulation process, the melted resin flow exerts drag force on wires, which causes them to deform. The amount of wire deformation, or sweep, during encapsulation is affected by the resin viscosity, wire mechanical properties, package dimension and process conditions. If wire deformation exceeds the limit, it can cause wire breakage or adjacent wires to touch, and the package will subsequently fail. The paddle, on the other hand, is the flat base of the leadframe and is used to support the chip. Usually, the paddle is connected to the leadframe by long and thin metal leads, which give the paddle system a quite flexible structure during mold filling. Uneven resin flow in the cavity, however, can apply unbalanced loading on the paddle system and, hence, deform or shift the paddle. When the amount of paddle shift is too large, it will significantly reduce the thickness of the cavity, further resulting in excessive wire sweep or exposing the die.

Applying the conventional trial-and-error method to resolve these problems is difficult and costly because of the complex interactions among fluid flow, heat transfer, structural deformation and polymerization of the EMC. However, through the coupling of Moldex3D® from Coretech System Co. and ANSYS Simulation products, the coupling of Moldex3D® and ANSYS Simulation products, the mass transfer, heat transfer, and structural deformation can be coupled to produce a comprehensive simulation model.
ANSYS Mechanical simulation software, engineers can analyze the complicated physical phenomena inherent in the encapsulation process and further optimize the package design. The nonlinear capability of ANSYS Mechanical technology is critical for wire sweep and paddle shift simulations, since the deformation of wire or paddle could be quite large.

The IC Package module of Moldex3D is a fully integrated analysis environment connecting pre-processing, post-processing, mold filling and structural analyses for microchip encapsulation simulation. One of the major challenges of three-dimensional modeling of microchip encapsulation is generating a suitable mesh for analysis. In the Moldex3D pre-processor, an efficient volume mesh generator with high-quality and arbitrary grid type (for example, tet, hex, wedge, pyramid or boundary layer elements) allows meshing of the package geometry with minimal model simplification. Furthermore, a parametric capability can assist in creating wires, thereby greatly reducing meshing effort. In the mold-filling stage, Moldex3D can calculate the resin flow considering non-linearities such as viscosity change and the curing reaction of the EMC. Engineers can thus predict how the mold will fill, identify potential air traps and weld lines positions, and evaluate the runner and gate design.

Having obtained the local flow field from the mold-filling analysis, the drag force distribution can be calculated along each wire. With a single click in Moldex3D, the drag force, boundary conditions and relevant mesh data are then exported to the ANSYS Mechanical solver, which performs the deformation calculation transparently in the background for each wire at the instant when the melt touches it. After the analysis in ANSYS Mechanical is completed, the wire deformation result can be checked either in Moldex3D or in ANSYS post-processors. As to the paddle shift problem, Moldex3D outputs the spatially and time-varying pressure loads on the leadframe directly to ANSYS Mechanical, which then performs steady-state simulations based on the data for each instant of time. For the wire formation calculation, both Moldex3D and ANSYS Mechanical can display the paddle shift results.

The combination of Moldex3D and ANSYS Mechanical software provides a promising simulation solution for the microchip encapsulation process. By using the integrated analysis, molding defects can be easily detected and moldability problems can be improved efficiently to reduce manufacturing cost and design cycle time. These reliable and powerful simulation tools can help IC package designers to meet the difficult demands from legacy devices to tomorrow’s innovative packaging solutions.
Muscle In A Box...

Harness the power of a high performance cluster - in a box. The Cray CX1™ deskide supercomputer is affordably priced to meet the needs of individuals and workgroups who require the power of a high performance cluster that is "ready to go" with the ease of use of a workstation. Available with Windows® HPC Server 2008 or Red Hat® Enterprise, and coupled with engineering simulation software from ANSYS, the Cray CX1 personal supercomputer enables higher-fidelity simulations and fast turnaround!

To download benchmark results and supporting data, visit:
www.cray.com/muscle

Cray CX1
We Take Supercomputing Personally

©2009 CRAY INC.  CRAY, the CRAY logo, and Cray CX1 are trademarks of Cray Inc. in the United States and other countries. Intel and Intel Xeon are trademarks or registered trademarks of Intel Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are trademarks of Microsoft Corporation in the United States and other countries. All other trademarks are the property of their respective owners. All rights reserved. The information and the product described herein is subject to change without notice.
Sailing Past a Billion

Racing yacht design researchers push flow simulation past a meshing milestone.

By Ignazio Maria Viola, Yacht Research Unit, University of Auckland, New Zealand, Raffaele Ponzini, High-Performance Computing Group, CILEA Consortium, Milan, Italy and Giuseppe Passoni, Maritime Hydrodynamics Department, Politecnico di Milano, Italy

Over the last few decades, the development of techniques in computational fluid dynamics (CFD) together with the increasing performance of hardware and software have helped engineers understand the role of geometrical and mechanical factors on external aerodynamics in ways that were nearly intractable in the past. In recent years, several leading America’s Cup sailing teams have become top-shelf users of flow simulation software by pushing the envelope of existing meshing and solver technology. Just a decade ago, experiments on physical models — using wind tunnels and towing tanks — were the main tools for the top teams in their external aerodynamic and hydrodynamic investigations. The option of simulating a number of boat designs in a virtual environment has been shown to have several advantages, including full control of all the parameters involved, repeatability of the measurements, and the ability to simulate nonstandard sailing condition scenarios.

In the 2003 America’s Cup, in New Zealand, only a few racing syndicates had adopted fluid flow simulation as an effective design tool, though by the 2007 Cup, in Spain, almost all of the 12 competing teams had recognized the value of investing resources in both experimental tests and computational research. Nevertheless, for several technological reasons, there is still a reliability gap between experimental- and simulation-based results. One of these is the extremely complex flow around a racing yacht, particularly in downwind conditions.

To design the sail plan for an International America’s Cup Class yacht, a model-scale boat is commonly tested in a wind tunnel. To perform the same test in a virtual environment, all of the turbulent scales of the wind need to be simulated — from the largest, which draw energy from the mean flow, to the smallest, which are associated with the viscous dissipation that extracts energy as heat. It is possible to estimate the overall number of cells required to simulate all of the turbulent scales. This theoretical cell count is directly related to the Reynolds number, which is the ratio of inertial forces to viscous forces, and it is of the order of 10 billion. If such a number

![Graph showing GFlops in top500.org Ranking](image)

![Graph showing Number of Cells in Sail-Plan Computations](image)
of cells were achievable, a direct numerical simulation (DNS) could be achieved, which is widely considered to be as accurate as a full-scale measurement. Unfortunately, generating such a high number of cells would require a huge amount of memory to create the mesh and to then perform the fluid dynamics computation. Prior to the 2007 Cup races, a state-of-the-art mesh had just 10 million cells, meaning that it was necessary to use turbulence models to account for the effects of the smallest turbulent scales on the mean flow.

In 2008, a researcher involved in design for a leading contender in the 2007 Cup [1] collaborated with members of the CILEA inter-university consortium and the Maritime Hydrodynamics Department from Politecnico di Milano to achieve the most realistic simulation of a racing yacht thus attempted [2]. The resulting 1-billion-cell CFD model was therefore two orders of magnitude greater than the previous state-of-the-art mesh size in wind engineering. To achieve such an enormous cell count, the researchers reconstructed the sail shapes along with a simplified model of the hull and rig using GAMBIT and TGrid pre-processors from ANSYS, which resulted in an initial grid of 16 million tetrahedral cells. This grid was then imported into ANSYS FLUENT flow simulation software and partitioned into 512 parallel processes so that it could be run on CILEA's powerful supercomputer, known as Lagrange. Using a hanging-node algorithm, each tetrahedral cell was subdivided into eight smaller cells, which grew the mesh to 128 million cells. By repeating this procedure a second time, the team arrived at a final mesh of just over 1 billion cells.

Running on 512 CPUs, the job occupied 2 terabytes (TB) of RAM for just over a week (170 hours) to complete the calculation of flow velocities and pressures. Performing such a large calculation in parallel was imperative, as the time necessary for a serial process to complete the same computation would be more than 10 years. By calculating the pressure at each cell of the computational mesh, the team could determine the aerodynamic coefficients and compare them with experimental tests performed in the Politecnico di Milano’s twisted flow wind tunnel.

Running an ANSYS FLUENT simulation with a billion cells — the first commercial simulation of its kind focused on a single computational model — shows the possibility of performing very accurate CFD modeling in the aerodynamics of downwind sails using leading-edge hardware and software resources. Though this simulation had 100 times more mesh density than other recent CFD computations performed in America’s Cup boat design, the mesh was still 10 times coarser than what would be needed to resolve all turbulence scales using DNS and, hence, numerical models, with their inherent assumptions, were still used to simulate the flow. As high-performance computing becomes cheaper and more accessible, however, the DNS goal is in sight. Another important goal in the near future of racing yacht modeling is to couple the CFD computations with a fully 3-D shape optimization procedure. This would overcome the sail designer’s requirement to perform a physical wind tunnel test to determine a finite number of trims for different sails beforehand. Until that time arrives, though, the benefits of complementing the global accuracy of wind tunnel testing with the local insight of the flow simulation continue to be affirmed by top America’s Cup design teams from around the world.

References
SIMPLIFYING HPC SOLUTIONS
FOR MAXIMUM SIMULATION POWER

By reducing the complexity of High Performance Computing solutions Dell can help you increase performance while also reducing infrastructure costs and power consumption.

Enabling you to focus your resources on creating the powerful computing environment you need to solve the toughest simulation challenges.

SIMPLIFYING HPC AT DELL.COM/HPC

Accelerate your ANSYS solutions using parallel tracks
Optimize ANSYS multiphysics simulations on multi-core processors
- ANSYS® Mechanical™
- ANSYS® CFD™
- Ansoft Products

www.ansys.com
Optimizing Options

Technologies converge in ANSYS Workbench for parametric fluid structure interaction analysis.

By Aashish Watave, Technology Specialist, ANSYS, Inc.

The ANSYS Workbench environment provides a convenient way to conduct parametric studies for geometry shape and size variations as well as boundary conditions, which can be extended to cover multiphysics simulations within a single working environment. In this example, the effect of a butterfly valve position on the fluid flow rate and pressure drop across the valve is calculated using ANSYS FLUENT software, and the effect of fluid pressure on the valve deformation is determined using ANSYS Structural capabilities.

The ANSYS Workbench project schematic shows the two analysis systems used in this project and how they are connected to share data with each other: a fluid flow analysis system using ANSYS FLUENT and an ANSYS static structural system. The fluid and structural systems share the same geometry. Results from the fluid flow analysis are used to assign boundary conditions for the structural analysis, and the blue lines in the project schematic indicate information sharing between different analysis systems.

1. The geometry can be defined in two ways: It can be imported from a CAD system and, if necessary, simplified, repaired or prepared for simulation using the ANSYS DesignModeler tool within ANSYS Workbench; alternately, the geometry can be created in ANSYS DesignModeler. Certain geometric inputs can be set as parameters in the tree outline view. For the butterfly valve, the disk angle is defined as an input parameter, and a 15-degree initial value was assigned.
All the design point parameters are defined in this manner using the design point table, which is created as soon as the first parameter is defined. This table is represented by a bar called Parameter Set on the project page. It may be viewed by double-clicking on the parameter set bar. Arrows in the ANSYS Workbench project schematic pointing toward the analysis system indicate input parameters, and those pointing toward the parameter set bar indicate output parameters from the given system.

For the fluid flow simulation, the geometry is first meshed using the ANSYS Meshing tool. Zones that are unnecessary for fluid flow simulation, such as the valve body thickness, are suppressed when meshing the fluid zones. However, the valve body and valve disk thickness are maintained as part of the actual geometry since they are required for the ANSYS structural analysis. In ANSYS Meshing, the boundary zones (such as inlet and outlet) required for fluid flow analysis are identified using named selections. These named selections are persistent throughout the project and appear in other tools, such as ANSYS FLUENT. Setup of the problem in ANSYS FLUENT proceeds as normal with input parameters defined as part of the problem setup. For our example, inlet pressure is defined as a named selection in ANSYS Meshing and an input parameter in ANSYS FLUENT, and is shown in the design table on the project schematic as P8-ip1.

The results obtained from the ANSYS FLUENT solution can be post-processed using the CFD-Post tool by clicking on the Results cell in the project schematic. Expressions can be used to define output parameters such as pressure drop (∆P) or average surface pressure (on a given surface; for example, the valve disk). The output parameters defined in CFD-Post are available in the design point table as output parameters. Alternatively, output parameters can be defined in ANSYS FLUENT.

The static structural analysis shares the same geometry used by the fluid flow system. Geometry parts that are not required for structural simulations, such as fluid flow volumes, are suppressed in the ANSYS Meshing tool. For each design point, the pressure distribution on the valve body and valve disk obtained from the ANSYS FLUENT results is used as a boundary condition for the ANSYS Structural simulation, with the data for the corresponding zones being mapped and interpolated between the two systems accordingly. Information such as maximum deformation and maximum/minimum stress levels from the static structural analysis are defined as output parameters. After completing the analysis for a base point (Design Point No. 0), additional design points can be directly defined in the design point table. Subsequent simulation results are obtained by updating all or selected design points from the table. ANSYS Workbench updates the geometry and mesh and then performs both simulations, including automatically updating the results in the design point table. The project instances for each design point can be saved separately using the Export checkbox in the design table.
Analyzing Nonlinear Contact

Convenient tools help analyze problems in which the contacting area between touching parts changes during the load history.

By Sheldon Imaoka, Technical Support Engineer, ANSYS, Inc.

In a wide range of structural applications, bonded contact element methods are sufficient to calculate stresses between parts in assemblies in which multiple components are bolted, welded, glued or otherwise joined. In cases such as gears, cams, levers and other assemblies with moving parts, however, the contact area between components changes during the load history. For these types of nonlinear analyses, mechanical solutions from ANSYS provide robust contact technology along with diagnostic tools that can help obtain converged, accurate solutions to problems that otherwise would be quite challenging to handle.

Initial Contact Information

Rigid-body motion in which parts are not initially in contact is often a common convergence problem. Defining and verifying contact between parts, therefore, is an important first step in the analysis. Initial contact status is easily checked in mechanical solutions from ANSYS, including whether or not parts that are thought to be in initial contact are truly touching.

Using mechanical simulation within the ANSYS Workbench environment, you may insert a Contact Tool underneath the Connections branch, as shown in Figure 1. Specific contact regions can be selected or deselected, and plotting/listing of only the contact or target side is possible from this worksheet. Multiple Contact Tool branches may also be inserted for reviewing contact regions in different groups.

The Initial Information branch is included by default, although users may insert contour results of initial Status, initial Penetration or initial Gap as well. If you right-click on Contact Tool and select Generate Initial Contact Results, the initial contact information will be calculated and presented in tabular form, as shown in Figure 2. The rows conveniently summarize the type of contact and highlight possible problems in different colors: orange (possibly large penetration or gap), yellow (frictionless or frictional contact pair having an initially open state) or red (bonded or no-separation contact initially having an open state). This allows models with large numbers of contact regions to be easily examined.

Contact Result Tracker

Nonlinear solutions of large models may consume considerable CPU time, after which users may be disappointed to find that incorrect model setup or unanticipated contacting areas led to an invalid solution. The ability to track results can help alleviate such problems. Prior to solving, you can request certain results for specific contact regions and monitor these results during the course of the analysis. Then, if the contact solution starts to deviate from the expected behavior, the analysis can be stopped without having to wait until the end of the run to find out that the analysis setup may not be correct.

To track contact results, drag-and-drop a contact region branch from the Connections branch to the Solution Information branch. In the Details view of the Result Tracker that appears, the user may select a number of items for a given contact region, including but not limited to the number of contacting elements and the maximum contact pressure. Add as many Result Tracker items as necessary.

As an example, Figure 3 shows the number of contacting elements for seven contact regions while the nonlinear solution is progressing. Note that the contact region Frictional-seal3 is in near-field (open) contact throughout the solution. On the other hand, the contact region Frictional-opening was open until a time of 0.4, when a large number of elements came into contact. This helps a
During the Newton–Raphson iteration, convergence is achieved when force equilibrium is satisfied. When using mechanical simulation in ANSYS Workbench, the user can request Newton–Raphson residual output, so regions of high out-of-balance forces can be reviewed. This helps in determining where force imbalance is high and, if the area is associated with a contact region, which contact regions may have defined a contact stiffness that is too high.

In mechanical simulation within ANSYS Workbench, prior to initiating a solution, select the Solution Information branch. In the Details view, a value of “4” can be entered for the Newton–Raphson residuals. In cases of an incomplete solution, contours of Newton–Raphson residuals for the last four iterations will be available under the Solution Information branch, and contour plots can be generated as shown in Figure 5. In this example, a solid cylinder pushes down on two hollow cylinders; half of the model is displayed. The highest residuals are between the two concentric hollow cylinders, indicating that the contact stiffness defined for that region may be too high and, consequently, should be lowered.

Nonlinear Diagnostics

The contact stiffness $k_n$ is the most important contact parameter for the penalty-based approach, influencing both convergence behavior and accuracy. During equilibrium iterations, if the force residuals plateau (as shown in the example in Figure 4), chances are high that contact stiffness is preventing force convergence. While contact stiffness may be a cause for the high residuals, you may not be certain simply by looking at the force convergence behavior.

During the Newton–Raphson iteration, convergence is achieved when force equilibrium is satisfied. When using mechanical simulation in ANSYS Workbench, the user can request Newton–Raphson residual output, so regions of high out-of-balance forces can be reviewed. This helps in determining where force imbalance is high and, if the area is associated with a contact region, which contact regions may have defined a contact stiffness that is too high.

In mechanical simulation within ANSYS Workbench, prior to initiating a solution, select the Solution Information branch. In the Details view, a value of “4” can be entered for the Newton–Raphson residuals. In cases of an incomplete solution, contours of Newton–Raphson residuals for the last four iterations will be available under the Solution Information branch, and contour plots can be generated as shown in Figure 5. In this example, a solid cylinder pushes down on two hollow cylinders; half of the model is displayed. The highest residuals are between the two concentric hollow cylinders, indicating that the contact stiffness defined for that region may be too high and, consequently, should be lowered.

Contact Post-Processing

Post-processing is the most important step of any analysis, and contact problems are no exception. Always review contour plots of contact status, pressure and penetration to verify that the mesh adequately captures the contact behavior and that results are correct. Contact penetration is in units of length, so deformation can be compared in the same direction as contact. If the penetration is a small fraction of the deformation, you can safely assume that any variation in penetration would not affect results significantly.

In Figure 6, maximum penetration is $4.256 \times 10^{-3}$ mm. This value can be compared to the deformation on the same contact surface to verify that the penetration is negligible. Checking the contact status may indicate that contact detection is occurring at a very localized region and may warrant a finer mesh.
Take a Test Drive

Test drive your models and simulations at the QLogic NETtrack Developers Center. We’ll show you how fast you can go!

When time-to-solution is important, you need the best combination of hardware, interconnects and software. **ANSYS, HP, Intel, and QLogic** are partnering to bring you industry-leading performance for ANSYS applications.

www.ansys.com
www.hp.com/go/cae
www.intel.com/technology/business/hpc.htm
www.qlogic.com

Technology for better business outcomes.