Get Inspired!

2008 International ANSYS Conference: Inspiring Engineering

Pittsburgh, PA, U.S.A.
August 26 – 28, 2008

Join us at the 2008 International ANSYS Conference to discover how simulation can lead to truly inspired engineering.

This exciting event will be the first of a series of ANSYS conferences occurring around the world in 2008.

Don’t Isolate Engineering Simulation

Leverage the tremendous value of the technology by tightly integrating simulation into your company’s product development process.

The benefits of integrating simulation into engineering processes are well known. By making analysis a routine part of design — especially up front in the cycle — companies can spot and fix problems early, cut down on testing numerous physical prototypes, optimize product performance and create innovative designs that often would not be feasible without the use of simulation to explore alternative concepts. Software vendors across the board have been touting this message for years. Yet ironically — and unfortunately for the CAE user community — software from many vendors does not support such integration.

One of the big problems is that many simulation programs are not interoperable with other software, especially with a broad range of CAD programs, other CAE technologies and data management tools. Because the architectures of these CAE programs are generally closed and rigid, programs simply don’t talk to one another and exchange information well. So users must go through inefficient and error-prone processes of converting data, reworking models, duplicating mesh representations and copying information from one system to another.

Another difficulty is that the programs often are aimed solely at particular user skill levels and specific types of analyses, or they may run only on certain computers. Moreover, users may find that some software is not all it’s cracked up to be in handling the wide range of complex problems and real-world applications that engineers routinely encounter in their work. As a result, engineering simulation at many companies may be performed sporadically in isolation from other tools and groups throughout the organization — working against the full integration of analysis into product development processes and negating many of the potential benefits of the technology.

Although such problems are rampant throughout the simulation industry, some software has been developed that takes the user community’s needs into consideration. Case in point can be found in this issue’s Spotlight on Products and Technology section covering the capabilities of simulation solutions from ANSYS Inc. An open architecture enables the software to work well with other programs through bi-directional CAD associativity, and direct links to other analysis solvers (including competitive programs) and product lifecycle management (PLM) systems. The software has been purposefully developed to be scalable across a wide range of users and computers. This enables companies to deploy simulation solutions in a flexible manner across the organization. It also enables users to drill down as deep as they need to in a broad range of disciplines — including the simulation industry’s most comprehensive multiphysics portfolio.

In this way, innovative companies can implement advanced simulation technology in a completely integrated manner using a single platform and common infrastructure, while competitors struggle with piecemeal analysis and fragmented workflow isolated from the rest of the organization’s programs and processes.

John Krouse, Senior Editor and Industry Analyst
Table of Contents

FEATURES

4 SPORTS
Simulating Swimwear for Increased Speed
Speedo’s new full-body swimsuit takes advantage of simulation technology in pursuit of gold medals and world records.

7 NEWS
ANSYS Signs Agreement to Acquire Ansoft
The complementary business combination of ANSYS and Ansoft will create the leading provider of best-in-class simulation capabilities.

Spotlight on Engineering Simulation
Products and Technology

9 Engineering Simulation for the 21st Century
Five key principles guide the development of simulation products and technology at ANSYS.

12 Putting Engineering Knowledge to Work
New technology enables efficient sharing of rich simulation information and provides enterprise-wide benefits.

14 Applying Six Sigma to Drive Down Product Defects
Probabilistic design and sensitivity analyses help engineers quickly arrive at near-zero product failures in the face of wide manufacturing variabilities and other uncertainties.

17 The New Wave of Fluids Technology
Fluid flow simulation software from ANSYS provides a broad range of scalable solutions.

20 Multibody Dynamics: Rigid, Flexible and Everything in Between
Advances in simulation solutions for machine features accommodate more complex designs.

24 Nonlinear Simulation Provides More Realistic Results
When parts interact and experience large deflections and extreme conditions, nonlinear technology is required to simulate real-life situations.

26 High Performance from Multiphysics Coupled Simulation
Engineers use ANSYS Multiphysics to study the mechanical strength and thermal performance of an innovative thermoelectric cooler design.
Recently we asked you, the readers of ANSYS Advantage, for help in charting the magazine's path to the future. Your response was overwhelming. Thanks for taking the time to share your opinions and comments.

We now have a better understanding of your needs and are already implementing some of your suggestions. You'll see some changes to the magazine in this and upcoming issues. For instance, we've already started a new section — Outside the Box — which, in each issue, will report on some aspect of simulation with a human interest slant. We hope you enjoy it. Other changes will be coming in subsequent issues.

As for the winners of the navigation prizes — GPS unit and compasses — they were Stéphane Trehorel of Geo Technology S.A., Andreas Wiese of ELAU AG and Stefan Jelenko of Litostroj E.I. Congratulations!

Although the survey is officially over, we are always happy to receive your suggestions or your article submissions. Send an email to ansys-advantage@ansys.com at any time. We look forward to hearing from you.

Thanks for Helping Us Navigate
Simulating Swimwear for Increased Speed

Speedo’s new full-body swimsuit takes advantage of simulation technology in pursuit of gold medals and world records.

By Leigh Bramall, PR Specialist, ANSYS, Inc.

Most of us are familiar by now with the use of simulation in sports such as Formula One racing, NASCAR® and, more recently, America’s Cup yacht racing. The potential applications are obvious with aerodynamics on a racing car or hydrodynamics on a yacht. But the use of simulation technology is spreading far beyond these more obvious applications.

Swimwear designer and manufacturer Speedo® is a name synonymous with the pool and swimming competition. The company has an 80-year history of developing swimsuits for elite swimmers, all the while successfully maintaining its leading position in the industry. In the 1920s, Speedo made history with the Racerback — the world’s first non-wool suit. More recently, the company introduced the full-body swimsuit to the competitive swimming arena with its FASTSKIN® suit, which was designed to reduce drag and optimize performance and was worn by the majority of competitors at the 2000 Sydney Olympics.

Computational fluid dynamics (CFD) first entered the sport of competitive swimming in a significant way with the development of Speedo’s FASTSKIN FSII swimsuit, developed for use at the 2004 Athens Olympics. February 2008 saw the further development of Speedo’s CFD program with the global launch of its LZR RACER® suit ahead of the Beijing games. Using FLUENT technology from ANSYS, Inc., Speedo used CFD analysis to guide, test and refine the final design of the suit, bringing together a range of research with the goal of improving performance.

The LZR RACER suit is the result of three years of work conducted by Aqualab, Speedo’s own in-house R&D group. Headed by Jason Rance, the Aqualab® group oversaw a research program that employed multiple partners — the University of Otago in New Zealand, the University of Nottingham in the United Kingdom, the Australian Institute of Sport, Optimal Solutions in the United States, NASA in the United States and ANSYS. The outcome is a swimsuit that has 10 percent less passive drag than the FASTSKIN FSII suit and 38 percent less passive drag than an ordinary LYCRA® suit.

To begin the research, the team attempted to identify the fabric with the least possible drag. They started by taking body scans of more than 400 elite swimmers to provide geometries for testing more than 100 different fabrics and suit designs. Work focused on developing two different fabrics: LZR Pulse®, an ultra-lightweight, powerful, lowdrag, water repellent, fast-drying fabric to be used as the base woven material for the suit; and LZR panels made of a low-drag, ultra-powerful polyurethane membrane to be bonded onto the base woven material to reduce drag at specific high-drag locations.

To test the fabrics and create a suit with the lowest drag, the Speedo researchers used one of the world’s most advanced water flumes at the University of Otago. NASA also was employed, with the space agency evaluating the surface friction of fabric candidates in its low-speed wind tunnel. The wind tunnel was operated at 28 meters per
second to simulate a swimmer moving at 2 meters per second in water. NASA used a smooth aluminium plate as a benchmark for the fabric tests. Results showed that the smoothest fabric had the lowest drag, with the final fabric chosen for the LZR panel producing test results comparable to the drag results for the aluminium plate.

Following the water flume and wind tunnel tests, the Speedo team put the results into practice in the real world at the Australian Institute of Sport, conducting time-trials with swimmers wearing the new suit and standard training wear. The drag reductions identified in the water flume and wind tunnel translated to a 4 percent increase in speed for swimmers when wearing the new suit as opposed to wearing training swimwear. The new suit also delivered a 5 percent improvement in the swimmers’ oxygen utilization when compared to trials performed in the training wear. These efficiencies are significant in a sport in which success or failure can be measured in hundredths of a second.

The Speedo team took further steps to reduce drag at every opportunity. They designed the suit to achieve a compression effect through the tensile properties of the materials — powerful fabrics that compress key body areas. Effectively, these features help to mold the swimmer’s body into a more streamlined shape, reducing the hydrodynamic profile and minimizing oscillations in muscles that might disturb flow. The team also used ultrasonic welding to bind the material sections of the suit together, eliminating all seams that could disturb flow and induce drag.

For further analysis, ANSYS performed CFD studies in association with Dr. Herve Morvan of the University of Nottingham. The focus was on passive drag, which occurs when the body is in the glide position with arms outstretched in front and legs outstretched behind. Swimmers maintain this position for up to 15 meters immediately after diving and for a similar period after kicking off under water following each turn. For this reason, this position is critically important in a competitive race situation. Testing focused on this position, providing indications of overall performance of the suit during racing conditions.

The researchers used CFD analyses to identify areas in which both skin- and form-drag occur. Skin-drag is inherent in swimsuit material properties, since fluid flows over the fabric, and in local flow conditions, speed in particular. It is induced by the local velocity gradients that create a shear force due to the viscous properties of the fluid. Form-drag, on the other hand, is a result of the swimmer’s body traveling through the fluid; the goal is to make the flow path as smooth and undisturbed as possible, thereby decreasing the drag. The CFD simulations involved precise boundary layer meshing techniques using software from ANSYS and resolved fine fluid flow details using the precision scanned geometries of elite swimmers. Speedo carried out all of these CFD studies as part of ongoing research conducted since the 2004 Olympics in Athens.
Using the wealth of detailed fluid dynamics data from the CFD studies generated with software from ANSYS, Speedo researchers were able to guide the final design of the new suit — in particular, the precise location of the ultra-low-drag LZR panels, which were bonded onto the LZR RACER suit. In the end, the strategically placed LZR panels were found to reduce skin drag by a startling 24 percent in comparison to Speedo’s previous FASTSKIN fabric.

Dr. Keith Hanna of ANSYS, who lectures on the application of CFD technology in sport, believes the scope for the application of CFD and simulation in general will only increase in the future. Hanna stated, “CFD is a powerful technology, and the accuracy of the results from this study have given Speedo confidence in the benefits of applying CFD in the design of future swimsuits. However, the big development in years to come could be the use of comprehensive multiphysics technology for elite swimsuit development — that is, the use of CFD with other physics such as structural simulation to actually simulate every aspect of real-world physics found in a competitive swimming scenario.

“The physics involved in simulating a moving swimmer are extremely complex, but the potential is there. Industries such as aerospace and automotive are increasingly turning to a multiphysics simulation solution as the only way of ensuring that all parameters are accounted for in their design process. In the instance of simulating performance in competitive swimming, a multiphysics approach would mean not only that CFD be used to analyze hydrodynamic flow and drag around a swimmer, but also that structural software be used to simulate how the suit itself may deform during a swimming stroke, for example, and how this affects drag.”

In the first ten weeks following its launch in February, swimmers wearing the Speedo LZR RACER swimsuit set 35 world records.
ANSYS Signs Agreement to Acquire Ansoft

The complementary business combination of ANSYS and Ansoft will create the leading provider of best-in-class simulation capabilities.

ANSYS, Inc. and Ansoft Corporation, a global provider of electronic design automation (EDA) software, announced on March 31 that a definitive agreement was signed in which ANSYS will acquire Ansoft for a purchase price of approximately $832 million. Ansoft is a leading developer of high-performance EDA software, which is based on more than 25 years of research and development by world-renowned experts in electromagnetics, circuit and system simulation. Engineers use Ansoft products to simulate high-performance electronics designs found in mobile communication and Internet devices, broadband networking components and systems, integrated circuits, printed circuit boards and electromechanical systems. Products from Ansoft are used by blue chip companies as well as small and medium-sized enterprises around the world.

The acquisition of Ansoft is the first foray for ANSYS into the broader EDA software industry and will enhance the breadth, functionality, usability and interoperability of the combined ANSYS portfolio of engineering simulation solutions. The combination is expected to increase operational efficiency and lower design and engineering costs for customers, as well as accelerate development and delivery of new and innovative products to the marketplace; it also is expected to give ANSYS one of the most complete, independent engineering simulation software offerings in the industry. This reaffirms and strengthens the ANSYS commitment to open interface and flexible simulation solutions that are driven primarily by customer demand, flexibility and choice. With more than 40 direct sales offices and 21 development centers on three continents, the combined company will employ approximately 1,700 people.

“No we are very excited about the state-of-the-art technologies that Ansoft adds to the simulation capabilities of ANSYS,” said James E. Cashman III, president and chief executive officer of ANSYS, Inc. “Both companies have a strong commitment to their customers and employees, share a passion for the development of innovative products and services, and have a history of world-class execution. It expands our offerings into electronic designs, a critical area. Now we can bring the richest engineering calculations to predict how a product will perform in the real world. Electronics and the electromagnetic field are obviously a big part of that. With Ansoft, we can take a complete system and the entire environment and allow designers to evaluate their entire product.”

“This merger brings together two great companies with a shared vision and strong engineering focus,” said Dr. Zoltan J. Cendes, the founder, chairman of the board and chief technology officer of Ansoft Corporation. “The combination of our R&D teams, complementary technological strengths and commitment to quality will enhance our ability to deliver comprehensive, innovative, world-class simulation software technologies that customers demand.”

“We are excited about bringing two great Pittsburgh-based companies together to create an exciting opportunity for aspiring engineers, computer scientists and professionals to join us in our mission to democratize the use of simulation across the globe,” added Cashman.
Analyzing Plastic Parts?

Fiber orientation in plastic parts can significantly affect structural properties.

Using Moldflow design analysis solutions, you can map Moldflow simulation results to your structural mesh and increase the accuracy of your structural analysis.

To learn more, register for your FREE DVD at www.moldflow.com/ANSYS

Moldflow is the world’s leading provider of plastics simulation software that empowers more users to optimize more designs across their enterprise.
Five key principles guide the development of simulation products and technology at ANSYS.

By Chris Reid, Vice President, Marketing, ANSYS, Inc.

Technology is the lifeblood of ANSYS, Inc., and the basis for everything we offer our customers. For more than 35 years, ANSYS has been a pioneer in the application of finite element methods to solve the engineering design challenges our customers face. During that time, the evolution of our industry, products and technology has been nothing short of amazing. Fueled by a corresponding increase in the power-to-price ratio of the computing world, the problem size and complexity of simulations have grown to impressive dimensions. The net effect of this is evident in almost every facet of life — from the cars we drive to the energy we use, the products we buy, the air we breathe and even the devices we insert into our bodies to maintain our health.

How have we accomplished this near 40-year run of groundbreaking achievement in engineering simulation and modeling? Staying true to our vision and strategy has certainly been a major factor. Unlike others, ANSYS has never wavered from its core business of engineering simulation software. Instrumental to that vision is our commitment to advanced technology — the cornerstone of our business and
value to our customers. After all, it is our products and technology that enable companies to create the most innovative and globally competitive products for their industry.

Also instrumental to our vision are five principles that guide the development of our products and technologies. The first is *unequalled depth*. Simply stated, for each of the key areas of simulation and modeling technologies — whether it be mechanical, fluid flow, thermal, electromagnetics, meshing or others — we offer a depth of capability that is second to none. This depth has been created over time by reinvesting in the research and development of new technologies, and supplemented by key acquisitions and partnerships along the way. Today, the results speak for themselves in the richness of what we offer our customers, regardless of their specific simulation requirements.

The second guiding principle is *unparalleled breadth*. In this regard, ANSYS has assembled a complete range of simulation capabilities — from pre-processing to multiple physics to knowledge management. Our customers see this as a tremendous benefit, because they know we can provide a solution for each specific area of analysis and that we provide rich depth across our entire portfolio of products and technologies. Some companies, perhaps, can lay claim to this in one or two areas, but we offer this depth and breadth for the full range of simulation and modeling techniques.

In offering both technological depth and breadth, our customers are able to run simulations that are more sophisticated, more complex and more representative of the real world. Utilizing such a *comprehensive multiphysics* approach — our third guiding principle — enables engineers to simulate and analyze complete systems or subsystems using true virtual prototyping. Increasingly, companies realize that a multiphysics approach is essential to attain the most accurate and realistic simulation of a new product or process design. At ANSYS, we not only provide the technologies to do this, but we make them all interoperable within the unified ANSYS Workbench environment. Thus, the user can configure a multiphysics analysis and avoid the need for cumbersome file transfers or intermediate third-party software links. Our technology inherently provides the infrastructure, saving implementation time while providing measurable benefits in speed and robustness as well.

The old adage “one size fits all” is certainly not the case in the world of engineering simulation. Despite the common threads that appear everywhere simulation is used, there are also real differences. Some industries, such as automotive and aerospace, are mature in their use of these tools, while others, such as healthcare, are relative newcomers. Companies within the same industry can be at markedly different stages of adoption, and users within any one company may have vastly different needs or experience with simulation tools. There is a need for flexibility. Customers must be able to adopt the appropriate level of simulation and know they will have latitude in how they move forward.

At ANSYS, we call this *engineered scalability* — guiding principle number four. Why “engineered”? Our scalability is by design and is specifically engineered into the technology we have developed. The depth of our technology allows customers to choose the appropriate level of technology for their needs yet scale upward as their requirements evolve and grow. If the customer is a small company with just a desktop or modest compute resources, or if it is large with hundreds of machines in large-scale compute clusters, our software runs efficiently and brings value. In a similar vein, if the number of users is very small or in the hundreds, scalable deployment has been factored in. Likewise, if the customer is an infrequent user, a designer who wants to perform a simple simulation or an expert analyst, we have the appropriate level of tool for each of those levels of experience. Underpinning this seamless range of capability — from the automated to the most sophisticated and customizable — is the same advanced technology, scaled up or down accordingly.

Technology isn’t of much use to the customer if it’s extremely inflexible to apply, scalable or otherwise. All of it must be usable in a way that makes sense for the company and its design and development processes, as well as alongside other programs it may have selected for their engineering systems strategy. The vision needs to be flexible and adaptive, not rigid and constraining. In this regard, ANSYS adheres to a fundamental tenet of *adaptive architecture* — the fifth
guiding principle. We recognize the mission-critical nature of what we provide and also how crucial it is that our technology fits within the customer’s overall system. There can be CAD systems, selected third-party codes for niche applications, or legacy and in-house software, all of which remain critical components of the overall process. We need to coexist with these and, in fact, enable them to be included in the overall workflow as painlessly as possible.

Many companies are investing in product lifecycle management (PLM) systems. These constitute a major investment and require data exchange with the simulation software. The ANSYS Workbench platform and the new ANSYS Engineering Knowledge Manager (EKM) technology are designed to provide functional coexistence with PLM systems, which actually improves their value to the customer. Adaptive architecture means what it says — ANSYS products and technology can adapt to the customer’s specific situation. We can be the backbone, coexist peer-to-peer or be a plug-in, whatever the need may be.

Five simple phrases — unequalled depth, unparalleled breadth, comprehensive multiphysics, engineered scalability and adaptive architecture. These five tenets are what drive our product development strategy with every dollar we invest. We also think they are the reason that 96 of the top 100 industrial companies on the FORTUNE Global 500 list, as well as another 13,000 customers around the world, use technology from ANSYS. The ANSYS simulation community today is the world’s largest, and by continuing to pursue our strategy of Simulation Driven Product Development and adhering to these five guiding principles, we see no reason why our vision of placing simulation tools in the hands of every engineer shouldn’t become a reality in the near future.

Putting Engineering Knowledge to Work

New technology enables efficient sharing of rich simulation information and provides enterprise-wide benefits.

By Michael Engelman, Vice President, Business Development, ANSYS, Inc.

The last three decades have witnessed the evolution of computer-aided engineering (CAE) from a tool used by analysts in a research and development department to one that is integral to the entire product design and lifecycle process. Companies around the world are making greater use of upfront analysis and complex system simulation. With the ongoing integration of CAE into the design process, the focus is shifting from technology issues such as improved simulation techniques, physics modeling and ease-of-use to usage-focused questions such as “How do I better manage and share the voluminous data that is being generated?” or “How do I better capture the engineering expertise that the simulation results represent?” The answer to these questions is often referred to as simulation process and data management (SPDM).

SPDM presents a whole new set of challenges to CAE practitioners. The current focus centers on accessibility — that is, how do the right people get the right data at the right time? More often than not, keeping track of this information is left to the individual analyst or engineer who generated it, so typically at the end of a project it is buried in obscurity somewhere on a hard drive. According to a recent survey by Collaborative Product Development Associates (CPDA), 47 percent of all simulation results are stored on the engineer’s local workstation. This valuable intellectual property is usually lost forever when individuals leave a team or company. As anyone who has tried knows, tracking down old data files and analysis models from past simulation projects is difficult or often impossible, even for the people who created them. Consequently, simulations are often redone from scratch — rather than by performing a simple modification to an existing case or model. The result is a significant loss in time and productivity.

With rapid globalization now permeating all aspects of many companies’ operations, engineering groups located at different locations around the world constitute virtual 24-hour-a-day development organizations. Effective collaboration and communication are essential to support this global mode of operation. Ineffective communication hurts the entire team, from the engineer who is trying to explain design challenges and concerns to his or her manager, to the teams that need to consider external factors affecting their development process and design. Using tools that can help convey simulation results to all members of a team at every level across the enterprise — regardless of their technical background — can dramatically boost the effectiveness of the team, the product development process and, finally, the quality and performance of the product.

Corporate knowledge is a key business asset in a company’s quest for innovation and competitive advantage. Creating, capturing and managing a company’s simulation
expertise is critical to enabling innovation. It empowers users to build on previous experience and fosters continual improvement and collaboration of the expert analysts and design engineers. Effective process management tools that capture simulation best practices, deploy managed simulation tasks and processes, and plug into internal applications within a unified environment are essential to achieve these goals — though they must also require minimal effort and maintenance costs.

Managing simulation data and processes within this context is a specialized subset of the broader product lifecycle management (PLM) vision. This discipline is based on the digital management of all aspects of a product’s lifecycle, from concept and design through manufacture, deployment, maintenance and eventual disposal. Unfortunately, those needs that are specific to simulation and SPDM are often overlooked or poorly addressed by today’s PLM systems. This is a result of SPDM being more demanding than the file/document-centric approach of PLM and related product data management (PDM) systems. Simulation data is richer, more complex and typically many orders of magnitude larger than other types of product data. An SPDM system is complementary to a PLM system and can add significant value when designed to work in close conjunction with PLM.

The ANSYS Engineering Knowledge Manager (EKM) technology, now in its initial release, is aimed at meeting these challenges with extensive capabilities: archiving and management of simulation data, traceability and audit trail, advanced search and retrieval, report generation and simulation comparison, process/workflow automation, and capture and deployment of best practices. It is a Web-based SPDM framework aimed at hosting all simulation data, workflows and tools, whether in-house or commercial, while maintaining a tight connection between them. While providing seamless integration with simulation products from ANSYS — including the ability to automatically extract and organize extensive information about ANSYS software–based simulation files when they are uploaded into the repository — the ANSYS EKM tool is an open system that can manage any type of in-house or third-party simulation products, files or information as well. Moreover, it is a scalable solution that can be effectively used by small workgroups, distributed teams of engineers or the entire enterprise.

With tools and developers that have histories stretching back to the formative years of simulation, ANSYS understands the complexity and challenges of simulation. ANSYS EKM technology was created with an appreciation that access to simulation; developing effective processes for incorporating simulation into individual, workgroup and enterprise-wide efforts; and managing simulation efforts within a larger development or industrial process is a complicated effort — one that can be made simpler. Having access to the right tools, developed by a team that has devoted years to understanding the challenges of simulation, can streamline the incorporation of virtual product development efforts into traditional workflows and environments. Adding ANSYS EKM tools to the capabilities of the family of products from ANSYS empowers organizations of all sizes to better achieve the goal of Simulation Driven Product Development.
Applying Six Sigma to Drive Down Product Defects

Probabilistic design and sensitivity analyses help engineers quickly arrive at near-zero product failures in the face of wide manufacturing variabilities and other uncertainties.

By Andreas Vlahinos, President, Advanced Engineering Solutions, Colorado, U.S.A.

Companies often are focused primarily on time-to-market, but the advantages of fast product introductions may be quickly overshadowed by the huge cost of poor quality, resulting in product recalls, rework, warranty payments and lost business from negative brand image.

In many cases, such quality problems are the result of variations in factors such as customer usage, manufacturing, suppliers, distribution, delivery, installation or degradation over the life of the product. In general, such variations are not taken into consideration as part of the development of the product. Rather, the integrity and reliability of a design is typically based on an ideal set of assumptions that may be far removed from actual real-world circumstances. The result is a design that may be theoretically sound but riddled with defects once it is manufactured and in use.

Design for Six Sigma (DFSS) is a statistical method for radically reducing these defects by developing designs that deliver a given target performance despite these variations. The approach is a measure of quality represented as the number of standard deviations away from a statistical mean of a target performance value. Operating at three sigma translates into about 67,000 defects per million parts, performance typical of most manufacturers. A rating of six sigma equates to just 3.4 defects per million, or virtually zero defects. Achieving this level of quality requires a focused effort upfront in development, with design optimization driven by integration of DFSS into the process and rigorous use of simulation. In such DFSS efforts, ANSYS DesignXplorer software is a particularly valuable tool. Working from within the ANSYS Workbench platform and in conjunction with ANSYS Mechanical and other...
PRODUCTS & TECHNOLOGY: DESIGN FOR SIX SIGMA

simulation software, the program performs Design of Experiments (DOE) and develops probabilistic design analyses functions to determine the extent to which variabilities of key parameters impact product performance.

The process is accomplished in four major phases: process automation, design exploration, design optimization and robust design. Utilizing the ANSYS Workbench environment, process automation ensures that simulation tasks are well defined and flow automatically to extract and evaluate key performance variables.

ANSYS DesignXplorer software then performs the DOE, running numerous (usually thousands) analyses using various combinations of these parameters. The ability to quickly and effortlessly execute such an extensive study on this wide range of parameters allows users to perform quick and accurate what-if scenarios to test design ideas. In this way, design exploration — combined with knowledge, best practices and experience — is a powerful decision-making tool in the DFSS process.

Next, design optimization is performed with the ANSYS DesignXplorer tool in order to select the alternative designs available within the acceptable range of performance variables. Design parameters are set to analyze all possibilities — including those that might push the design past constraints and violate design requirements. Finally, robust design is performed, arriving at the best possible design that accounts for variabilities and satisfactorily meets target performance requirements.

Throughout the process, ANSYS DesignXplorer software employs powerful sampling functions and probabilistic design technology. It also provides valuable output in the form of probability function design, scatter plots and response surfaces that are critical in DFSS. Seamless interfaces with parametric computer-aided design (CAD) programs — used to import geometry for analysis and to set up parametric models in mechanical solutions from ANSYS — is essential for ANSYS DesignXplorer software to automatically perform numerous iterations in which various design geometries are created and analyzed. In this way, ANSYS DesignXplorer software is an effective means of integrating DFSS into a company’s product development process. The software provides individual engineers a unified package for quickly performing probabilistic design and sensitivity analyses on thousands of design alternatives in a few hours; otherwise, this would take weeks of effort by separate statistics, simulation, DOE and CAD groups.

One recent project designed to improve hyper-elastic gasket configurations in proton-exchange membrane (PEM) fuel cells illustrates the value of the ANSYS DesignXplorer tool in DFSS applications. In this example, several gaskets provide a sealing barrier between the cell and approximately 200 bipolar cooling plates. In designing the fuel cells for commercial use in harsh environments, the goal was to lower the failure rate of the gaskets, which tended to leak on occasion — even in a carefully controlled research lab setting.

ANSYS DesignXplorer software generated a response surface showing sensitivity of each input variable to contact force.

Probability density functions of parameters that vary with each analysis iteration
First, design variables were established — gasket profile, gasket groove depth and the opposing plate’s recessed pocket groove depth — that determined the overall compressive force of the gasket under a given bolt load. These were considered to be randomly varying parameters with given mean and standard deviations as determined through probability density functions generated by the ANSYS DesignXplorer tool. The software then was set up to automatically perform a series of DOE analyses in order to determine the gasket contact force for 10,000 different combinations of these variables. Variables were randomly selected by the software for each round of analysis using the Latin hypercube sampling technique.

Using ANSYS Mechanical analysis, solutions were arrived at in which (1) nonlinear capabilities characterize the hyper-elastic gasket material properties; (2) contact elements represent contact between the gasket and plates; and (3) parametric features automatically change the geometry of the gasket configuration for each of the 10,000 analyses.

Based on these analysis results, ANSYS DesignXplorer software generated a response surface of the contact force per unit length of the gasket in terms of probabilistic input variables. With the sensitivity established for each input variable on the contact force, scatter plots of the analysis results were generated along with bell-shaped probability density functions, which were compared to the upper and lower load limits of the fuel cell and cooler interfaces. Axial forces could not be so high as to break the plates, yet not so low as to cause leaking. From this data, the ANSYS DesignXplorer tool determined the sigma quality level based on the contact force target level.

The process succeeded in arriving at an optimal gasket shape that exceeded the sigma quality level, dropping the failure rate to an impressive three parts per million — a tremendous improvement over the 20 percent failure rate that the gaskets were experiencing previously.

The entire process — including creation of the mesh models and completion of the 10,000 DOE analysis cycles — was completed in a matter of days by a single individual, as compared to months of effort that otherwise would have been required by separate design, statistics and analysis groups. Moreover, with the workflow captured in the ANSYS Workbench platform, the process now is highly repeatable and can be efficiently applied in optimizing the design of other gaskets merely by changing the CAD model and the upper/lower contact force limits.

The New Wave of Fluids Technology

Fluid flow simulation software from ANSYS provides a broad range of scalable solutions.

By Paul Galpin, Director of Product Management and André Bakker, Lead Product Manager, Fluids Business Unit, ANSYS, Inc.

The world of product engineering has come a long way in its quest to advance analysis from laborious hand-drawn sketches and simplistic models to virtual computer-created models initiated at the touch of a button. There has been a long evolutionary path from the inception of computational fluid dynamics (CFD) to today’s integration of this technology into Simulation Driven Product Development (SDPD) processes. Throughout its history, ANSYS, Inc. has been a technological champion for such commercial engineering simulation. The company has viewed simulation as the key to predicting how products will perform; it has enabled the rapid comparison of many different alternatives prior to making a design decision — well before customers might identify problems. ANSYS now has a fluids product line that is both broad and deep, along with a large commercial and academic user base that is reaping the benefits.

This CFD evolution has required, and continues to demand, that ANSYS go beyond merely providing advanced mathematical flow solvers. ANSYS espouses a multiple physics approach to simulation in which fluid flow models integrate with other types of physics simulation technologies. The ANSYS vision is clear: to provide a system of high-fidelity multidomain analysis tools to truly enable SDPD.

SDPD is centered on the highly adaptive ANSYS Workbench architecture. The next major ANSYS Workbench release will provide another big step toward this vision. It will be the first release in which a number of the original Fluent CFD products will be data-integrated into the ANSYS Workbench platform, and thus the tools will work together with various other applications from ANSYS.

The ANSYS Workbench approach allows ANSYS to provide a large variety of software choices tailored to meet individual needs while ensuring interoperability and a clear future upgrade path. This includes a very broad fluids product line with all tools falling into one of three categories: general-purpose fluid flow analysis, rapid flow modeling and industry-specific products.

**General-Purpose Fluids Solvers**

The well-known FLUENT and ANSYS CFX products are the main general-purpose CFD tools from ANSYS. These two solvers, developed independently over decades, have a lot of things in common but also some significant differences. Both are control volume-based for high accuracy and rely heavily on a pressure-based solution technique for broad applicability. They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies.
The ANSYS CFX solver uses finite elements (cell vertex numerics) to discretize the domain, similar to those used in the mechanical analysis side of the business. In contrast, the FLUENT product uses finite volumes (cell-centered numerics). Ultimately, though, both approaches form "control volume" equations that ensure exact conservation of flow quantities, a vital property for accurate CFD simulations. ANSYS CFX software focuses on one approach to solve the governing equations of motion (coupled algebraic multigrid), while the FLUENT solver offers several solution approaches (density-, segregated- and coupled pressure-based methods). Both solvers contain a wealth of physical modeling capabilities to ensure that any fluids simulation has all of the modeling fidelity required.

These two core CFD solvers represent more than 1,000 person-years of research and development. This effort translates into the key benefits of fluid flow analysis software from ANSYS: experience, trust, depth and breadth. The fluids core solvers from ANSYS are trusted, used and relied upon by companies worldwide.

Rapid Flow Modeling

ANSYS addresses the fluid flow analysis needs of designers, who work on the front lines of their company’s product development process and often need to make important design decisions quickly with no time to set up and solve complex mathematical models. For these time-limited engineers, ANSYS offers a choice of rapid flow modeling (RFM) products. RFM technology from ANSYS compresses the overall time it takes to do a fluid flow analysis by providing a high level of automation and focusing on only the most robust physical models. Three RFM tools are available: FloWizard, ANSYS CFX-Flo and FLUENT for CATIA® V5 software.

FloWizard software integrates all steps in the fluids process into one smooth interface. Computer-aided design (CAD) files can be sent to the FloWizard product, flow volumes extracted, models set up, calculations completed and HTML reports generated. FloWizard software is fully compatible with the FLUENT product, making it a good choice for designers in companies that use FLUENT technology in the analysis department.

Industry-Specific Fluids Simulation Tools

Flexibility and generality are important, but sometimes not required for specific applications. In addition to providing general-purpose CFD and rapid flow modeling products, ANSYS makes fluids simulation even more accessible and focused with its industry-specific analysis tools. These products are often called vertical applications because of the way they integrate all the steps for the analysis of a specific type of system into one package. The technologies offer industry-specific functions as well as employ the language of the industry in which they are used.

Turbomachinery is one of the world’s single most successful CFD vertical applications, due to the similarity of the geometry and physics across a broad range of rotating
The turbosystem technology from ANSYS includes custom geometry and meshing tools as well as special modes within the general-purpose fluids simulation tools.

The ANSYS Icepak product is a family of applications focused on electronics design and packaging. In order to improve the performance and durability of electronic boards and other components for optimized cooling systems, the product calculates the flow field and temperatures in electronics and computer systems.

ANSYS POLYFLOW software is focused on the needs of the materials industry, such as polymer processing, extrusion, filmcasting and glass production. It can model the flow of fluids with very complex behavior, such as viscoelastic fluids. The ANSYS POLYFLOW product offers unique features such as the ability to perform reverse calculations to determine the required die shapes in extrusion. It also can calculate the final wall thickness in blow-molding and thermoforming processes.

The ANSYS Airpak product is aimed at the design of heating, ventilation and cooling systems in buildings, such as offices, factories, stadiums and other large public spaces. It accurately and easily models airflow, heat transfer, contaminant transport and thermal comfort in a ventilation system.

Finally, end-users can create their own vertical applications within the general-purpose fluids simulation products: ANSYS CFX software offers user-configurable setup wizards and expression language; FLUENT technology provides user-defined functions; and the FloWizard tool offers Python scripts. All of these can be used to create custom vertical applications. It is not uncommon for an analysis department to create such vertical applications for deployment within a design department. The main benefit of this approach is to ensure repeatable simulation process control, and hence quality control, for any CFD process.

The extrusion of a viscoelastic food material is simulated with ANSYS POLYFLOW software. The pressure drop between the inlet and the five outlets is shown. The outlet shape is computed as part of the analysis.

FLUENT for CATIA V5 software works within the CATIA V5 PLM environment, as shown in this simulation of a heat exchanger.

The Future of Fluids Simulation from ANSYS

To help customers replace more and more of their traditional capital-intensive design processes with a Simulation Driven Product Development method, ANSYS will continue to innovate and integrate.

In the very near future, users will see tremendous progress toward the ANSYS integration vision, including common geometry, meshing and post-processing tools for all users of CFD products from ANSYS. Many steps in the fluids simulation process will be automatically recorded, enabling parametric simulations. Improvements in fluid–solid connectivity will be evident, enabling a number of new multiphysics possibilities.

The upcoming ANSYS 12.0 release will lay a firm foundation for the future while carefully preserving and extending current software value. Over time, ANSYS plans to achieve the tightest possible integration of all its fluids technologies as well as an intimate integration with ANSYS mechanical technologies. The goal is to combine the best of the best into a simulation system with unprecedented power and flexibility.
Simpler is better — that’s what we’ve all been told. The more complicated something is, the more ways there are for it to break. This seems logical and is something we should consider as we invent new machines. The challenge is that simple machines do simple things and often can only do one thing well. A simple bottle opener, for instance, probably isn’t the best tool for anything other than opening bottles, but it does what it was designed to do. Complicated machines — both mechanical and biological — have more parts, and often can be used to do more than one thing. As an example, the adult human body typically has 206 bones and can be used for all kinds of things from opening bottles to competing in triathlons. Inventing machines that can do a variety of things requires that the machines have multiple parts that work together, preferably without failing. Simulation tools in the product portfolio from ANSYS help make designing useful machines easier and faster, as well as more fun.

**Joints**

When machines were simpler, there were fewer options, and multiple parts could be connected in mechanical software from ANSYS only using shared nodes, beam elements, coupling, constraint equations and node-to-node contact. These methods were adequate for many years, but eventually general surface contact was released to address the limitations. With this new functionality, parts undergoing large rotations, deformations, sticking, sliding and a host of other real-world behaviors could be modeled.

General surface contact became popular and widely used. It also became more robust and efficient with each successive ANSYS release for mechanical applications and is now considered mature, proven technology. One problem with the widespread use of general surface contact, however, is that sometimes it is more than is required. The relatively new capability to connect parts via joints has some potentially huge advantages that can be applied to many situations.

Joints were first released with the COMBIN7 element, which was used to model only pinned, or revolute, joints. At ANSYS 10.0, major advances to joint technology were made via the MPC184 element, which could be used to model multiple joint types, such as those that are translational, cylindrical, spherical, slot, universal, general or fixed. Joint elements are particularly interesting to those involved with the design of multipart machines because they can be used to enable large rotations and translations between parts at a very low computational expense. To illustrate the potential computational savings of using joints, a metal hinge is used as an example. (Figure 1.)
There are many ways to set up a model for a metal hinge, but the two used in this investigation are a traditional general surface contact approach and a revolute joint approach (Figure 1). To simplify, the parts are set to be rigid so that problem size changes can be compared more easily. For each approach, a single CPU laptop is used to run the simulations.

In the general surface contact approach, to enable rotational freedoms but constrain all translations except one, three contact surfaces are required (Figure 2) and one remote displacement, which rotates the hinge 90 degrees counter-clockwise. Using a few user-defined mesh specifications for surface contact size (body and edge sizing), the problem consisted of 7,188 elements (Figure 3) and took 2,249 seconds to solve.

By changing from a general surface contact approach to a revolute joint–based approach, there are three rigid parts and two joints connecting those parts to each other at the hinge: one revolute joint between the ear and the pin, and one fixed joint between the base and the pin. The pin could be suppressed since it won’t perform any function once it is replaced with a revolute joint, but it is included in the model to make the run-time comparison equivalent with the general surface contact approach. The total problem size, as expected, is far smaller, uses only 14 elements (Figure 4) and requires a solution time of only 1.625 seconds.

So what have we learned? First, if detailed contact information at the hinge pin is unimportant, it is a lot more efficient to replace thousands of contact elements with a single revolute joint element. Doing that, the model can be solved in a fraction of the time it took to solve without the use of joints. Second, as can be seen from the element listing in Figure 4, even in a model in which contact surfaces are not specified, there are still contact elements — which come from use of the joint or MPC184 element — but far fewer of them.

<table>
<thead>
<tr>
<th>TYPE</th>
<th>NUMBER</th>
<th>NAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>MASS21</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>MASS21</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>MASS21</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>CONTA176</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>TARGE170</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>CONTA174</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>TARGE170</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>TARGE170</td>
</tr>
<tr>
<td>9</td>
<td>178</td>
<td>CONTA174</td>
</tr>
<tr>
<td>10</td>
<td>180</td>
<td>CONTA174</td>
</tr>
<tr>
<td>11</td>
<td>178</td>
<td>TARGE170</td>
</tr>
<tr>
<td>12</td>
<td>576</td>
<td>CONTA174</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>CONTA176</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>TARGE170</td>
</tr>
<tr>
<td>15</td>
<td>832</td>
<td>CONTA174</td>
</tr>
<tr>
<td>16</td>
<td>1408</td>
<td>CONTA174</td>
</tr>
<tr>
<td>17</td>
<td>1408</td>
<td>TARGE170</td>
</tr>
<tr>
<td>18</td>
<td>288</td>
<td>CONTA174</td>
</tr>
<tr>
<td>19</td>
<td>832</td>
<td>CONTA174</td>
</tr>
<tr>
<td>20</td>
<td>288</td>
<td>CONTA174</td>
</tr>
<tr>
<td>21</td>
<td>832</td>
<td>TARGE170</td>
</tr>
</tbody>
</table>

Figure 1. Hinge model

Figure 2. For this hinge model, general surface contact joints are used in three locations. First, where the ear meets the base, frictionless surfaces prevent translation along the axis of the pin and still allow rotation of the ear and base against each other at the joint. Second, bonded surfaces between the pin and the base prevent the pin from spinning or translating relative to the base. Then lastly, frictionless surfaces between the ear and the pin allow the ear to rotate freely about the pin.

Figure 3. Element description for hinge joint modeled with general surface contact

Figure 4. Element description for hinge joint modeled with a revolute joint
The ANSYS Rigid Dynamics module, first released at Version 11.0, makes extensive use of joints for connecting parts. This is an ANSYS Workbench add-on tool for users who have ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics licenses. The module enhances the capability of those products by adding an explicit solver that is tuned for solving purely rigid assemblies. As a result, it is significantly faster than the implicit solver for purely rigid transient dynamic simulations. The ANSYS Rigid Dynamics module also has added interactive joint manipulation and ANSYS Workbench Simulation interface options.

Interactive joint manipulation allows the user to solve a model essentially in real time — the explicit solver produces a kinematic solution with part positions and velocities — using the mouse to displace the parts of the model. This tool is on the menu bar in the Connections folder. New Configure, Set and Revert buttons can be used to exercise a model that is connected via joints, set a configuration to use as a starting point or revert back to the original configuration as needed. In the case shown in Figure 5, before finding a solution, the hinge has been rotated a little more than 46 degrees to verify that the joint is, in fact, behaving like a hinge.

The ANSYS Rigid Dynamics module is run using the same techniques that are used in ANSYS Workbench Simulation — attaching to the CAD or the ANSYS DesignModeler model, using the model tree, populating the Connections folder and inserting New Analysis, for example.

The combination of the explicit Runge-Kutta time integration scheme and a dedicated rigid body formulation creates a product that while limited to working only with completely rigid parts,

Figure 5. Interactive joint manipulation is possible within the ANSYS Rigid Dynamics module, performed on a computer screen by using the mouse to move the model.

Figure 6. Folding arms of John Deere agricultural sprayer model to be subjected to time-history loading

Image courtesy Brendan L. Stephens, John Deere
is extremely well suited to solving multi-jointed assemblies, such as the folding arm agricultural assembly (Figure 6). This scheme is adept at handling complex time–history input (Figure 7) and is extremely fast compared to more traditional solvers. Solve time, even for complex assemblies, is typically measured in seconds and minutes rather than in hours and days. One caveat worth mentioning is that, at release 11.0, parts need to be connected with joints rather than contact when using the ANSYS Rigid Dynamics capability. If contact is required to accurately represent the part interactions, flexible dynamics simulation is required.

The ANSYS Rigid Dynamics tool should be used first on any complex, multi-part assembly with connections. Fast solution times can help users quickly find joint definition problems, inadequate boundary conditions, over-constraints and other problems. With the time saved, multiple design ideas can be analyzed in the same amount of time that it previously would have taken to simulate a single concept.

**ANSYS Flexible Dynamics**

Is the ANSYS Rigid Dynamics tool all that is needed to fully understand a prototype of a machine? What happens if the parts deform? Will they break? Will they fatigue and fail after a short time or only after extreme use? If parts bend, twist and flex, will the machine still perform its intended function?

The ANSYS Rigid Dynamics capability, for all its strengths, doesn’t provide a complete picture of a machine’s performance. In a thorough machine prototype investigation, the next step is a flexible dynamics analysis, which allows some or all of the machine’s parts to behave as they would in the real world — flexing, twisting and deforming. Flexible dynamics allows users to examine parts to identify whether they are stiff and light, as they would be if made from titanium, or heavy and flexible, as they would be if made from rubber.

A more in-depth explanation of the use of ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics products running flexible nonlinear dynamics simulations is necessary to demonstrate the steps required to take an all-rigid dynamics model and turn it into a partially or completely flexible model. This translation from a rigid to a flexible model includes material assignment, meshing and solver setup. Without writing a spoiler to any future articles on this subject, this is remarkably easy to do.

The simple machines have already been invented. We don’t really need a more efficient bottle opener. With the addition of more realistic and faster modeling solutions — achieved by combining the ANSYS Rigid Dynamics module and ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics software — complicated machines can be less prone to failure and produce fewer career-limiting disasters.
Nonlinear Simulation Provides More Realistic Results

When parts interact and experience large deflections and extreme conditions, nonlinear technology is required to simulate real-life situations.

By Siddharth Shah, Product Manager, ANSYS, Inc.

In today’s competitive environment in which everyone strives to develop the best design with the best performance, durability and reliability, it is unrealistic to rely on linear analysis alone. Analyses must be scaled from single parts and simplified assembly-level models to complete system-level models that involve multiple complex subassemblies. As more parts get added to a simulation model, it becomes more difficult to ignore the nonlinear aspects of the physics, and, at the same time, expect realistic answers.

In some situations involving single- or multiple-part models, analysis with linear assumptions can be sufficient. However, for every assumption made, there is some sacrifice in the accuracy of the simulation. Ignoring nonlinearities in a model might lead to overly conservative or weak design in certain situations, or might result in the omission of unexpected but valuable information about the design or performance of the model. It is essential to understand when and when not to account for nonlinearities. The following are some situations in which nonlinearities are commonly encountered.

Contact
Currently, auto-contact detection in ANSYS Workbench Simulation allows users to quickly set up contact (part interactions) between multiple entity types (solids, sheets, beams). However, in cases in which two parts interact with each other, the parts might stick or slide against each other instead of remaining static. Also, their stiffness might change depending upon whether they touch each other or not, as is seen with interference or snap-fit cases. Ignoring sliding may be acceptable for a large class of problems, but for those with moving parts or that involve friction, it is unwise to make this assumption.

Geometry
In certain situations, the deflections of a structure may be large compared to its physical dimensions. This usually results in a variation in the location and distribution of loads for that structure. For example, consider a fishing pole being bent or a large tower experiencing wind loads. The loading conditions over the entire body of the structure will change as the structure deflects. Also, in certain slender types of structures, membrane stresses may cause the structure to stiffen and, hence, reduce displacements. One example of this is fuel tanks used for satellite launchers and spacecraft. If accurate displacements are to be computed, geometry nonlinearities have to be considered.

Material
Material factors become increasingly important when a structure is required to function consistently and reliably in extreme environments — such as structures that must operate at high temperatures and pressures, provide earthquake resistance, or be impact-worthy or crash-worthy. Plastics, elastomers and composites are being used as structural materials...
with increasing frequency. These materials do not follow the linear elastic assumption of stress–strain relationships. Structures made with these materials may undergo appreciable changes in geometric shape before failure. Without accounting for this material behavior, it can be impossible to extract meaningful and accurate information from their simulations.

In the past, nonlinear analysis was associated with heavy investment in training, resources and manpower. For many, the software appeared cumbersome, challenging and intimidating. It was acceptable and often preferable to get by with physical testing alone. That is not the case today, however. Nonlinear structural simulation is no longer an intimidating tool, but rather one that ANSYS has made available to all engineers by fusing its complex physics into an easy-to-use interface in the ANSYS Workbench environment.

"Through the ANSYS Workbench platform, we have a tool that allows us to increase the performance of our products. Drastic reductions in weights and inertia of the couplings have been achieved without compromising the strength of the unit. Lateral vibration of couplings is now being estimated to a level of confidence previously unattainable without days of computation and cost."

— Ron Cooper, Technical Director
Bibby Transmission, U.K.
Thermoelectric coolers (TECs) serve as small heat pumps, utilizing semiconductors for the cooling action in an enclosed package without any moving parts. Because of their quiet operation and small size, the devices are used extensively for spot-cooling electronics in aerospace, defense, medical, commercial, industrial and telecommunications equipment. In the extreme environments found in satellites and space telescopes applications, TECs often are stacked on top of one another to achieve the required cold-side temperatures. The traditional multistage configuration is pyramidal in shape, with the unavoidably tall profile posing packaging problems in applications with limited vertical space.

To address these issues, Marlow Industries developed an innovative new planar multistage TEC (patent pending) that reduces overall device height by arranging the thermoelectric elements side-by-side in a single plane, instead of stacking them. Because this configuration radically changed the structure, engineers used ANSYS Multiphysics software in evaluating the thermoelectric (TE) performance and thermomechanical stresses of the device, enabling the company to meet critical deadlines for launching the new product in a competitive market.

By Robin McCarty, Senior Engineer for Product and Process R&D, Marlow Industries, Texas, U.S.A.

Thermoelectric coolers serve as small heat pumps, utilizing semiconductors for the cooling action in an enclosed package without any moving parts.
The company selected the ANSYS Multiphysics product because it is recognized as the only commercial finite element analysis package with the capability to model 3-D thermoelectric effects with the required level of accuracy. Given the multiphysics capabilities of the software, a fully coupled thermoelectric simulation could be performed, calculating the current densities and temperatures in the TEC considering both Joule heating and the Peltier effect. Marlow engineers used the calculated temperatures from the thermoelectric analysis of the TECs to perform a static structural analysis, which then was used to predict thermal stresses in the thermoelectric materials due to temperature differences in the TEC assembly.

The objective of the thermoelectric simulation was to determine temperature distribution throughout the device. For creating the analysis model, a constant temperature condition was applied to the bottom of the mounting solder, and a radiation boundary condition was applied to the cold-side ceramic. A heat load (simulating the heat-producing device to be cooled) was applied to the cold side of the TEC, and a DC current was applied to the TEC’s electrical terminals to drive the thermoelectric cooling. From this coupled-physics simulation, the minimum cold-side temperature, temperature uniformity of the top stage, voltage drop and electrical resistance of the TEC were determined.

Once the temperature distribution of the TEC assembly was calculated from the thermoelectric model, it was applied to the TEC assembly in a static structural analysis. To mimic the TEC’s mounting conditions, the solder on the

How a Thermoelectric Cooler Works

A thermoelectric cooler operates based on a principle known as the Peltier effect, in which cooling occurs when a small electric current passes through the junction of two dissimilar thermoelectric materials: a “p-type” positive semiconductor with a scarcity of electrons in its atoms and an “n-type” negative semiconductor with an abundance of electrons. Current is carried by conductors connected to the semiconductors, with heat exchanged through a set of ceramic plates that sandwich the materials together.

When a small positive DC voltage is applied to the n-type thermo element, electrons pass from the p- to the n-type material, and the cold-side temperature decreases as heat is absorbed. The heat absorption (cooling) is proportional to the current and the number of thermoelectric couples. This heat is transferred to the hot side of the cooler, at which point it is dissipated into the heat sink and surrounding environment.
hot side of the device was fixed on the bottom surface. Maximum principal stress was used to evaluate and compare the TEC designs because it can be directly related to the failure of a brittle material, such as bismuth telluride.

The testing team identified the thermoelectric element with the maximum stress and then refined the finite element mesh in that area to ensure that stress convergence had been obtained for the structural simulation. Using a plot of maximum principal stress distribution in a typical TE element, the engineering team found that the maximum stress occurs on the corner of the TE element, which correlated to where Marlow historically had seen cracking in thermoelectric elements that resulted in device failure.

To validate the new planar multistage designs, Marlow evaluated the mechanical stress levels for a thermoelectrically equivalent traditional multistage device and a planar multistage device. Each device consisted of three stages equivalent with thermoelectric element dimensions and thermoelectric element count per stage. In the model, three different currents were evaluated, and the maximum principal stress located in the most highly stressed thermoelectric element was noted.

Through these analyses, Marlow configured planar designs with maximum principal stress levels comparable to the traditional multistage devices. Thermal performance also was nearly equivalent. The correlation between the stress results for the traditional multistage and planar multistage devices provided confidence in the new planar multistage design concepts. This type of evaluation would not have been possible without the multiphysics simulation capabilities available in software from ANSYS.
In the extremely selective large sailboat market, Perini Navi sailing yachts have long been acknowledged for their high performance in terms of balance, speed, reliability and safety, as well as their aesthetic details and passenger comfort. With 42 yachts launched around the world since 1983 and many more under construction at its shipyards in Italy and Turkey, Perini Navi now meets more than half of the global demand for sailing yachts greater than 50 meters in length — the so-called megayachts. To meet the growing requirements of its customers, Perini Navi has found it necessary to combine the distinguishing and well-consolidated features of its yachts with constant improvement in technical solutions over a wide range of conditions.

In this context, Perini Navi developed a pilot project to take advantage of automatic optimization techniques based on virtual prototyping. The project was born out of Perini Navi’s collaboration with EnginSoft of Florence, Italy, and was concerned with the optimization of the main mast structure for a special series of new heavy cruiser motorsailers ranging in length from 50 to 60 meters (164 to 197 feet). The goal of this effort was to identify the solution best able to minimize the mast’s weight while maximizing strength to avoid specific global and local buckling factors. Accurate modeling using structural simulation offered the possibility of not only a weight reduction of the mast structure but also a large balancing reduction in keel weight, thus increasing the overall sailing performance of the boat. Other important factors that stimulated the project included the potential cost savings associated with using less aluminum for the mast and lead alloys for the keel, an easier product construction due to a reduction in the weight and thickness of the mast plates, greater sailing comfort resulting from the reduction in the boat’s tilting, and overall shorter time to market.

The mast design methodology consisted of first analyzing the global buckling factors on the transverse and longitudinal planes. Starting with models developed by Perini Navi in ANSYS Mechanical software for structural simulation, EnginSoft analysts parameterized the models using the ANSYS Advantage • Volume II, Issue 2, 2008
ANSYS DesignModeler application within the ANSYS Workbench platform. They then analyzed three different models of the entire mast structure — including the mast panels and cross-trees, cable shrouds and stays — with ANSYS Mechanical software.

The first model consisted of a simplified section of the mast and included the cross-sectional area and relative moments of inertia as parameters. The range of values for these parameters, resulting from varying the mast geometry, were then used in the second model, which consisted of the entire mast structure; this model was used to analyze the global buckling of the mast. Since the structure was hyperstatic, or over-constrained, the solution depended on the constitutive equations. Thus, the global buckling model took into account the nonlinearities resulting from high deformations of the mast during the initial preloading stage in which the load is the result of the cable shrouds tightening. This model also allowed the structural coupling between the longitudinal and transverse planes. Finally, to analyze local buckling phenomena, a third model — a shell model of the mast section — was created. To carry out the analysis of both global and local buckling, the preloading of the shrouds was taken into account, first by a preliminary static analysis, then by a linear buckling analysis.

EnginSoft engineers developed the procedures to integrate the three ANSYS structural models with the associated optimization process using the modeFRONTIER™ multi-objective optimization software platform from ESTECO. Using modeFRONTIER, the analysts automatically processed the same geometric parameter variables that mast designers would normally be responsible for based on their experience.

This integration process had an impact on Perini Navi’s previous design methodology because it imposed a set of parameters under strict requirements; however, the process was able to exploit the entire potential of software from ANSYS through the ANSYS Workbench interface. The ANSYS Workbench environment allowed for the robust regeneration of the numerical model according to the variation of the input parameters, the automatic application of boundary conditions, the automatic recognition of connections between the cable shrouds and mast, and the ability to carry out advanced visualization and reporting using pre-processing and post-processing features.

The results of virtual prototyping, achieved in terms of both effective weight reduction of the mast and savings of time and materials during machining, led to a decision by Perini Navi to widen its ANSYS Workbench applications to other design protocols, such as the verification of welded joints. The advantages to Perini Navi were notable, as EnginSoft engineers were able to reduce the mast weight by 3 to 5 tons, or approximately 20 to 25 percent. They also could then apply the structural model within the ANSYS Workbench environment to different masts for other boats. Furthermore, the model can now be detailed and adjusted to different load configurations, such as open sail or heeling, to benefit both the efficiency of the development process and the reliability of the final product.
Predicting Charged Particle Trajectories

Simulation helps researchers obtain electromagnetic field solutions for predicting charged particle trajectories in a wide range of industrial, medical and research applications.

By Andreas Hieke, CEO, GEMIO Technologies, Inc., California, U.S.A.

The origins of charged particle optics date back to the mid-1800s, when some of the first studies examined the effects of electric currents in gases, and later when cathode ray tubes were studied. A formal theory of charged particle optics — namely, using electric and magnetic fields to accelerate and guide electrons for imaging applications — was developed in the 1920s, providing the theoretical foundation for the first electron microscopes in the 1930s.

Today, modern applications based on charged particle optics span a wide range of instruments and devices, including electron microscopes, electron-beam lithography machines, vacuum/pressure gauges, x-ray tubes, free electron lasers, electron tubes and related radio-frequency (RF) devices, field emitter–based displays, high-power electron beam guns, ion traps, mass spectrometers, and ion sources. Such instruments are indispensable for building and testing the latest integrated circuit chips, creating advanced medical imaging equipment, and finding molecular biomarkers for the prediction and treatment of human diseases. Specifically, due to the advent of ion sources for biomolecules and the resulting increased importance of mass spectrometry in life science, the field of charged particle optics is experiencing a healthy rejuvenation [1].

A growing number of these applications are based on ion optics using particles much larger and heavier than electrons. Therefore, researchers must account for far greater mass-to-charge ratios that exceed the ratio relevant for electron optics by factors ranging from approximately 2,000 for hydrogen ions (protons), more than $10^8$ for very large macromolecules. Regardless of the types of charged particles used, accurate solutions for the electric and electromagnetic fields in these applications are of critical importance [2].

For most conventional configurations in charged particle optics, analytical mathematical expressions have been developed for determining the fields and particle trajectories. For example, in an electric mirror time-of-flight configuration (also referred to as reflectron TOF), the trajectory of an ion has an analytical mathematical solution. These mathematical expressions are extremely valuable in understanding how variations in key parameters affect the operation of such devices.

Simulations based on finite element analysis (FEA) may be used in cases in which no exact, or even approximate, mathematical expression exists, or when additional physical effects acting on the particles or the device itself must be taken into account. Numerical simulation is also useful in accounting for imperfections due to variations in manufacturing, thermal expansion, mechanical stress, or aging of field-generating components, such as
electrodes, isolators and current-carrying conductors. Such variations usually break certain symmetries in the device or particular spatial ratios that are frequently assumed in the mathematical expression. In these cases, simulation is usually the only option that can provide more depth of understanding for the situation. For many practical problems, the electromagnetic simulation capabilities provide field solutions for extremely complicated geometries with sufficiently high accuracy.

Charged particle trajectory simulations based on these field solutions need to satisfy different and frequently conflicting requirements. The experienced computational physicist, therefore, must carefully balance these demands and choose the appropriate numerical methods and techniques for the simulation.

Among such constraints are:
- Degree of spatial accuracy of the final position of a trajectory
- Long-term stability of a solution (trajectory)
- Execution speed of the computation (the ability to simulate a very large number of trajectories using techniques such as Monte Carlo simulation)
- Degree of smoothness of a trajectory [3]
- Relevance of coupled effects (particle-field coupling such as space charge, induced currents, etc.) and other more advanced effects, usually relevant only in high-energy physics applications

Finite element model of quadrupole tip region

Isosurface plots of electric field

In one recent study of a complex trajectory problem, physicists at GEMIO Technologies, a company that utilizes advanced numerical methods to design ion optical devices for life science applications, used ANSYS Emag capabilities for the simulation of an RF quadrupole, a system of four parallel rods that can act as a guide or filter for charged particles. Very good mathematical approximations for the electric field inside an infinitely long quadrupole can be developed, and stability conditions for trajectories can be derived [4]. However, no accurate mathematical relationships are available characterizing the electric field and particle behavior at the tip of a quadrupole. As a result, researchers are heavily dependent on simulation at that critical location.

In this study, first the researchers created a finite element model representing the 3-D geometry of the quadrupole tip region. They used the simulation results to generate 3-D isosurface plots of the electric field resulting from applying various potentials to the rods. The simulation also provided information about the electric potential and magnitude of the electric field in a cross section of the quadrupole. The numerical values obtained from the simulation solution are remarkably accurate and require little numerical effort while using fairly coarse discretization. At the axis of the quadrupole, the predicted potential

Electric potential (left) and electric field magnitude (right) for a cross section through the quadrupole
has a relative error of less than $10^{-8}$, while that of the predicted electric field is only $10^{-6}$ relative to the maximum value inside the quadrupole.

A particular advantage to having a numerical simulation model is the ability to test the response to certain conditions that are, effectively, impossible to study in most experimental setups. In addition, such ideal conditions provide a method to test the accuracy of the simulation, because frequently the expected result is known. In the case of an RF quadrupole, for example, a particle that is perfectly orthogonally injected into the multipole with certain suitable initial conditions (location, speed and timing) will remain on a stable, periodic trajectory forever. A numerical simulator must be able to replicate such particle capture effects and provide stable long-term solutions. This is somewhat comparable to orbital injection of a satellite around a planet — although gravitational forces are only attractive and obviously static in nature, whereas charged particle capture in multipoles requires RF electric fields, and the resulting trajectories are of more complicated shapes compared with the elliptical orbits of satellites.

Based on certain values for these conditions, the GEMIO team computed the trajectory of an orthogonally injected particle. For this analysis, the quadrupole operates at RF frequencies on the order of 1 MHz with potentials of several hundred volts. Researchers executed the computation in an external module written in FORTRAN™ and based on the field solutions obtained with ANSYS Emag technology. They then imported the resulting data back into software from ANSYS for post-processing.

The particle trajectory simulation indicated that the particle remains on a nontrivial but stable trajectory in a plane orthogonal to the quadrupole axis for a few orbits but, in some rare instances, is suddenly ejected from the plane. From a slightly different angle, the simulation indicated tetrahedrons of nodes in which disturbances of the trajectories occur. Following careful investigation of this unexpected effect, the researchers determined that it was caused by an unusual numerical problem in the external FORTRAN module that executed the trajectory integration. After the team corrected this numerical issue, the effect no longer occurred, and subsequently the team has used this approach to obtain stable trajectories for at least $10^5$ RF cycles.

As this study demonstrates, ANSYS Emag simulation features are capable of providing electric and magnetic field solutions with very high accuracy, making these solutions suitable as a basis for different types of charged particle trajectory simulations in a wide range of applications. As always, the numerical behavior of such tools must be carefully evaluated and tested to gain trust in the results. If executed correctly, charged particle simulations provide a valuable solution that substantially reduces physical testing in the development of these systems. Furthermore, simulation continues to represent the only effective method to study configurations for which no analytical mathematical expressions are available.

Dr. Andreas Hieke is an internationally recognized expert in computational physics and the founder of GEMIO Technologies. The author welcomes reader feedback and may be contacted at ahi@ieee.org or ah@gemiotech.com.

References

The supercomputing power of high-performance computing (HPC) is now available on a Microsoft® platform with Windows® Compute Cluster Server 2003. Work more efficiently with ANSYS® engineering simulation solutions in a familiar Windows environment.

Microsoft is taking HPC further with easy cluster management, enhanced collaboration, and common client and cluster development tools.

**Great performance. Better efficiency. Accelerated time to insight.**
**The next generation of HPC is here.**
Environmental concerns have made reducing water consumption a requirement in many countries. Domestic water usage in toilets can be a significant contributor to overall consumption. A family of four makes an average of 20,000 toilet flushes per year, representing approximately 120,000 liters of water. To cope with the environmental demands of treating large amounts of sewage, legislation in many areas has become more restrictive with regard to water used per flush. This has led to the need to optimize the performance of toilet flushing devices.

With this in mind, engineers at the Department of Mechanical Engineering of the University of Coimbra and the University of Aveiro in Portugal helped Oliveira & Irmão S.A, a manufacturer of toilet flushing valves, to improve the performance of their products. Their collaborative effort focused on increasing the average water discharge velocity in order to optimize the washing efficiency in the toilet bowl. To achieve their goals, they used numerical simulation coupled with experimental validation and performed several shape studies aimed at achieving higher instantaneous flow rates.

As a first effort in the optimization process, the researchers employed steady-state computational fluid dynamics (CFD) simulations in order to estimate the pressure drop across the discharge valve. The team used ANSYS CFX software to perform their CFD analyses. In order to compute the volumetric flow rate, they analyzed the geometries from the initial simulations — which had exhibited lower pressure drops and, therefore, higher flow velocity — with full three-dimensional, unsteady, gravity-driven, free surface simulation models.

The challenge of this project was to determine the best modeling approach for the physical problem. For a reliable numerical estimate, a good definition of the free surface was crucial. The team carefully tested parameters such as boundary conditions, meshing types, modeling schemes and convergence criteria to find the best solution. Grid dependence studies showed that a blend of structured and unstructured meshes presented the most accurate results. The engineering team employed an unstructured mesh near the valve, but in the rest of the cistern where a detailed description of the water free surface is crucial, they adopted a structured mesh. They connected the two mesh types in regions of low-velocity gradients using a generalized grid interface.

The transient simulations used the compressive discretization scheme available in ANSYS CFX software for the volume fraction, so that the free surface would be resolved as sharply as possible as the simulation progressed. The k-ω-based shear stress transport (SST) model was chosen to capture the effects of turbulence. Experimental validation confirmed the fidelity of the numerical model, with relative errors below 5 percent when compared with experimental data. As a result of these efforts, the analysis team determined that ANSYS CFX software was a reliable tool for the simulation of this type of gravity-driven, free surface flow.

This work was funded by Fundação para a Ciência e a Tecnologia, Portugal, through the Research Project POCT/EME/46836/2002.
Adapting A High-Speed Train to Winter in Russia

Simulation helps prevent traction problems for rail travel during extreme weather conditions.

By Philipp Epple, Stefan Becker, Časlav Ilić, Branimir Karić, Irfan Ali and Antonio Delgado, Institute of Fluid Mechanics, Friedrich-Alexander University, Erlangen-Nuremberg, Germany
Uwe Endres, Burkhard Halfmann and Sven Pook, Siemens AG, Erlangen, Germany

When operating a high-speed train in the winter, one thing certain to be encountered is snow. At such high travel velocities, snow can swirl up and accumulate in the cooling air channels of the individual cars. This buildup of snow can be so large that the outside air taken in to cool the cars is reduced or the intake duct completely blocked. As a consequence, there can be a rise in temperature in the cars and under the floors that must be alleviated. If the temperature under the floors gets too high, the result can be failure of traction system components such as the main air compressor, traction converter, traction converter cooler, exhaust air unit and cooling fan of the traction bogie (the chassis carrying the wheels), which can lead to shutdown of the train.

At Siemens AG in Germany, designers faced an extreme snow buildup challenge while engineering the Velaro RUS, a 10-car, 600-passenger train built in Germany for operation at speeds of up to 155 mph (250 km/h). To solve the snow problem faced during Russian winters, Siemens installed an air intake on the roof to cool the traction components under the floor panels. In the summer, ventilation grates at the side slots of the under-floor components are opened so that some of the cooling air can be drawn in laterally. When the snow starts to fall, however, these grates are closed, which reduces the total cooling air flow by about one-third.

To completely seal the air flow in such a way that it can properly cool the traction components and also prevent snow from building up between the components, engineers designed a tray that encloses the traction component system. On most high-speed train cars, the traction components are fixed to the chassis and form the lower part
of the car. On the Velaro RUS, the tray is fixed rigidly to the chassis, while the components are connected to the chassis elastically.

The ventilation system itself combines two main strategies. In the first strategy, the air is drawn in from above the roof and diverted to the lower floor panel. In the second, the air is distributed through the panel using a series of fans to direct the air at the components that need extra cooling.

In order to examine the cooling system of the Velaro RUS, researchers at the computational fluid dynamics (CFD) lab at the Friedrich-Alexander University, Erlangen-Nuremberg decided to use software from ANSYS, including ANSYS ICEM CFD tools to generate the grids and the ANSYS CFX product to perform the CFD computations. To ease the mesh generation process, university researchers divided the geometry into eight sections; the tetrahedral meshing capability of the ANSYS ICEM CFD tool was able to generate meshes even for the most complex geometry details. The research team then assembled the eight meshes employing general grid interfaces and a set of dynamic boundary conditions using the CFX Expression Language.

Researchers were able to specify the total flow rate at the roof inlet; however, they did not know the flow rates at the fans in the lower floor panel in advance. Only the corresponding pressure versus flow rate characteristics were available for the fan, which meant that the fan boundary conditions would need to converge dynamically within the CFD simulation.

The accurate calculation of the mass flow rates at the dynamic fan boundaries was a significant goal for this flow problem. Thus, researchers enabled the solution monitoring feature to observe the convergence of physical quantities beyond the standard solution residuals. The team then initiated the computation using 16 CPUs within an AMD Opteron™ cluster. Only 25 iterations were required to achieve initial convergence of the mass flow rates.

In performing a post-processing examination of the converged results with CFX-Post, the university researchers made extensive use of the CFX power syntax, which is based on Perl™ and the CFX Command Language (CCL). With this approach, they determined precise information about the flow conditions at each traction component, including mass flow rates, pressures, temperatures, velocities and many other physical quantities. The analysis included a study of the temperature distribution for both summer and winter operation. The research group created a series of plane cuts to evaluate the effectiveness of the cooling environment under the floor of the train car. They then were able to easily generate all of the streamline visualizations that were required to provide detailed information about the flow around the traction components, which allowed for a qualitative and intuitive understanding of the cooling air pathways.

In conclusion, CFD software allowed the university research team to manage a complex problem. Simulation contributed significantly to understanding where the cooling efforts were effective and where they were not, which led to adjustments and design modifications that improved the system. With the results from the university study, Siemens was able to confirm the dimensioning of the air intake at the roof and the cooling fans, as well as the operating temperature of the underfloor traction components.
Cerebral aneurysms are arterial outpouchings in the brain that result from weak spots in the vessel wall. Rupture of a cerebral aneurysm can be dangerous for a patient and occurs most commonly between 40 and 60 years of age. When an aneurysm ruptures, blood leaks from the ruptured wall into the subarachnoid space, or the brain itself, potentially causing serious damage. This may lead to major permanent neurological deficits or even death. While it is estimated that approximately 3 percent to 6 percent of the general population have cerebral aneurysms, there is a rupture in only eight in approximately 100,000 person-years. The overall prognosis is poor for the patients with aneurysm rupture — 50 percent will die one month after rupture and an additional 20 percent will be unable to live on their own.

Aneurysm growth and rupture depends on multiple factors: geometrical factors such as aneurysm size and shape or the ratio of the aneurysm dome height to the neck width; biological factors such as decreased concentration of structural proteins of the extracellular matrix in the intracranial arterial wall; and hemodynamic factors, especially wall shear stresses. While patients with unruptured aneurysms may have symptoms such as headache, peripheral visual deficits, loss of balance and coordination, or other neurological deficits (depending on the location of the aneurysm), often cerebral aneurysms are asymptomatic, especially if they are small in size.

When an aneurysm is detected, treatment options other than conservative management include surgical clipping or minimally invasive therapy that approaches and occludes, or blocks, the aneurysm using endovascular techniques. Both treatment options are associated with an overall morbidity and mortality of about 11 percent. Because not all aneurysms will rupture and bleed, it would be highly beneficial to understand the underlying mechanisms that lead to rupture in order to minimize risk to the patient. Currently, there is no medical imaging modality that can provide complete quantitative information about hemodynamic parameters, such as wall shear stresses and dynamic pressure.

In order to better understand the dynamic forces of blood flow within an aneurysm and the conditions that may cause rupture, an interdisciplinary team of researchers at The Methodist Hospital Research Institute (TMHRI) in Houston, Texas, is studying hemodynamics using computational fluid dynamics (CFD) simulations with FLUENT software. More...
specifically, this group has performed unsteady computational fluid dynamics simulations of an anterior communicating artery (AComA) aneurysm and has calculated pathlines and velocity profiles at different time points in the cardiac cycle. The results obtained with the CFD analysis were compared with blood velocity measurements performed using phase-contrast magnetic resonance imaging (pcMRI), a noninvasive imaging technique that allows the visualization of velocity patterns of flowing blood and the quantification of the volumetric blood flow rate.

At first, researchers created a stereolithographic (STL) file of the surface of an AComA aneurysm. This model was based on both a 3-D dataset of the cerebral vasculature of the patient acquired with time-of-flight magnetic resonance imaging (MRI TOF) and a 3-D dataset acquired with digital subtraction angiography (DSA). Afterward, this STL file was imported into GAMBIT software and the model was meshed using 61,663 tetrahedral volume elements.

Researchers then defined inflow boundary conditions as velocity inlets and outflow boundary conditions as pressure outlets. The values of the volumetric flow rate for inflow into the aneurysm were based on the volumetric flow rate waveform measured with pcMRI in combination with the noninvasive optimal vessel analysis (NOVA) system, a technology for the quantification of volumetric blood flow rates. Also, researchers recorded the outflow of the aneurysm and velocity profiles inside the aneurysm at two perpendicular planes, or cross sections, using the same technique.

For the CFD simulation, blood was modeled as an incompressible Newtonian fluid with a density of 1,050 kg/m³ and a viscosity of 0.004 kg/ms. To ensure elimination of initial transients in these unsteady cases, the researchers simulated five cardiac cycles. Results are reported from the fifth cardiac cycle. Mathematical cross sections were defined inside the CFD mesh at the same locations at which the velocity profiles were measured using the NOVA system. The CFD simulation illustrated the pathlines in the aneurysm during average inflow and also showed a swirling motion of the flow inside the aneurysm, a flow pattern that is typical of these vascular pathologies.

The CFD results showed the same main features in the velocity profiles as the ones measured with pcMRI. Regions with opposite directions of blood flow velocity could be identified in both the profiles measured with pcMRI and the cross sections derived from the CFD simulations. These regions reflect the multi-directionality of the blood flow inside the aneurysm, corresponding to the rotational flow motion. The absolute values of the measured velocities were lower than the ones calculated with CFD in both cross sections, a fact that most likely is due to an inherent measurement uncertainty caused by partial volume averaging, in which the MRI signal is averaged over the slice thickness of the scan plane (5 mm), while the CFD cross sections are mathematical planes with zero thickness.

In conclusion, the TMHRI group showed that the velocity values calculated inside a cerebral aneurysm by CFD with patient-specific input boundary conditions were in good qualitative and quantitative agreement with actual velocity values measured with pcMRI and NOVA. These results are encouraging and point to the huge potential of CFD simulations to better understand the hemodynamic effects in cerebral aneurysms that lead to rupture so as to assist in making clinical decisions about the need for surgery.
In my work at Cornell University, I have been using traditional ANSYS Mechanical software for the last seven years to teach students the basics of finite element analysis (FEA) applications. Recently, a few passing encounters with the ANSYS Workbench platform — mostly through students returning from industry stints — piqued my interest in the sleek, modern interface. Two new books published by Schroff Development Corporation helped me cut my teeth on ANSYS Workbench release 11.0: ANSYS Workbench Tutorial by Kent Lawrence and ANSYS Workbench Software: Tutorial with Multimedia CD by Fereydoon Dadkhah and Jack Zecher.

ANSYS Workbench Tutorial

Lawrence’s book kicks off with three chapters on solid modeling using the ANSYS DesignModeler package. The first few tutorials cover solid model creation through extrusion, revolution and sweeping of an L-shaped cross section. Modeling an assembly is demonstrated through the creation of a three-part clevis yoke assembly. It is well worth the time it takes to complete this tutorial successfully, since the book uses this model in a subsequent tutorial on simulation. The solid modeling portion of the book wraps up with exercises on parameters and surface/line models. The ANSYS Workbench interface simplifies working with computer-aided design (CAD) models created in other programs, and Lawrence briefly covers this topic in two separate places. I expect to be referring students to this coverage often.

FEA coverage begins in chapter 4 with an introduction to the ANSYS Simulation module. The book assumes that readers already know the fundamentals of FEA, and it launches straight into the tutorials. The first considers the classic problem of a plate with a central circular hole — a favorite of college professors. This tutorial solves a 3-D plate model with finite thickness using solid elements. Results are verified by comparing with published stress concentration factors and plotting estimates of the structural error, a very useful parameter.

The next tutorial considers the case of a square hole, as opposed to a circular one. The effect of the stress singularity at the corner of the square hole is explored through the automated mesh convergence capability in the ANSYS Workbench environment. This powerful feature successively refines the mesh until a selected solution quantity changes by less than a specified percentage. The author explains how to eliminate the stress singularity by adding a fillet at the concerned corner.

A subsequent chapter deals with the analysis of a cylindrical pressure vessel, an angle bracket and the clevis yoke assembly from the solid modeling section. The clevis yoke example introduces the use of contact capabilities.
Lawrence revisits the plate-with-a-hole, pressure vessel and bracket problems, and solves them using a 2-D or surface model. In each case, results for the solid and simplified models compare well. Other tutorials include analysis of natural frequencies, buckling, heat transfer and thermal stresses.

Instructions in the tutorials are terse, so the reader must peruse both the text and menu snapshots to be able to successfully complete the tutorials, as the two are not redundant. In some cases, the menus on my computer were slightly different from the ones shown in the book, even though I was using version 11.0. With some digging around, however, I was able to compensate for these differences and proceed with the tutorials.

ANSYS Workbench Software: Tutorial with Multimedia CD

While Lawrence’s book focuses on ANSYS Workbench applications, Dadkhah and Zecher’s book integrates discussion of basic finite element modeling concepts with tutorials using the ANSYS Workbench platform. Fundamental concepts considered include stiffness matrices, loads and supports, solid elements, 2-D/axisymmetric models, and mesh refinement/quality. The chapter on stiffness matrices analyzes a system with two springs, so shape functions are not covered.

Discussions about modeling concepts are accompanied by tutorials that focus on illustrating the concepts in ANSYS Workbench applications. For instance, discussion of mesh refinement and quality precedes a tutorial on using convergence to improve results for the normal stress in the fillet region of a T-section. This lets readers see the concepts in action in an actual FEA calculation.

Most tutorials deal with fundamental problems, such as a straight beam, plate-with-a-hole and cylindrical pressure vessel. This allows the authors to check their FEA results against hand calculations or handbook results, which they are very diligent about. This is a sound pedagogical approach to teaching the fundamentals to an FEA novice, and it will help instructors emphasize the importance of verifying FEA results.

Dadkhah and Zecher present tutorials on solid modeling using the ANSYS DesignModeler tool at the start of the book. This material covers some of the same territory as Lawrence’s book. FEA chapters include tutorials on 3-D solid elements, plane stress/strain, shell elements, vibration (natural frequencies and mode shapes) and steady-state heat transfer. More advanced concepts, such as contact, assemblies and fatigue loading, are not considered. Overall, the focus is on understanding the fundamentals through solving canonical problems.

There are a few exercises at the end of each chapter that provide opportunities for further exploration and practice. These would work well as homework problems. The book comes with a CD containing videos of each tutorial in AVI format, as well as model files and a material database used in some of the tutorials and exercises. The book ends with a short chapter on the role of FEA in engineering and issues related to its appropriate use. This chapter raises several issues worth chewing on, such as absolute versus comparative answers, establishing goals for the analysis, interpreting results and performing failure analysis.

In the educational setting, there are two important aspects in learning to use the ANSYS Workbench environment to obtain reliable engineering solutions. First, students need to develop the necessary skills to use the software interface in order to set up and solve a variety of engineering problems. Second, they need to understand the underlying concepts to apply the software correctly in order to obtain validated results. Either book will help students learn the interface, with Lawrence’s book additionally addressing more advanced topics such as contact, assemblies and fatigue loading. Dadkhah and Zecher’s book has more substantial discussion of the underlying concepts and, thus, is more suitable for the novice. Both books certainly are recommended additions to your engineering bookshelf.

For further information on the books ANSYS Workbench Tutorial by Kent Lawrence and ANSYS Workbench Software: Tutorial with Multimedia CD by Fereydoon Dadkhah and Jack Zecher, contact the publisher, Schroff Development Corporation, at 913.262.2664, or visit www.sdcpublications.com.
Today’s undergraduate students are tomorrow’s engineers and researchers. Deploying software from ANSYS, Inc. as an integral part of the engineering curriculum allows academia to train the future workforce with world-class simulation tools and technology and familiarize them with Simulation Driven Product Development processes. It is also critical in that it ensures that academic research continues to push the technology envelope.

ANSYS offers two broad product license categories — commercial and academic — the primary difference between the two being the terms of use. Commercial product licenses are intended for use by for-profit companies and organizations in which the analysis work performed is often proprietary in nature. Academic product licenses, on the other hand, are intended for use by academic organizations, such as universities, for nonproprietary teaching and research.

The differences in these terms of use allow ANSYS to provide academic licenses at significantly reduced cost compared to the commercial licenses, which in turn helps to meet academic budget requirements (Figure 1).

Academic products from ANSYS are also packaged differently from commercial products, with product names and license files that differ from the commercial product portfolio. Academic products are bundles of analysis technology, often incorporating many commercial products and add-on modules, with some containing more than 10 commercial products in a bundle. A single academic product license may contain multiple tasks (such as five, 25 or 50 tasks) in which each task maps to a separate user. With a few exceptions, the academic products are derived directly from the commercial products. For a release such as ANSYS 11.0, both commercial and academic products are included.

In use, the academic products have exactly the same look and feel as the commercial products. For example, a user accessing the ANSYS Multiphysics capability bundled with an academic product will have the same GUI, workflow, pre-processing, post-processing and solver as the commercial product. This helps ensure an easy transition for the user from academia to the workplace. In most cases, the academic user will actually have access to many more features than the average commercial user.

<table>
<thead>
<tr>
<th>Product Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACADEMIC ASSOCIATE</td>
</tr>
<tr>
<td>ANSYS Academic Associate</td>
</tr>
<tr>
<td>ANSYS Academic Associate AUTODYN</td>
</tr>
<tr>
<td>ACADEMIC RESEARCH</td>
</tr>
<tr>
<td>ANSYS Academic Research</td>
</tr>
<tr>
<td>ANSYS Academic Research CFD</td>
</tr>
<tr>
<td>ANSYS Academic Research LS-DYNA</td>
</tr>
<tr>
<td>ANSYS Academic Research AUTODYN</td>
</tr>
<tr>
<td>ACADEMIC TEACHING</td>
</tr>
<tr>
<td>ANSYS Academic Teaching Advanced</td>
</tr>
<tr>
<td>ANSYS Academic Teaching Introductory</td>
</tr>
<tr>
<td>ANSYS Academic Teaching Mechanical</td>
</tr>
<tr>
<td>ANSYS Academic Teaching CFD</td>
</tr>
<tr>
<td>ANSYS Academic Teaching AUTODYN</td>
</tr>
<tr>
<td>ACADEMIC TOOLBOX</td>
</tr>
<tr>
<td>ANSYS Academic Meshing Tools</td>
</tr>
<tr>
<td>ANSYS Academic CFD Turbo Tools</td>
</tr>
<tr>
<td>ANSYS Academic LS-DYNA Parallel</td>
</tr>
<tr>
<td>ANSYS Academic Mechanical HPC</td>
</tr>
<tr>
<td>ANSYS Academic AUTODYN HPC</td>
</tr>
<tr>
<td>ANSYS Academic CFD HPC</td>
</tr>
</tbody>
</table>

Figure 1. The intended use of academic products from ANSYS

Figure 2. The academic products from ANSYS release 11.0. Refer to academic solutions Web page for details on features for academic products: www.ansys.com/academic
The ANSYS 11.0 academic product portfolio includes ANSYS Multiphysics, ANSYS CFX, ANSYS ICEM CFD, ANSYS TAS and ANSYS AUTODYN products, plus a broad selection of computer-aided design (CAD) geometry interfaces (Figure 2). The Fluent academic products are not part of the release 11.0 academic portfolio, but some will be integrated in the ANSYS 12.0 release, creating an academic product portfolio that includes all major technologies from ANSYS. The guiding philosophy is to provide academia with very high value bundles of analysis technology, negating the need to purchase multiple products while reducing complexity and improving scalability.

Academic products from ANSYS are organized into four product subfamilies — Teaching, Research, Associate and Toolbox — with each subfamily having specified terms of use and capabilities. The Teaching subfamily is the lowest priced and includes entry-level products intended for undergraduate and graduate class demonstrations and hands-on instruction. Teaching-level products have numerical problem size limits, which vary by physics, with higher limits assigned to external field physics such as electromagnetics and fluid dynamics (Figure 3). The Research and Associate subfamilies have broader terms of use and can be used for both research and teaching. They have no problem size limits, providing unlimited numerical headroom for doctoral and post-doctoral research work. The Toolbox subfamily addresses high-performance computing (HPC) and specialized pre-processing concerns.

Each academic product can be purchased in defined task increments, with a task defined as a single user. For example, a 25-task license will allow a maximum of 25 simultaneous users. Any combination of these tasks is allowed, and volume discount is built in. The academic product licenses are floating local area network (LAN) licenses, utilizing a single designated server. Since the majority of the academic products are multiple task licenses, this means that all of the tasks are floated on the server’s LAN. The products can be installed on an unlimited number of machines connected to the LAN, but the number of users who can simultaneously access the software is limited by the number of tasks purchased (one, five, 25, 50, etc.).

ANSYS is at the forefront of providing simulation software worldwide for academic users — for both teaching and research applications. Academic products from ANSYS are used by thousands of universities and colleges in more than 60 countries, with hundreds of thousands of users.

The Ansys Advantage • Volume II, Issue 2, 2008
In the classes offered by Professor Lawrence at the University of Texas at Arlington, there is a focus on structural and thermal physics, and students are exposed to both the ANSYS Workbench platform and the traditional GUI interface. Academic products from ANSYS are used exclusively, together with two textbooks authored by Professor Lawrence and published by SDC publications. A typical project for undergraduate students is a reverse engineering task in which they generate part and assembly models of actual devices. A specific example is the analysis of an automotive valve spring compression device (Figure 4). To accomplish this task, students use the ANSYS Workbench platform to perform structural and thermal analyses. In addition to the above classes, academic products from ANSYS are used for research in the following areas: structural response, failure mechanisms in composite structures, and thermal and structural behavior of electronic packages. Professor Lawrence estimates that 2,000 undergraduates and 900 graduate students have been trained since the classes were made available.

For his undergraduate laboratory course, Professor Imas of the Stevens Institute of Technology uses academic products from ANSYS to demonstrate a numerical experiment and expects further integration of academic products during the next couple of years. For his Numerical Hydrodynamics course, the students use ANSYS CFX capabilities to demonstrate and study canonical problems involving turbulence, multiphase flows including free surface and cavitation, and fluid structure interactions. The academic products from ANSYS are used alongside other FEA/CFD tools, and the course is taught from a set of lecture notes together with tutorials. Academic products from ANSYS are also used for free surface hydrodynamics research, focusing on the hydrodynamics of sailing craft, high-speed multi-hulls, surfboards and whale fins (Figure 5). Professor Imas states that approximately 12 graduate students have completed the Numerical Hydrodynamics class and use academic products from ANSYS for research.
Professor Bhaskaran provides his Cornell students with a basic (2-D, linear behavior) introduction to finite element analysis (FEA) using an in-house toolbox built on MATLAB®. The classes then build on this foundation using academic products from ANSYS for real-world FEA. Professor Bhaskaran focuses his classes on structural and thermal physics, both separately and as coupled field effects. Student projects have included analyzing a bicycle crank and comparing the numerical analysis results with those from physical testing. Another project has involved modeling a bucket used on a Caterpillar® hydraulic excavator (Figure 6). In this project, students analyzed the stresses under normal loading conditions and optimized the bucket’s design for increased load capacity, maintaining bucket volume and a safety factor. Professor Bhaskaran has developed a suite of tutorials using ANSYS products, which can be found on the web: http://courses.cit.cornell.edu/ansys. Cornell has also integrated Fluent academic products into their curriculum. Professor Bhaskaran estimates that 600 students have been trained since these classes were made available.

**Workshop**

Cornell University will host an educational workshop titled “Integration of Simulation Technology into the Engineering Curriculum: A University-Industry Workshop” on July 25 to 26, 2008. The primary objective of this workshop is to promote the advancement and sharing of curriculum materials with a focus on simulation technology. More information is available at www.mae.cornell.edu/swanson/workshop2008.

“Software from ANSYS allows us to provide students with a strong foundation in the intelligent and appropriate use of state-of-the-art FEA technology, an important skill for a mechanical engineer.” Professor Rajesh Bhaskaran, Sibley School of Mechanical and Aerospace Engineering, Cornell University, New York, U.S.A.

“The ANSYS Workbench environment allows the students to work with software that is tailored to product engineers.” Professor Jack Zecher, Purdue School of Engineering and Technology, Purdue University, Indiana, U.S.A.

Professor Zecher's computer-aided machine design class at Purdue University focuses on structural physics. A typical project for undergraduate students is to determine the maximum von Mises stress in a support bracket that is subjected to a 500-pound distributed vertical load (Figure 7). The geometry of the part is provided in the form of an initial graphics exchange specification (IGES) file. In this class, students are exposed to the ANSYS Workbench environment and are expected to use the convergence feature of the platform. Professor Zecher uses academic products from ANSYS, along with a new textbook co-authored by Fereydoon Dadkhah and himself. Professor Zecher estimates that approximately 500 students have received CAE training since this class was made available.
Using New Meshing Features in ANSYS Workbench Simulation

Knowing when and how to apply key features of the latest structural meshing tools can result in greater efficiency.

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

As solvers become faster and computers more powerful, the solution portion of finite element analysis shortens and a larger portion of overall simulation time is spent on pre-processing, including generating the mesh — the fundamental element-based representation of parts to be analyzed. Recognizing this trend, ANSYS, Inc. has addressed the need for faster and more reliable structural meshing with new technologies in ANSYS Workbench Simulation 11.0 (also known as ANSYS Workbench Meshing 11.0). These new capabilities result in very robust meshing and save considerable amounts of time (especially for complex geometries) with features that automate many routine tasks while providing users high levels of control of their model.

During the past several years, meshing in ANSYS Workbench Simulation has not only grown to encompass typical meshing algorithms available in traditional structural software from ANSYS but has also included many features requested by its large base of users worldwide. This wealth of new meshing capabilities includes:

- Physics-based meshing and element shape checking
- Higher degree of mesh sizing controls
- Patch independent surface and volume meshing
- Flexible sweep and hexahedral meshing, including automated generation of SOLSH190 solid-shell elements

Physics-Based Meshing and Element Shape Checking

Traditionally, software from ANSYS requires users to select the appropriate element type first; meshing algorithms and conservative shape-checking criteria are typically independent of the physics of the problem. On the other hand, ANSYS Workbench Simulation provides users with the ability to set default global meshing options under the DETAILS view of the MESH branch that is dependent on the analysis physics.

The ANSYS Workbench platform can generate meshes for structural, thermal, electromagnetics, explicit dynamics or computational fluid dynamics (CFD) analyses, but the meshing considerations vary for each. For example, lower-order elements with a finer mesh density tend to be used in CFD analyses, whereas higher-order elements with a coarser mesh density may be preferred in structural analyses.

For each physics, different criteria are used for element shape checking to ensure that the elements provide accurate results for that particular analysis. For mechanical analysis users, STANDARD and AGGRESSIVE shape checking are also available. STANDARD shape checking is suitable for linear analyses, while AGGRESSIVE checking provides more...
Higher Degree of Mesh Sizing Controls

Mesh sizing controls are available in ANSYS Workbench Simulation under the MESH branch, allowing users to specify element size on vertices, edges, faces or bodies (parts), with number of divisions and mesh biasing available on edges. Two features introduced in ANSYS Workbench Simulation 10.0 are SPHERE OF INFLUENCE and CONTACT SIZING.

Under a surface or body mesh sizing branch, instead of specifying a uniform mesh density for the entire geometric entity, a user can use a defined COORDINATE SYSTEM and a radius to designate a sphere where elements will have a certain size. This is helpful in specifying a smaller mesh density without requiring existing geometry to identify that region. The sphere of influence mesh control is also useful in propagating a mesh density inside the geometry, rather than concentrating a finer mesh only on the surface of the model. In analyzing the stress distribution of 2-D contact between gears, for example, a user may create a sphere of influence to generate a fine mesh on the edges of the gear teeth as well as within the geometry of the gear body (Figure 1).

Many users already know that contact regions are associated with the geometry in ANSYS Workbench Simulation, so remeshing an assembly does not require re-creating contact pairs. Another useful related feature is CONTACT SIZING, which allows users to define a more uniform, finer mesh density in a contact region to provide a better distribution of contact pressure. Instead of having users manually select surfaces on which to define element sizes, however, the CONTACT SIZING control allows users to specify mesh densities that are applied only in areas of initial contact for the defined contact regions. Thus, a user only has to drag-and-drop a contact region from the CONNECTIONS branch to the MESH branch and specify an element size — so only the actual areas that are in initial contact will have that finer mesh (Figure 2).

Meshing and Defeaturing

The default volume mesher in ANSYS Workbench Simulation automatically includes some defeaturing, unlike the mesher in traditional software from ANSYS, which meshes all surfaces, including any sliver areas present in the model. The user can control the percentage of defeaturing by specifying the DSMESH DEFEATUREPERCENT variable in the VARIABLE MANAGER located under the TOOLS menu. Consequently, instead of meshing small slivers (which would generate more nodes and elements), ANSYS Workbench Simulation internally ignores these surfaces, providing a more robust, efficient mesh that requires little cleanup of CAD geometry (Figure 3).

While automatic defeaturing is helpful, this built-in defeaturing is not meant to compensate for larger surfaces the user wants to ignore. Instead, either VIRTUAL TOPOLOGY or the PATCH INDEPENDENT MESHER can be used for this purpose.

A VIRTUAL TOPOLOGY branch can be inserted from the MODEL branch. Once added, virtual cells can be defined that effectively merge surfaces and edges for meshing and pre-processing purposes. Although the functionality is similar to concatenation for mapped meshing in ANSYS (ACCAT and LCCAT commands), virtual cells are used for tetrahedral meshing in ANSYS Workbench Simulation, thus giving the user greater flexibility.

Using the METHOD mesh control, a user can also utilize the patch independent meshing algorithm (also named UNIFORM QUAD/TRI or UNIFORM TRI for surface meshing). The patch independent meshing algorithm takes a different approach, as it does not start off with a surface mesh but uses an Octree algorithm instead, so the mesh is not constrained by all of the surfaces present in the model. This algorithm is useful when a user may want to perform gross defeaturing of a very complex part or if a user wants to
generate a uniform mesh. Although the mesher can skip over small features, any scoped surfaces (that is, loads applied on certain faces) will have their boundaries respected. Although VIRTUAL TOPOLOGY can be used in conjunction with the PATCH INDEPENDENT MESHER, the author recommends using either of the following procedures for meshing (Figure 4):

- Use the default patch-conforming meshing algorithm for surfaces/parts, and use virtual cells to group small surfaces to adjacent ones, if needed. This method provides a mesh that conforms to the geometry, although the user can merge together unimportant, small surfaces to reduce the node/element count. This technique is helpful if a lot of manual defeaturing (via VIRTUAL TOPOLOGY) is not required. The user can also set the DEFEATUREPERCENT variable for global defeaturing of very tiny geometric features.

- For bulky, complex geometry whose surfaces the user may not need to mesh in detail, add a METHOD branch to specify the patch independent meshing algorithm and any defeaturing or curvature/proximity refinement that is desired. Scope (via NAMED SELECTIONS, CONTACT REGIONS, LOADS & SUPPORTS and RESULTS) all geometric entities whose features should be kept. This technique is useful if gross defeaturing is required, since manual specification of regions is not required.

Thin Solid Volume Sweeping

The SOLSH190 solid-shell element is a specially formulated eight-node hexahedral element that has special shape functions to prevent locking, even when the thickness is very small. The SOLSH190 element provides a straightforward way to account for variable shell thickness and allows for a natural transition to regular solid elements.

Because of the special shape functions in the thickness direction, the user needs to pay special attention to the SOLSH190 node numbering, the generation of which can sometimes be a cumbersome task in some software from ANSYS. ANSYS Workbench Simulation, however, provides a convenient meshing capability to automatically generate SOLSH190 elements with the proper orientation. For sweep-meshable parts, the METHOD mesh control provides automatic or manual sweep meshing. There is an additional option for THIN MODELS, which generates SOLSH190 elements. Besides automatic detection of source and target surfaces, users can specify multiple source areas and not be limited to a single source area (Figure 5).
The FLUENT computational fluid dynamics (CFD) solver has undergone extensive development to extend its robustness and accuracy for a wide range of flow regimes. Since its initial release, the FLUENT solver has provided two basic solver algorithms: The first is a density-based coupled solver (DBCS) that uses the solution of the coupled system of fluid dynamics equations (continuity, momentum and energy); the second is a pressure-based algorithm that solves the equations in a segregated or uncoupled manner. The segregated pressure-based algorithm has proven to be both robust and versatile, and has been utilized in concert with a wide range of physical models, including multiphase flows, conjugate heat transfer and combustion. However, there are applications in which the convergence rate of the segregated algorithm is not satisfactory, generally due to the need in these scenarios for coupling between the continuity and momentum equations. Situations in which equation coupling can be an issue include rotating machinery flows and internal flows in complex geometries.

The ANSYS CFX solver relies on a pressure-based coupled solver approach to achieve robust convergence rates. ANSYS now offers a similar pressure-based coupled solver (PBCS) for the first time with version 6.3 of the FLUENT software. As its name implies, the algorithm solves the continuity and momentum equations in a coupled fashion, thereby eliminating the approximations associated with a segregated solution approach where the momentum and continuity equations are solved separately. While these approximations do not affect solution accuracy at convergence, they can hamper the convergence rate for certain classes of problems. With the coupled approach, removing the approximations due to isolating the equations permits the dependence of the momentum and continuity on each other. 

Figure 1. Flowchart illustrating FLUENT solver algorithms
other to be felt more directly. This leads to a more rapid and monotonic convergence rate and hence faster solution times. In addition, the coupling leads to improved robustness such that errors associated with initial conditions, nonlinearities in the physical models, and stretched and skewed meshes do not affect the stability of the iterative solution process as much as with segregated algorithms. The coupled algorithm can also be used with a wide range of physical models such as reacting flows, porous media, and many multiphase models, including volume of fluid (VOF) models.

Algorithm Overview

A flowchart illustrating the pressure-based and density-based solver algorithms is shown in Figure 1. This diagram depicts the process by which the pressure-based, segregated algorithm solves the momentum equations, for the unknown velocity components one at a time, as scalar equations and then solves a separate equation for mass continuity and pressure. The pressure solution is used to correct the velocities such that continuity is satisfied. When the flow equations are coupled together, the coefficients that are computed for each equation contain dependent variables from the other equations. In the segregated algorithm, these variables are supplied simply by using previously computed values, which introduces a decoupling error. The decoupling error can delay convergence in cases in which strong pressure–velocity coupling exists.

The pressure-based coupled solver differs from the segregated algorithm in that the continuity and momentum equations are solved in a fully coupled fashion. That is, a single matrix equation is solved in which the dependent variable is now a solution vector containing the unknown velocities and pressures. This is similar to the density-based implicit solver, except that the density-based solver also includes the energy equation in the coupled system and employs a different discretization of the flux terms, among other differences. The tradeoff with respect to computational resources is that the PBCS requires about twice the memory per cell as the segregated algorithm. This is due to the storage required for the coupled (matrix) equations. In general, the storage requirements are comparable to, though slightly less than, the density-based implicit algorithm.

It is important to note that, unlike the density-based schemes, the PBCS does not include the energy equation in the coupled system. This means that the density-based solver may still be preferable for high-speed compressible flow cases, in which coupling the energy equation is important. The PBCS can be used instead for all cases in which one previously would have used the segregated scheme, including low-speed compressible flows, incompressible flows or cases that require models available with only the pressure-based solver.

Using the Pressure-Based Coupled Solver

Activating the pressure-based coupled solver in FLUENT software is very straightforward. First, select the Pressure-Based option in the Define→Models→Solver panel as shown in Figure 2. (Users of older versions of FLUENT software should note that this manner of selecting solver algorithms is new in version 6.3.) Once this option is activated, the user then can select the coupled solver from the Pressure-Velocity Coupling list in the Solve→Controls→Solution panel, as shown in Figure 3.

Activating the pressure-based coupled solver exposes new solver
A Pressure-Based Coupled Solver Example

To demonstrate the efficacy of the pressure-based coupled solver, a standard compressible flow validation case, the RAE 2822 airfoil, was solved using the pressure-based segregated, pressure-based coupled and density-based implicit algorithms in FLUENT 6.3 technology. The airfoil was modeled as an external air flow problem with a freestream Mach number of 0.73 and a Reynolds number based on chord of 6.5x10^6. The 2-D numerical model utilized a quad mesh with 126,900 cells, as shown in Figure 4. The flow was assumed to be steady-state, viscous, turbulent and compressible, with the ideal gas law used for the equation of state. Turbulence was modeled using the realizible k-ε turbulence model with non-equilibrium wall functions. Second-order discretizations were used for all equations. For the pressure-based coupled solver, the CFL (Courant) number was set to 200 and the explicit relaxation factors to 0.5 for both momentum and pressure. Default solver settings were employed for the segregated and density-based implicit algorithms.

The solutions obtained are illustrated in Figures 5 and 6. All three algorithms capture the suction surface shock wave crisply and show excellent agreement with the experimental data for this case. Figure 7 presents a table with the solver performance and resource requirements for each algorithm. As expected, the segregated solver uses the least memory. However, because of the close coupling of the momentum and continuity equations for this case, the data shows that the segregated solver required 2,570 iterations to reach convergence. In contrast, the pressure-based coupled solver required only 298 iterations to converge, which also compares favorably to the 976 iterations required by the density-based implicit solver.

<table>
<thead>
<tr>
<th>Solver</th>
<th>Memory (MB)</th>
<th>Time per Iteration (seconds)</th>
<th>Iterations to Convergence</th>
<th>Time to Convergence (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segregated</td>
<td>172</td>
<td>2.10</td>
<td>2570</td>
<td>1.50</td>
</tr>
<tr>
<td>PBCS</td>
<td>259</td>
<td>3.26</td>
<td>298</td>
<td>0.27</td>
</tr>
<tr>
<td>DBCS</td>
<td>317</td>
<td>3.82</td>
<td>976</td>
<td>1.04</td>
</tr>
</tbody>
</table>
Fluid Dynamics on the Big Screen

Fluid simulations have come of age in the world of movie-making.

Of this year’s ten Scientific and Technical Achievement Oscars®, five were awarded to organizations and individuals who made advances in fluid simulation tools and technology. Rhythm & Hues, one of Hollywood’s top visual effects and animation studios, was one of the winners for a system that allows artists to create realistic animation of liquids and gases using novel simulation techniques for accuracy and speed, as well as a unique scripting language for working with volumetric data. Rhythm & Hues has 115 feature films to its credit, including most recently The Golden Compass, Alvin and the Chipmunks, Evan Almighty, and the upcoming The Incredible Hulk and The Mummy: Tomb of the Dragon Emperor.

ANSYS Advantage magazine interviewed Dr. Jerry Tessendorf, Principal Graphics Scientist, who was one of the four people on the Rhythm & Hues team that was honored.

What is the technological leap in fluid simulation that occurred to warrant five Academy Awards this year?

The visual effects industry has been interested in fluid-like behavior for decades. Around 10 years ago, it was recognized that computational fluid dynamics (CFD) codes could possibly produce interesting behavior that is not available using other methods. But at that time the codes were difficult to use in our production environments because they required high levels of technical expertise, were slow or both. Also, the use of volumetric data in visual effects was just emerging, so the tools for exploiting CFD data were not adequate. Since then, visual effects companies and commercial software vendors have been working to find the right mix of CFD methods that work efficiently in production. The CFD tools have matured sufficiently so that they are now used in many different visual applications, even in scenes with only modest effects.
How are the requirements for simulation in motion picture animation different than for engineering simulation involving real-world problems?

The differences are enormous and primarily relate to the end goal. For motion picture effects, the goal is to create something that satisfies the visual expectations of the director. If the fluid behaves inaccurately, or even unphysically, that is not a problem if it fits within the look and story of the film. Frequently, the story is about an unusual situation, so unusual fluid behavior fits with that theme.

How do the fluid physics engines driving animation software differ from those driving commercial engineering simulation software?

At the core of the simulation code, the differences between our simulation algorithms and those in commercial systems are not that great. We work toward solving the Navier–Stokes equations through a mix of Eulerian gridded methods, Lagrangian smooth particle hydrodynamics methods and some others that are proprietary. The two codes diverge in the way the simulations are accessed or driven. Animation artists work in specialized software packages designed for their specific workflow and have the ability to create imaginative methods of forcing the fluid and applying boundary conditions. Also, there are a number of post-simulation tools available either to alter the simulation results to enhance or suppress behaviors or to use the simulation to drive geometry, particles and volumes. We rarely use raw simulation output in final imagery.

How much time does it take in terms of processing, person-hours, etc., to produce a challenging fluid simulation-based animation?

As with all visual effects elements, simulations are set up, run and revised incrementally over an extended period, because the goals for the simulation evolve during the course of the work. For a single iteration, the artist sets up the simulation conditions (initial fluid state, boundaries, objects, forcing) and sends multiple versions of the simulation conditions to the simulation farm, producing three to 20 variations on a simulation per day. Early in development, setup may take a day to accomplish, but as the work evolves, setup time is reduced to just a few minutes. By running multiple simulations simultaneously, the artist can vary factors such as forcing strength, viscosity, accuracy, etc., over a short period of time. The goal is to have a daily cycle of simulation results, at least for testing and development. Late in the development, the simulations may be run with higher resolution or over more frames if the scene requires it.

What fluid phenomenon is the hardest to simulate for visualization in motion picture animation and why?

We have been able to realistically simulate liquids, gases and fires with our tools. The most difficult problem is to simulate very large regions of fluid at very high resolution. Limited computing resources ultimately limit our ability to simulate big and small scales simultaneously, which is one of the most difficult problems in fluid dynamics as a whole. We continue to improve our tools and hardware in order to simulate greater volume and detail, but we also have evolving tools for the post-processed enhancement of simulations. These tools, implemented in a proprietary scripting language, are based on a combination of physical reasoning and artistic inspiration.

Can you provide an example of a challenging fluid simulation from a recent movie?

You might not expect it, but there was a hard simulation problem in Alvin and the Chipmunks. For a concert scene, the chipmunks performed near and on top of a cauldron with dry ice vapors, which had to be simulated using our CFD system. Since the camera was very close in many shots, we needed to simulate at high resolution, resulting in about one week of simulation time. Also, the chipmunks danced in the vapor, inducing compromises to the simulation stability. In the end, since the effect was a minor priority, we simulated at a lower resolution and then enhanced the simulation output to achieve a nice visual quality for the final result.
Control complexity and complete projects twice as fast
When your applications all run in a single, powerful data environment designed specifically for engineering—projects can move from design to simulation, then on to completion, with ease. That’s why we’ve built our storage and server solutions—including innovative Altix® XE servers based on the Quad-Core Intel® Xeon® 5300 processor—to work together and control complexities, even in the most challenging engineering environments.

Learn more at sgi.com/go/ansysaltix