ANSYS

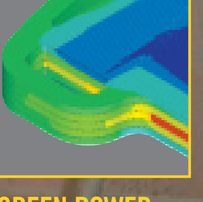
ADVANTAGE

EXCELLENCE IN ENGINEERING SIMULATION

VOLUME I ISSUE 3 2007

HEATING THINGS UP IN THE GLASS INDUSTRY

PAGE 8

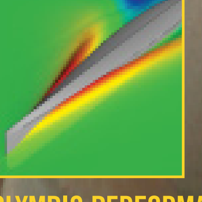


GREEN POWER

PAGE s6

RELIABLE WHEELS

PAGE 14

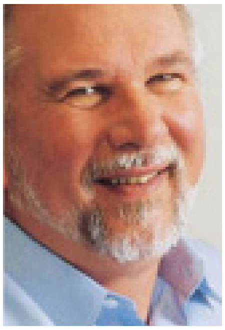


OLYMPIC PERFORMANCE

PAGE 20

Where's the Data?

Companies embracing digital product development must implement tools for better managing simulation information.



One unmistakable global trend is the oncoming wave of digitalization of the product development process. Manufacturers are making greater use of upfront analysis, collaborative tools, digital mock-ups, assembly modeling and complex system simulation. One recent study, "The Digital Product Development Benchmark Report" from the Aberdeen Group, quantifies compelling reasons why companies are moving to a paperless process. Specifically, evaluating and refining designs early in development enables top companies to eliminate an entire prototype cycle, in some cases getting products to market a full three months faster and saving up to $1.2 million in development costs, depending on product complexity.

The study also notes major challenges to digital product development. Topping the list is accessibility to digital product information: that is, how the right people can get to the right data at the right time. And therein lies a big problem that the engineering simulation community has faced since the early days of the industry: Analysis files — including models, results data and the processes that go into the simulation — are not well managed. More often than not, keeping track of this information is left to the individual who generated it, so typically it is buried in obscurity somewhere on a hard drive — or possibly deleted — at the end of a project. Also, this valuable intellectual property may be lost forever when individuals leave the company. Tracking down old data files and analysis models from past simulation projects is difficult and often impossible, even for the people who created them.

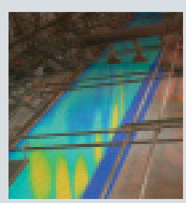
Manufacturers are implementing product lifecycle management (PLM) solutions in record numbers to deal with issues such as this for engineering data and documents across the enterprise. According to statistics from CIMdata Inc., the PLM market grew 10.4 percent in 2006 to reach $20.1 billion; it is forecast to increase at an 8.5 percent compound annual growth rate — exceeding an estimated $30 billion by 2011. Handling computer-aided design (CAD) files, part lists, technical documents and change orders is a lot easier than managing simulation data, however, which is, inherently, a much more demanding task for PLM because of the huge file sizes and the complexity of capturing the context of the simulation.

A major step forward in closing this gap is the development of the ANSYS Engineering Knowledge Manager (EKM) tool. Scheduled for release this year, the Web-based tool will be targeted at managing simulation processes and data along with capabilities for backup and archival, traceability, process automation, collaboration, capture of engineering expertise and intellectual property protection. By managing simulation data and processes within such a framework, companies can more effectively leverage the full power of this critical information and the tremendous expertise of the analysts and engineers who created it. ■

John Krouse, Editorial Director

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

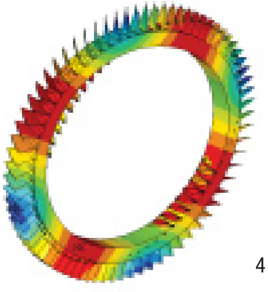
About the Cover
Glass has fascinating and unique properties, and producing it can be a complex undertaking. Read about how PFG Building Glass in South Africa is changing glass-making from an art to a science, on page 8.



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

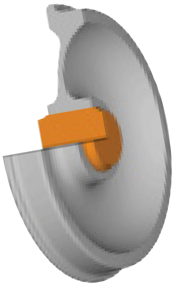
ANSYS Advantage is published for ANSYS, Inc. customers, partners and others interested in the field of design and analysis applications.

Table of Contents

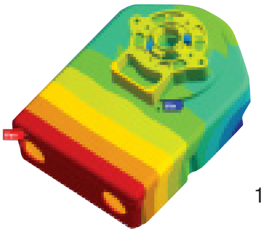


4

6



14



16

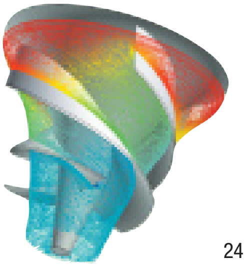
20

ARTICLES

DEPARTMENTS

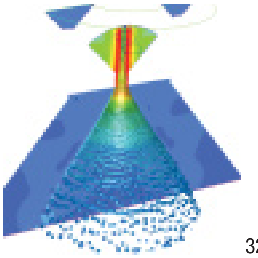
Spotlight on Engineering Simulation for

Environmental Design



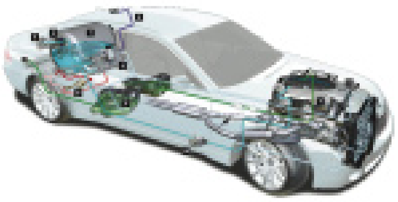
24

26



32

s1

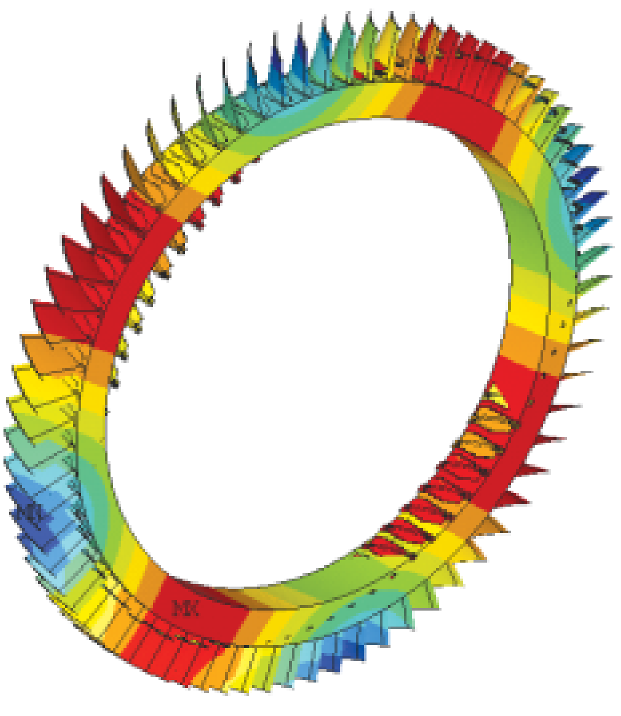


s10

Streamlined
Flutter Analysis

Integrated fluid structure interaction enables high-fidelity turbomachinery blade flutter analysis.

By Robin Elder and Ian Woods, PCA Engineers Limited, Lincoln, U.K.
Simon Mathias, ANSYS, Inc.



ANSYS Mechanical analysis tools can predict vibration modes that occur over an entire wheel from a single blade component model. Shown here are exaggerated deformations for a four-nodal diameter mode shape, meaning that the mode repeats itself four times over the entire wheel circumference. Engineers are interested in determining whether vibration modes such as these will be amplified by interaction with the fluid or safely damped out.

The "flutter" of blades within compressors and turbines is a serious cause of machine failure that is difficult to predict and expensive to correct. This aeromechanical phenomenon usually occurs at a blade natural frequency and involves sustained blade vibration resulting from the changing pressure field around the blade as it oscillates. For the process to occur, it is necessary that, over one cycle, there is an input of energy from the gas stream to the blade of a sufficient magnitude to overcome the mechanical damping.

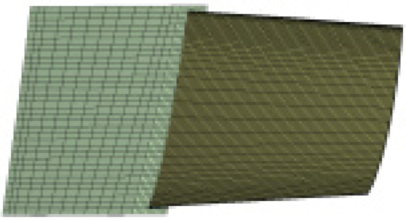
Clearly, flutter is dependent on both the aerodynamic and structural characteristics of the blade, and, until recently, it has been beyond the design capability to satisfactorily investigate and avoid this phenomenon. Historically, empirical design criteria have been used based on parameters involving blade natural frequencies and flow transit times, but these methods fail to take into account generally found vibrational modes or the influence of adjacent blades.

Improvements in unsteady computational fluid dynamics (CFD) capability combined with the ability to easily and accurately transfer information between CFD and finite element analysis (FEA) has enabled the development of an advanced yet efficient and cost-effective methodology for analyzing forced vibration processes.

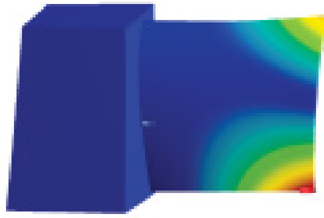
A key enabling development now provided by ANSYS, Inc. is the ability to deform the CFD computational grid in response to deformations at the fluid structure interface and integrate this with unsteady flow computations. The process is straightforward to set up and is facilitated by the intuitive and intrinsic functionality of the user interface and layout in the ANSYS Workbench platform. PCA Engineers Limited, based in the U.K., has utilized this capability by mapping time-dependent deformations computed from a finite element analysis to the CFD computational grid.

As a rule, blade flutter occurs at a blade natural frequency that is determined together with its corresponding mode of vibration using traditional finite element techniques.

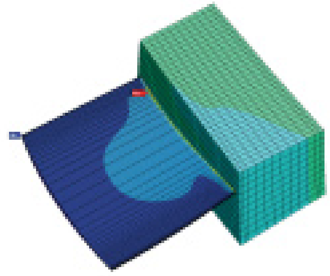
A bladed disc assembly can be classified as a rotationally periodic structure, and, therefore, the mode shape of adjacent blades within a row are fully defined by a phase



Finite element (FE) mesh at the fluid–structure interface



A typical torsional blade mode, where the relative amplitude of each node point on the gas swept surface of the blade is known as a function of time



Equivalent stresses

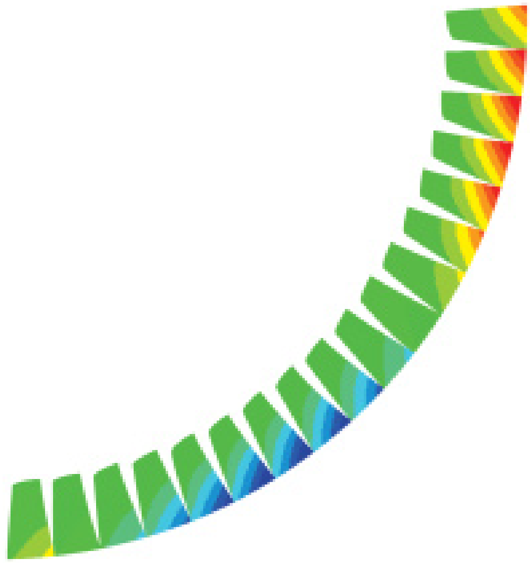
difference. This phase difference (the inter-blade phase angle or IBPA) depends on the number of blades in the row and the number of patterns repeating around the annulus. This latter parameter is often called the nodal diameter (ND) and can move either in the direction of rotation or against the direction of rotation.

The significant development is that this modal displacement information now can be applied to the computational grid and the resulting time varying flow through a blade row as well as the dynamic pressure field over each defined blade calculated using ANSYS CFX software. The computed dynamic pressure distribution and the corresponding modal displacements then are used to compute the work done on the blade over one complete cycle. If the net work done on the blade is positive, then work is being imparted to the blade, creating negative damping, a potentially unstable situation leading to a self-sustained vibration (flutter) likely to cause a material fatigue failure. On the other hand, if the aerodynamic work done on the blade is negative, the blade motion is doing work on the fluid and leads to a stable or damped vibration.

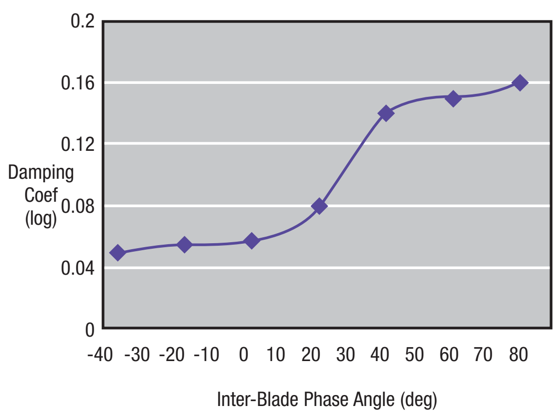
In the aerodynamic damping case illustrated, the blade is stable (no flutter) because the damping is always positive. This information is critical to the designer as blades are relatively easy to modify before manufacture but extremely costly to rectify in an operational plant. By utilizing blade flutter prediction early in the design cycle, costly damage and repairs can be avoided. This integrated design and analysis approach in multiphysics technology from ANSYS can lead to improved quality and dependability of the design process, realizing further cost benefits to clients.

ANSYS, Inc. and PCA Engineers now are applying such technology to a wide range of applications extending from large steam turbines to small turbochargers. These techniques are assisting engineers to design compressor and turbine blading in which both aerodynamic efficiency and structural integrity are paramount over the operational range of the machine. ■

www.pcaeng.co.uk



Deformations of a four-nodal diameter which repeat once over each quarter of the wheel, were exported from the modal analysis vibration mode to ANSYS CFX software as a boundary profile. The mode shape is used to create a periodic boundary motion in the CFD software and to evaluate the net work input due to the blade motion.



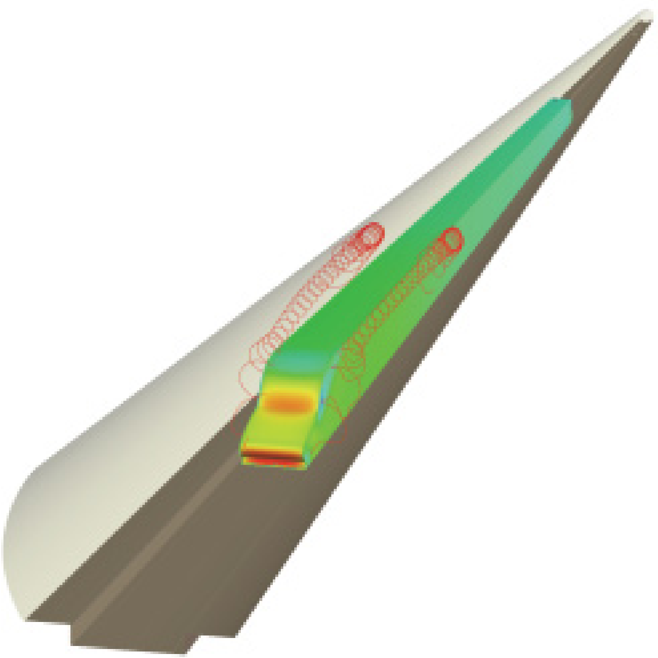
Damping coefficients can be calculated from the CFD results. Negative net work input due to blade motion results in a positive damping coefficient. Negative damping coefficients induce sustained blade vibration, or flutter, which could lead to blade failure. These results show positive damping for all inter-blade phase angles.

Ventilating Giant Railway Tunnels

High-speed trains in Spain cross more than just the plain.

Image courtesy Eurorail Group.

By José Carlos Arroyo and Pedro Luis Ruiz, INECO-TIFSA, Madrid, Spain
Yannick Ducret and Roberto Garcia, ANSYS, Inc.

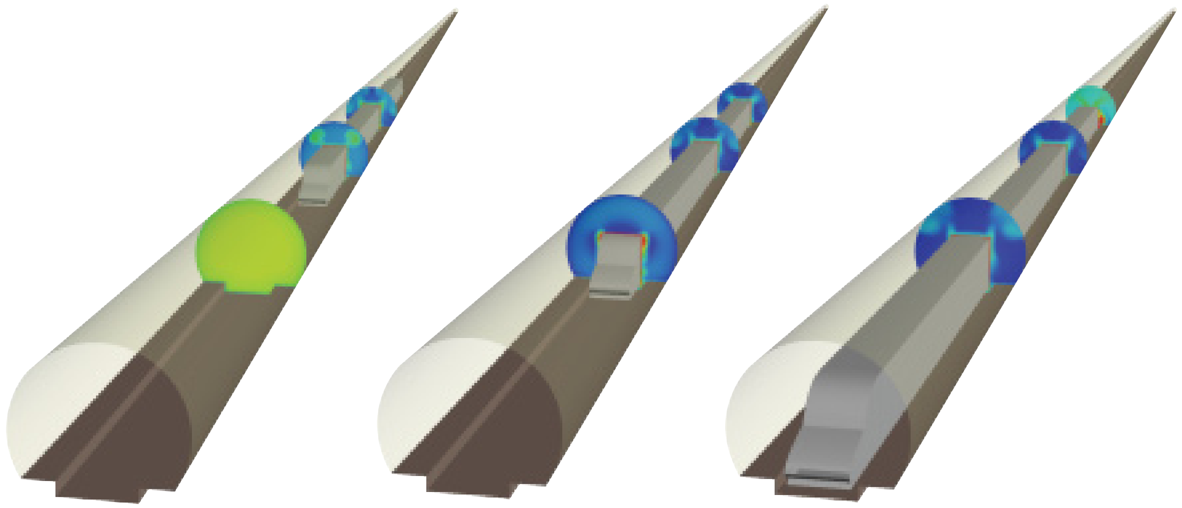


Pressure contours on a train with vortices shown by streamlines in a tunnel

Sunny beaches filled with sunbathers may be the first thing that comes to mind when imagining Spain. However, the reality of its geography is a lot more variable. In fact, the Iberian Peninsula sports many mountain ranges that largely hinder the development of complicated infrastructure, such as the high-speed rail network that is planned to connect Spanish urban areas. This project has led the Spanish railway industry to bore some of the world's longest high-speed transit railway tunnels, such as the 28-km Guadarrama tunnel and the 24.5-km Pajares tunnel. INECO-TIFSA, a transport and telecommunication company located in Madrid, Spain, has contributed to the ongoing expansion of high-speed railways throughout the country by participating in the design of superstructures, such as these giant tunnels.

When designing a railroad tunnel, the ventilation system is a critical component. The ventilation units themselves consist of longitudinal jet fans that are placed at several positions along the tunnel. Their performance is affected severely by air disturbances that result from the motion of a train traveling in the tunnel at speeds of up to 350 km/h (218 mph). Not only does the train movement quickly force the air toward both tunnel exits, it also creates a complex system of pressure waves that propagate throughout the space. A positive-amplitude pressure wave is created when the train enters the tunnel. When the train's rear end passes into the tunnel, another wave, of negative amplitude, originates at the tunnel entrance. Both waves propagate toward the tunnel exit where they are primarily reflected.

Traditionally, 1-D or 2-D simulations have been satisfactory to predict the average pressure correctly inside the



Computational fluid dynamics (CFD) contours at three locations within a tunnel show how the longitudinal velocity changes in the tunnel as a train passes through it. The plane cuts represent the position of jet fans where fluctuating velocities have been monitored.

tunnel. Only by means of a complete 3-D simulation, though, is it possible to obtain an accurate estimate of the velocity components, magnitude and their distribution in the tunnel sections in order to allow for accurate fan sizing.

To design the ventilation system for a long (>5 km) tunnel, INECO-TIFSA chose FLUENT software to perform a full 3-D unsteady simulation of a train passing through such a tunnel. The movement of the train was simulated using the FLUENT sliding mesh capability, in which the train and the domain that it encompasses slide along a non-conformal interface. The interface was placed at the tunnel wall, and the mesh was extruded accordingly. Due to the speed of the train, the ideal gas model was used to account for the effects of compressibility. The computations were performed using the pressure-based solver, which was chosen because the flow is only slightly compressible — that is, there is only a weak coupling between density and velocity, and, thus, the computation does not require the density-based solver. This unsteady simulation was performed using non-iterative time-advancement (NITA) in order to reduce the computational time required. This approach was validated by a series of 2-D computations. Special consideration was given to the determination of the appropriate time step, since it needed to be small enough to predict the wave's propagation correctly.

The velocity components and the static pressure were monitored in seven different locations along the tunnel length corresponding to the ventilators' positions. The flow patterns also were analyzed using velocity contours in various sections of the tunnel. The results showed the amplitude of the flow created by the train's passage. When a train enters a tunnel, air first escapes at the tunnel entrance at the side of the train, both because it is the closest exit and because the mass of air between the front of the train and the tunnel exit has yet to be put in motion. When the train has passed, the flow then changes direction. At that moment the air is pushed by the train and travels backward in the narrow gap between the train and the tunnel. Speeds of up to 35 m/s were observed at the fan positions. Furthermore, some sudden changes of slightly higher amplitude could be seen when the front of the train reached the jet fans, showing how carefully this equipment needs to be selected.

As expected, the pressure waves created by the train's motion do have a discernible effect on flow patterns within the tunnel. Seconds after the train has passed the jet fans at the entrance of the tunnel, the wave patterns form such that they accelerate the air flow by up to 25 m/s. When the air is compressed by a positive-amplitude wave, the air velocity diminishes according to conservation of mass, while the inverse (acceleration) occurs if the wave is of negative amplitude. The train creates both of these types of waves as it passes through the tunnel, thus inducing both accelerations and decelerations in the surrounding tunnel airflow. This complex and decaying phenomenon then continues long after the train has exited the tunnel. Even though the highest velocities observed are longitudinally oriented, the transversal velocity profiles revealed the benefits of a 3-D study, since velocities of the same order of magnitude were observed. Overall, this modeling approach has shown interesting results and proven beneficial for INECO-TIFSA by the level of detail achieved. ■

Glass-Making Goes from **Art to Science**

Modeling glass furnaces helps improve batch transition time and reduce product defects.

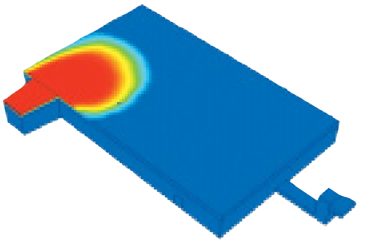
By Eddie W. Ferreira, PFG Building Glass, Springs, South Africa

To create glass from its raw materials is to invest in both the art and the science of the process. Glass is a fascinating engineering material with unique properties; however, producing it can be a complex undertaking and is often thought of as an art. As a result, commercial glassmakers strive continually to understand the science of its manufacture in order to optimize and improve the process. As one such manufacturer, PFG Building Glass in South Africa is using FLUENT computational fluid dynamics (CFD) software to model the flow inside its glass furnaces, track processing defects and improve overall production systems.

At the basic level, glass-making consists of three steps. The first is melting a blend of raw materials, which can include sand, limestone, soda ash, feldspar and saltcake. The next is refining, in which bubbles contained within the molten raw materials are removed. Finally, there is conditioning, in which the glass is cooled to a suitable working temperature. There are various methods of accomplishing each step that affect the process differently. Different glass compositions require different operating envelopes, due to the change in physical and chemical properties.

Because glass-making requires furnace temperatures of 1500 degrees C (about 2700 degrees F), heat transfer and chemical diffusion dominate the process kinetics, and the reaction tank itself is slowly dissolved by the molten glass. These factors make experimental studies difficult. As an alternative, simulation arises as a good way to understand how furnaces behave and how process improvements can be made.

Using the 3-D version of FLUENT software and the pressure-based solver, PFG Building Glass developed a CFD solution for steady-state glass processing conditions. The company created a simplified initial simulation, one that did not include any time-dependent events.

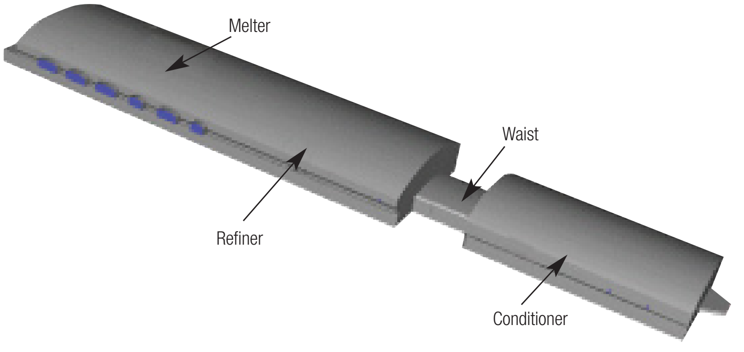


Contours of batch species fraction in the glass domain of a container furnace. The red area represents the introduction of a new species into the glass flow.

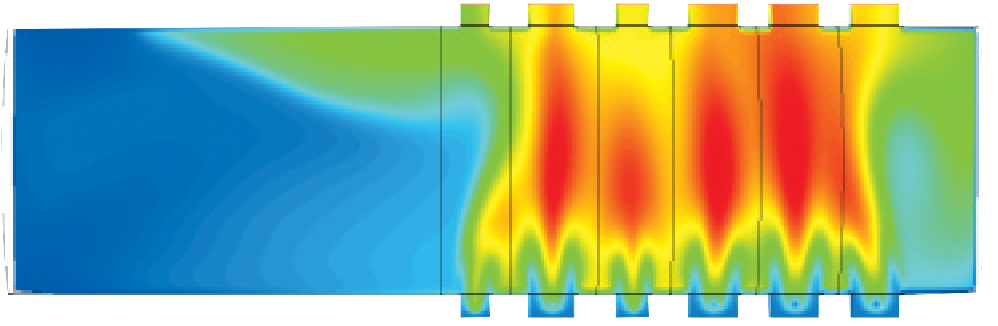
Once these simulation results were acquired, time-dependent events, such as color transitions, were incorporated into the simulation and accounted for by switching to the transient solver in the FLUENT product. To simulate this more complicated type of process, PFG enabled the species transport model in FLUENT software and incorporated its own batch models via user-defined functions (UDFs) for the species properties, boundary conditions and sources. These additions allowed the team to observe factors such as mixing.

Apart from the glass flow itself, what happens in the combustion space above the processing glass is very important. Combustion that occurs in this region of the furnace is a heat source for melting and heating the glass mixture. In order to include this region in the analysis, PFG incorporated combustion, radiation and turbulence modeling into the simulation. By including these factors, the model complexity was greatly increased.

The combustion space model and the glass flow model then were



Model of a float glass furnace. The most common method for glass production is floating molten glass on top of molten tin, thus giving it the name "float glass." This process results in the formation of plates or ribbons of glass.



Contours of temperature in the combustion space over the melter region of an oil-fired, six-port float glass furnace. Red areas identify regions of higher temperature.

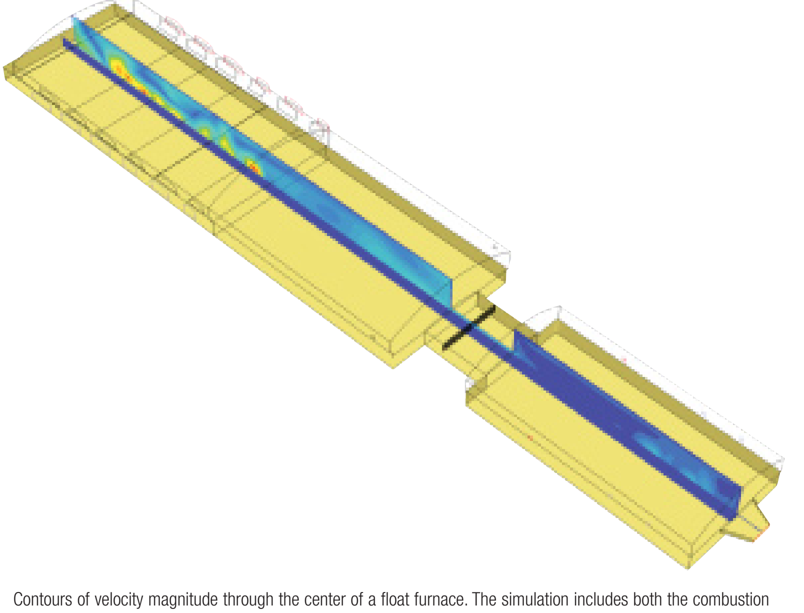
combined into a coupled simulation, making use of the FLUENT non-premixed combustion model, discrete ordinates (DO) radiation model and realizable k-ε turbulence model. Further UDFs were used to define the material properties and source terms. For boundary conditions, it was important to maintain the glass zone as a laminar zone and the combustion zone as a reaction zone.

The quality of the final glass product is influenced by the presence of small bubbles, which manufacturers try to remove from the batch during a refining phase because bubbles can lead to discrete faults in the final product. There are numerous sources that can lead to an unacceptable rise in the number of faults. Using simulation for defect tracking has helped PFG to pinpoint the areas that are most probably the origin for the faults. PFG accomplished this with the reverse particle tracking capability in the FLUENT product. By defining tracking locations throughout the glass fluid domain and using the FLUENT discrete phase model (DPM), PFG was able to examine a particular particle's flow path history and determine statistically probable fault positions in the final glass ribbon.

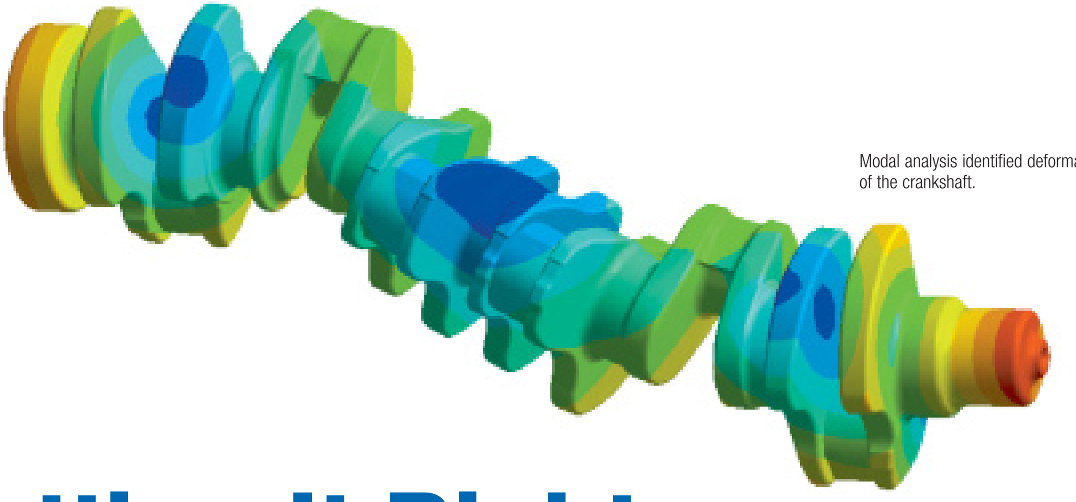
One other complication of a batch process is the transition from one batch to another, which involves moving the complete furnace glass volume. Usually the transition process takes a number of days. As a result of factors driven by this transition time, glass that does not fall within approved specifications is produced with an associated loss of revenue. Any reduction in transition time is, therefore, of great value. PFG has been able to partially model this transition process using FLUENT software, leading to modifications in operating procedures.

The use of CFD modeling has led to a better understanding of glass flows and combustion conditions inside glass furnaces. This has allowed PFG Building Glass to achieve its objective of producing high-quality glass at the lowest possible price, while also maintaining long furnace life, all without a hit-and-miss approach. Product quality has improved as a result of defect tracking, and losses have been reduced as the process has become more of a science than an art. This experience and the models drawn up allowed PFG to simulate planned expansions before they were installed and, thereby, eliminate problem areas before installation. ■

The authors would like to acknowledge Peet Drotskie and Corne Kritzinger from PFG, who laid the groundwork for these modeling efforts, as well as Danie de Kock and his support team at Qfinsoft for their invaluable input.



Contours of velocity magnitude through the center of a float furnace. The simulation includes both the combustion space above the glass melt and the melted glass itself.

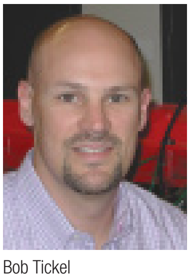


Modal analysis identified deformation of the crankshaft.

Getting It Right the First Time

In a corporate-wide initiative, Cummins Inc. refines designs early with Analysis Led Design to shorten development time, reduce costs and improve product performance.

By Bob Tickel, Cummins Inc., Indiana, U.S.A.

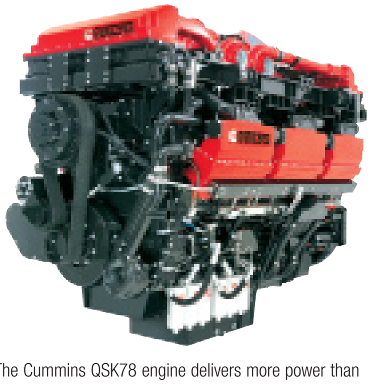


Bob Tickel
Director of Structural and Dynamic Analysis

In developing complex mechanical products such as diesel engines, going through multiple build-and-test hardware prototype cycles to verify performance, stress and fatigue life is tremendously expensive and time-consuming. This issue can be addressed by evaluating and refining designs with analysis tools up front in development, so fewer test cycles will be needed later in development.

Five years ago, such a Simulation Driven Product Development approach was started at Cummins Inc., a corporation of complementary business units that design, manufacture, distribute and service engines and related technologies, including fuel systems, controls, air handling, filtration, emission solutions and electrical power generation systems. Applications include trucks, construction and mining equipment, agricultural machinery, electrical generators, fire trucks, recreational vehicles, buses, cars, SUVs and pickup trucks. The Cummins Analysis Led Design (ALD) strategy is a corporate-wide initiative to change the prevalent test-first culture; it has had a major impact at the company, with significant benefits that include shorter development time, lower costs and improved products.

ALD can shorten product development time by getting designs right the first time. Many Cummins-designed parts have extensive lead times because tooling needs to be created. Beyond this, traditional hardware testing can take weeks or even months to validate a design. Leveraging analysis early in the process can eliminate tooling changes and repetition of lengthy endurance testing, thus providing significant reductions in overall development time.



The Cummins QSK78 engine delivers more power than any other engine for gigantic haulers in the mining industry. The 18-cylinder, 12-ton super-engine is rated at 3,500 horsepower and stands almost eight feet high. Cummins also provides engines for agricultural and industrial equipment and heavy-duty pickup trucks.

Simulation also radically lowers the total cost of product development through less dependency on hardware tests and a reduced number of long-hour tests, which sometimes can last for days. At Cummins, some of this traditional endurance testing can cost in the range of $50k to $100k per test, so eliminating even a single cycle can result in significant savings. The intention is not to eliminate all testing but, rather, to use targeted component and assembly-level testing first to validate analysis models and then to validate the overall design with only a few long-hour tests of the entire engine. Savings also are achieved by eliminating redesigns, in which costs are lowered by reducing resources required to manage the design process (engineering, drafting, clerical time, etc.) as well as reducing retooling costs.

This customized Kenworth tanker truck has a Cummins 565-horsepower engine.

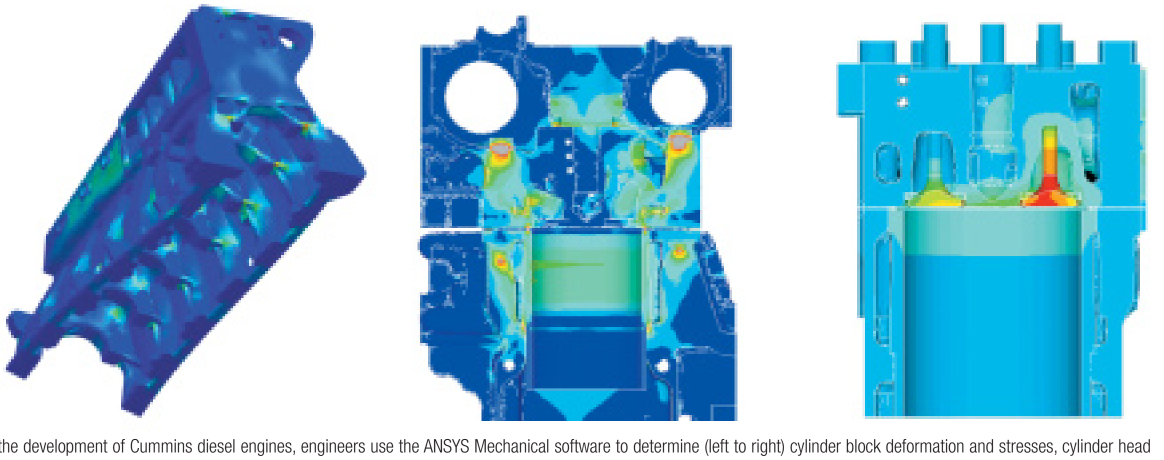
While shortening development time and lowering costs are important aspects of ALD, it can be argued that the most significant benefit of the approach is the ability to create improved products by considering a broad range of design alternatives. Simulation allows engineers to readily perform what-if studies and large-scale design of experiments in order to understand more fully the design space and trade-offs involved. Otherwise, once the first set of hardware is created, the design space narrows and designs are much harder to modify.

Various measures have been used within Cummins to help determine the effectiveness of ALD. In looking at test time and cost in one example, validation testing for a cylinder block traditionally required $72k of rig testing and $30k to $80k for engine testing for a single block design. Each repetition costs the same amount: in the range of $100k to $150k. Testing usually took about one month, once hardware was available. Lead time for the tooling and part procurement took about 12 weeks.

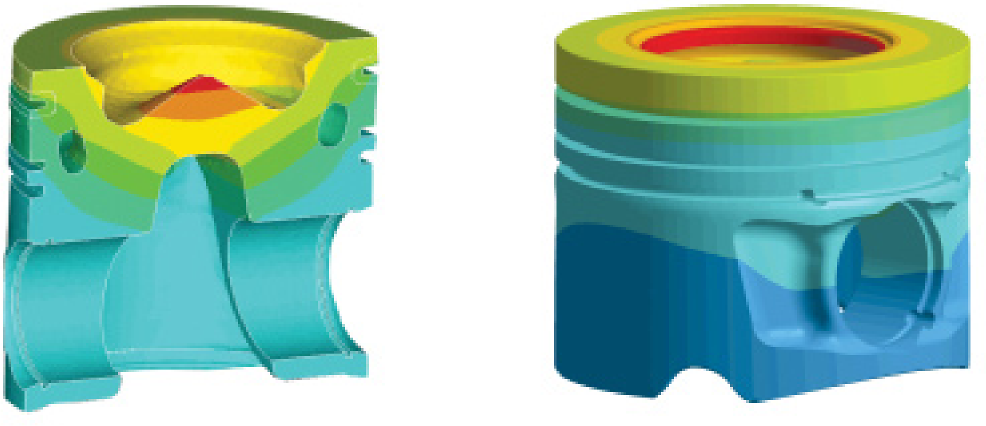
Through the ALD initiative, engine testing has been removed as a requirement for some cylinder block validations. Now when a new heavy-duty engine design is being developed, a series of repetitions are done through simulation until the entire block meets the design limits. This requires the time of one analyst for about a month of work, or approximately $7k. Once the hardware is procured, rig testing is completed on the initial pass — a first for this type of design. The result is that a minimum $30k of engine testing is eliminated. Also, redesigns are eliminated that, most likely, would have occurred over many more weeks or months and at an additional cost of $100k ($72k of rig and $30k of engine testing), which does not include the significant additional expense of prototype hardware.

There are several reasons why ALD has been successful at Cummins: It is a top-down initiative that was driven by upper management, appropriate resources were allocated, and an infrastructure was established to support the initiative.

From the beginning of the program, top management has been a strong proponent of ALD. Cummins' chief technical officer coined the acronym ALD, and he has continued to push the initiative. The progress of ALD has



In the development of Cummins diesel engines, engineers use the ANSYS Mechanical software to determine (left to right) cylinder block deformation and stresses, cylinder head assembly stresses and temperature distribution in the cylinder head and valves.

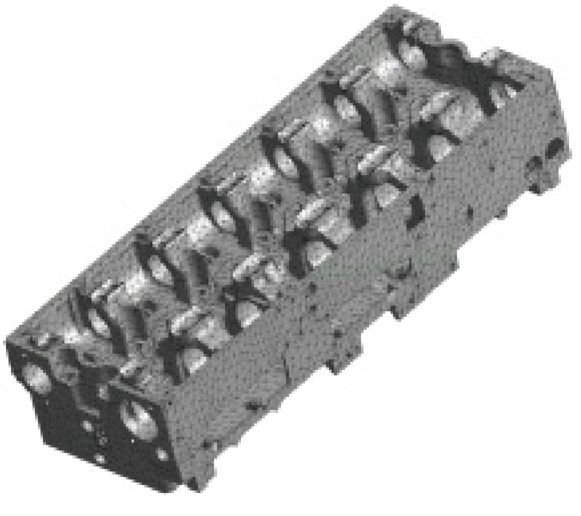


Thermal analysis shows temperature distribution for a diesel engine piston.

been monitored continually and reported in quarterly messages by Tim Solos, Cummins CEO. At the executive levels, there has never been a question about whether to reduce testing and increase analysis but rather how to best accomplish this objective with limited resources.

Along with driving ALD, Cummins management provided resources to do more analysis. Shortly after the ALD initiative was started, a technical center was set up (Cummins Research and Technology, or CRT, in India). This analysis center focuses solely on design, computational fluid dynamics (CFD) and structural analysis in supporting all Cummins business units.

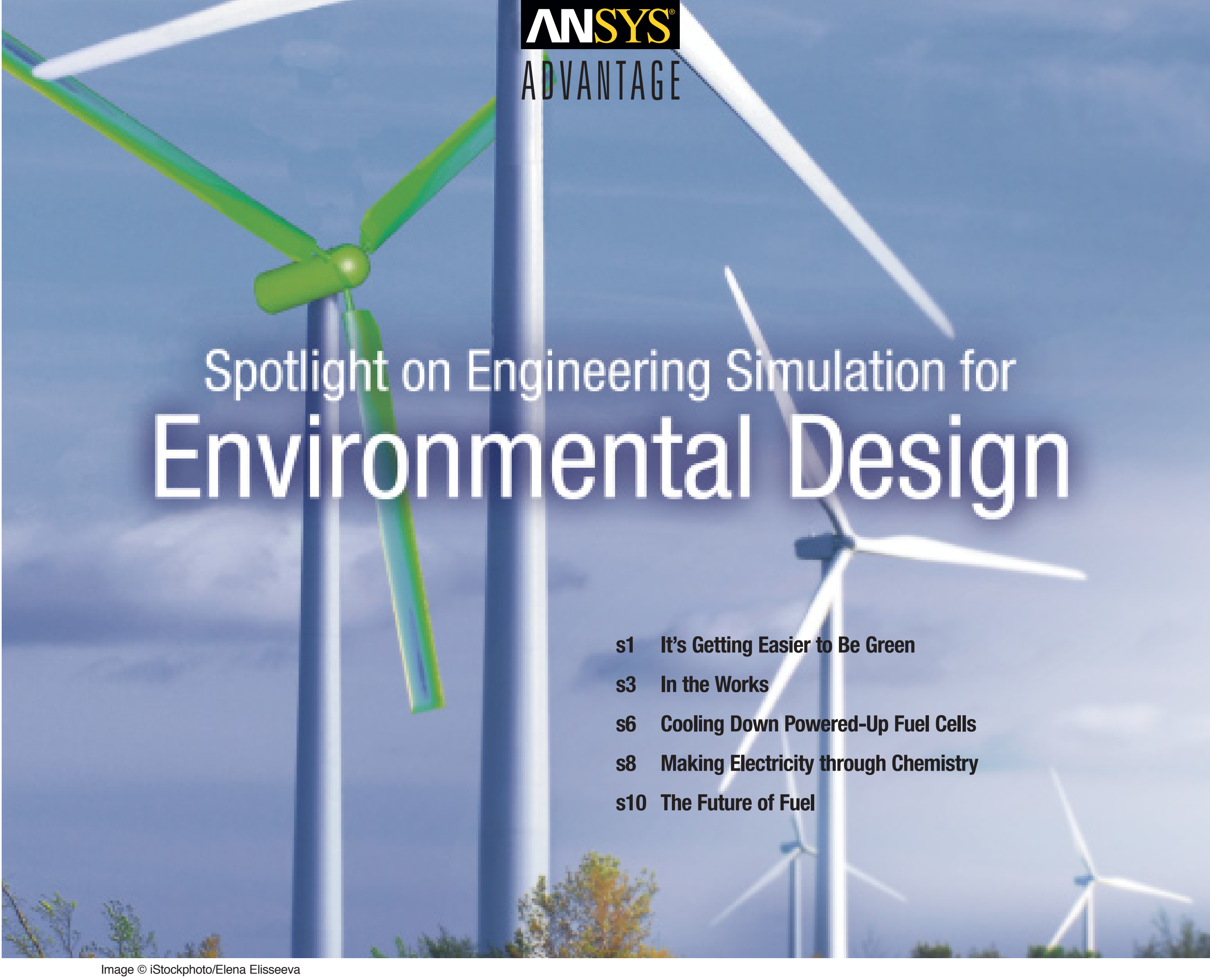
Infrastructure to support ALD at Cummins has taken two forms: Engineering Standard Work (ESW) processes and Six Sigma tools. ESW defines the work, tools and limits required to release a part for production. This became a natural focus for ALD as Cummins examined where testing was being reduced and where analysis was being increased. Six Sigma has been an invaluable support for ALD in validating new tools and methods to ensure that analysis can be used to replace testing. So, ALD is the initiative, ESW is the process to ensure that all necessary work is completed and Six Sigma is the set of tools used to determine that the appropriate work is included.



Coarse mesh of detailed geometry for an inline six-cylinder head created using the ANSYS Workbench platform

In performing the underlying work for ALD, the Structural and Dynamic Analysis group within the Cummins Corporate Research and Technology organization is responsible primarily for developing tools and methods as well as conducting analyses to ensure that structural components meet both reliability and durability requirements. The group partners with key software vendors in efforts to develop improved simulation tools, and one of the primary relationships is with ANSYS, Inc. In fact, the relationship has been the benchmark set for subsequent partnerships. Technology from ANSYS has become the primary finite element tool within all Cummins business units for conducting static structural, thermal, transient thermal, modal, harmonic and other analyses. This partnership with ANSYS has resulted in joint development of advanced features in continuing to meet analysis needs at Cummins.

Any culture shift is difficult, requiring vision, leadership, planning and tangible benefits. The ALD initiative, in particular, has driven considerable change and has proven to be of tremendous value at Cummins. While significant progress has been made, there is room for expansion, and Cummins will continue to evaluate new and improved technologies, processes and strategies in using simulation to further strengthen its position in the diesel engine industry. ■



Spotlight on Engineering Simulation for Environmental Design

Image © iStockphoto/Elena Elisseeva

It's Getting Easier to **Be Green**

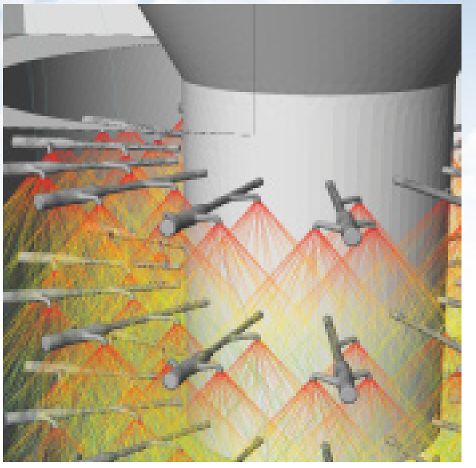
From air to water to power, industries are using engineering simulation to uncover new ways to be environmentally responsible.

By Dave Schowalter, ANSYS, Inc.

The term "green engineering" has become ubiquitous in recent years, with references even on the covers of trade journals and magazines. The U.S. Environmental Protection Agency defines green engineering as "the design, commercialization and use of processes and products that are feasible and economical while reducing the generation of pollution at the source and minimizing the risk to human health and the environment." So while green engineering encompasses environmental engineering, it also can refer to any engineering field in which environmental and human health impacts are minimized. Increasingly, the term has become associated with sustainable development, in which processes and products can continue to be produced indefinitely with a minimum of resource depletion or environmental degradation.

Along with increased awareness of environmental impact, well-known corporations have launched campaigns that show how they are developing green technologies. Of particular note are General Electric's ecomagination™, the BP™ campaign Beyond Petroleum and Chevron Corporation's willyoujoinus.com advertising promotion. One thing is clear: Major companies believe there is money to be made in developing environmentally friendly technology, which should encourage even the most contrarian environmentalist.

In building a better world, global companies are learning that the right engineering simulation can improve efficiency in the design of real-world systems. Simulation capabilities from ANSYS, Inc. are particularly visible in the areas of pollution control, architecture, energy and sustainable technology. This spotlight on the environmental industry provides details about how hard-working users of engineering solutions from ANSYS are improving the environment. Perhaps readers will find themselves inspired.



Particle tracks in a wet SO_2 scrubber in which simulation was used to optimize pollutant capture efficiency. Image courtesy URS Corporation.

Clean Air

Air pollution comes primarily from transportation and point-source industrial processes. In the transportation arena, there is particular emphasis on particulates and nitrous oxides (NOx), with increasing efforts to reduce carbon dioxide emissions through efficiency improvements. Reducing any type of pollution can involve heavy simulation usage for flow, chemistry, heat transfer and thermal stress minimization.

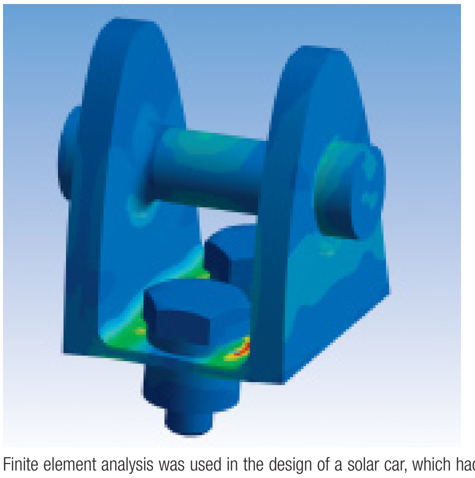
Industrial sources are concerned with particulates, NOx, and carbon dioxide, as well as sulfur oxides (SOx) and mercury. The low pollutant levels achieved today through optimized furnace combustion and optimized flow distribution in downstream pollutant capture systems would not be possible without virtual prototyping through computational fluid dynamics (CFD). Additionally, minimization of material usage requires an understanding of thermal stress loads through structural analysis.

Clean Water

Engineers are using fluid flow modeling — including solutions from ANSYS — to optimize both municipal and commercial purification processes, such as tank mixing, ultraviolet disinfection, chlorination and ozone contactors. Modeling also comes into play in wastewater treatment, which involves similar processes, in addition to phase separation.

Protection of fish is another aspect of clean water, and simulation has been used to design oxygenation systems and retrofits in hydropower dams, that are aimed at increasing downstream oxygen levels. Modeling of water intake structures at industrial plants also is contributing to reduction of ecosystem impact.

Run-off and drift from commercial and residential pesticide treatments can affect water as well as air; simulation is used to optimize chemical dosing, and to model and understand dispersion.



Finite element analysis was used in the design of a solar car, which had severe weight limitations. Image shows the stress analysis on A-Arm clevis for the car's suspension. Image courtesy University of Toronto Blue Sky Solar Racing.

Green Building

Green building refers to designing commercial and residential buildings that minimize non-renewable energy usage; use materials whose production has a minimal environmental impact; and use heating, ventilation and air conditioning methods that maximize air quality. Safely minimizing material usage and maximizing passive ventilation through natural circulation makes this an active and growing area for simulation.

Renewable Energy

Of all renewable energy technologies, wind power has taken the most advantage of simulation capabilities. Today's large wind turbines require advanced materials, increased efficiency, reduced weight while avoiding fluid structure interaction, and the ability to withstand seismic vibrations. Because the power that can be extracted scales as the cube of the wind velocity, placement decisions can have a major impact on the profitability of a project. Other renewable energy technologies that take advantage of products from ANSYS include tidal power systems, solar power installations, and biomass power and energy.

Sustainable Technology

Drastic reductions in energy usage and pollution production are possible with new technologies such as fuel cells, advanced nuclear power plants (including nuclear fusion research), advanced coal power (including gasification) and hybrid automobiles. For these technologies, simulation is in on the ground floor of development, playing an especially active role in next-generation products.

In order to support the ever increasing rate of technology development that is required for global environmental sustainability, computer aided engineering tools themselves must be scalable and sustainable, which is why ANSYS gives the highest priority to developing multidisciplinary, multiphysics tools all within a single accessible environment, deployable on the desktops of engineers in the small venture start-up as well as on the large parallel servers in engineering departments of major multinational corporations. ■

References

[1] U.S. Environmental Protection Agency, http://www.epa.gov/oppt/greenengineering, 2006.

In the **Works**

Using simulation to model wastewater treatment plants effectively.

By David J. Burt
MMI Engineering
Bristol, U.K.

In response to various European environmental legislative drivers — which include urban wastewater treatment, fresh-water quality standards for protection of fish and water framework directives — U.K. water companies have embarked on a new asset management plan. Part of this plan requires the treatment of significantly greater amounts of wastewater, either by building new treatment plants or by increasing flows through existing plants or works. At the same time, many sites face additional tighter constraints for effluent discharge. The majority of wastewater is treated in modern, large-capacity activated sludge process (ASP) plants. Water companies have been making increased use of analytical process modeling tools, such as computational fluid dynamics (CFD), to find capital cost savings, achieve performance improvements and improve energy savings for these plants.

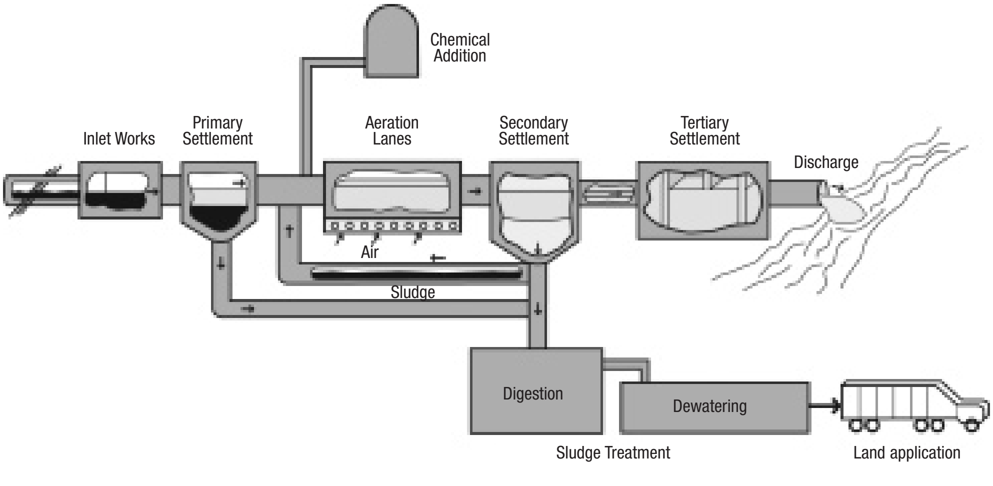
A modern wastewater ASP includes several operational stages that may be modeled with CFD. However, using CFD to investigate these unit operations successfully requires some process knowledge. This article illustrates a few of the processes and explains how they are best addressed with ANSYS CFX software and multiphase modeling techniques.

The basic sequence of operations at a wastewater treatment site with an ASP plant includes the following stages:

- Inlet works with de-gritting and flow balancing
- Primary settlement
- Activated sludge treatment in aeration lanes
- Secondary settlement
- Tertiary treatment

Inlet Works

In most U.K. works, the wastewater enters from an upstream combined sewer system. This wastewater is a mixture of rain water and sewage loaded with solid particles of irregular size, shape and density. A large inlet works removes gross solids and delivers equal flows and loads to the multiple lanes of an ASP; otherwise, the lanes may



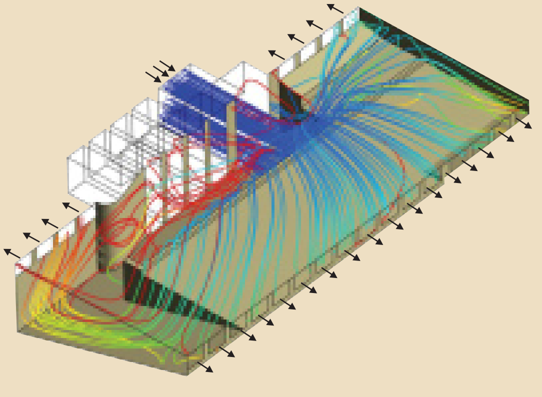
become overloaded or underloaded, and, subsequently, they will not work as well. CFD modeling of the inlet works can be used to determine the equality (or inequality) of the flow distribution among the lanes, as well as the trajectory and final resting place of solids that move independently of the bulk fluid. For example, a discrete particle tracking model may be used to determine the solids retention efficiency of grit traps and balancing tanks, whereas a continuum multiphase model may better show how solids move independently of the water down the different lanes of a distribution chamber.

Primary Settlement

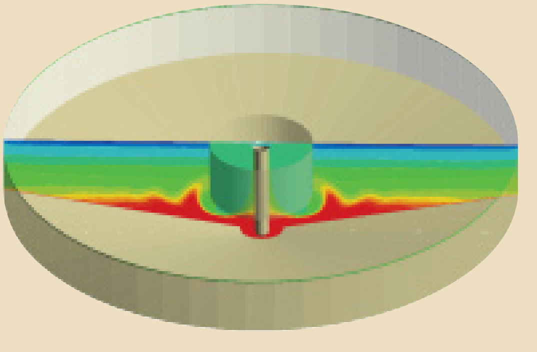
After removal of the larger solids in the inlet works, the wastewater passes into a primary separation zone. The primary tanks are often circular with a central influent, or riser pipe, at the center of the tank. Separation of solids occurs by settlement. The ability to retain solids depends on the balance between the radial up-flow velocity in the tank and the solids' settling velocity. In order to model settlement in a primary tank with CFD, a multiple drift flux model is used in which the influent solids particle size distribution is defined as a series of size groups (mass fractions). Each size group has a drift settling velocity pre-calculated from knowledge of the wet solids density. The total solids concentration thus is determined from the sum of the size groups progressing through the tank. This multiple drift flux modeling technique has been used to determine the optimum number of primary tanks and their required side wall depth for new build sites in the U.K., thus minimizing the land use requirement and reducing overall civil engineering costs.

Activated Sludge Treatment in Aeration Lanes

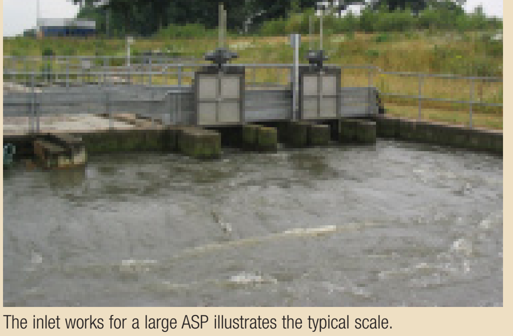
After primary separation, the wastewater stream passes into a series of aeration lanes in which bio-chemical reactions occur that convert the solid particulate waste into activated sludge. The sludge then can agglomerate (or flocculate) into large clusters of particles that can be more readily separated by sedimentation. The bio-chemical reaction rates depend on the levels of dissolved oxygen present within the wastewater. These levels can be modeled with multiphase CFD. A surface aerator, which draws liquid and solids from the lower region of the tank up through a draft tube and then sprays them back across the surface of the tank, influences the solids distributions within the tank and also introduces oxygen into the aeration lane. A study that varied the length of a draft tube diffuser was performed to investigate how the geometry affected the sludge bed



The interstage chamber was modeled with ANSYS CFX software. Streams from an inlet culvert demonstrate the typical flow patterns at the inlet distribution chamber. The streamlines are colored by time, with blue representing the initial time at 10 seconds.



A primary settling tank was modeled with multiple drift fluxes. This plot shows the stratified distribution of solids through a typical 30-m diameter tank.



The inlet works for a large ASP illustrates the typical scale.

Storm settling tank influent

entrainment. This research found that a longer draft tube should be used with the surface aerator under investigation. This change was shown to maximize the aeration and mixing capacity.

Secondary Settlement

After traveling through the aeration lane, the wastewater undergoes secondary treatment in a clarifier. The activated sludge settles out and the effluent passes over a v-notched side weir. The secondary clarifier may be modeled with an extended drift flux model incorporating both sludge settlement and rheology models defined as functions of local concentration. The results of simulation provide both the gradient of solids within the tank ranging from less than 5mg/l in the surface water to greater than 20,000 mg/l in the compressive zone near the bottom of the tank and a measure of the likely effluent solids concentration (solids going over an exit weir, typically in the range of 10 to 30 mg/l). This method has been used extensively to prove clarifier performance — as compared with idealized mass flux theory — and to optimize the position of retrofit baffles to allow a higher flow throughput for the same effluent solids concentration on existing units. MMI Engineering has used these techniques to design optimum clarifier influent arrangements that increase throughput and maintain the effluent solids at more than 40 sites.

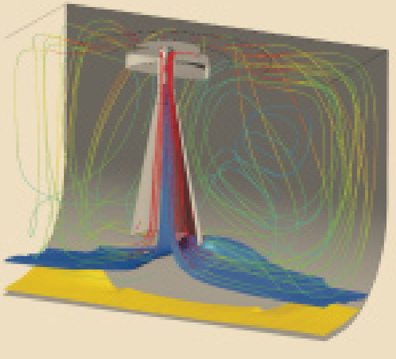
This article illustrates four examples of applying CFD to wastewater systems. Many other unit operations may be examined with similar models to those described here. The extension of aeration lane modeling to include microbial population balances and bio-kinetic reactions (the ASM1 model) currently is being investigated at MMI Engineering. ■

For further guidance on using CFD for wastewater modeling, consult the Aqua Enviro training course "Introduction to CFD Modeling for Water and Wastewater Treatment Plants" at www.aqua-enviro.net/calendar.asp.

References

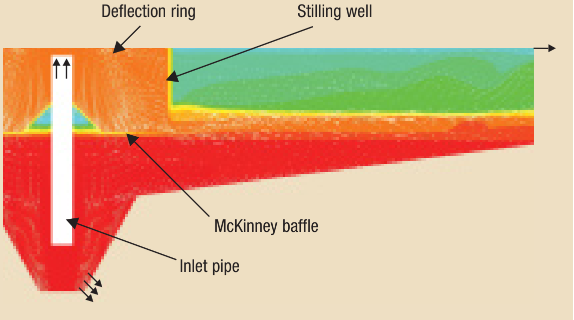
[1] Burt, D.; Ganeshalingam, J., "Design and Optimisation of Final Clarifier Performance with CFD Modelling," CIWEM/Aqua Enviro joint conference, Design and Operation of Activated Sludge Plants, April 19, 2005.

[2] Robinson C.; Wilson R.; and Hinsley S., "Calculating Primary Settling Tank Performance with Computational Fluid Dynamics," 4th Annual CIWEM Conference, Newcastle, U.K., September 12–14, 2006.



CFD was used to determine the influence of the draft tube depth on sludge bed entrainment for the surface aerator in the bio-reactor modeled here. Iso-surfaces of solids concentration are shown using blue at 3,000 mg/l and yellow at 20,000 mg/l. Streamlines identify flow patterns that pass up through the draft tube and are projected out, by way of the aerator, across the tank surface.

The surface aerator of this bio-reactor is used to resuspend the solids bed from within an aeration lane and to entrain air into the reactor.



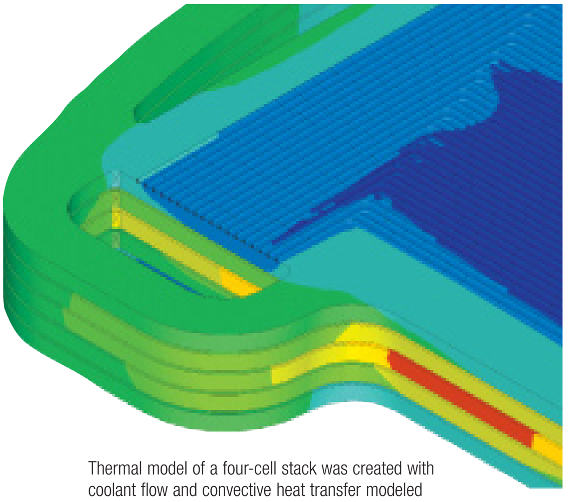
These simulation results depict solids concentration for a radial cross section of a wastewater clarifier. Red indicates areas of higher concentration.

Activated sludge is settled out in this clarifier. Flow enters the tank from the top and flows radially outward.

Cooling Down Powered-Up **Fuel Cells**

Researchers use probabilistic methods and design optimization to improve heat-transfer characteristics of fuel cell stacks.

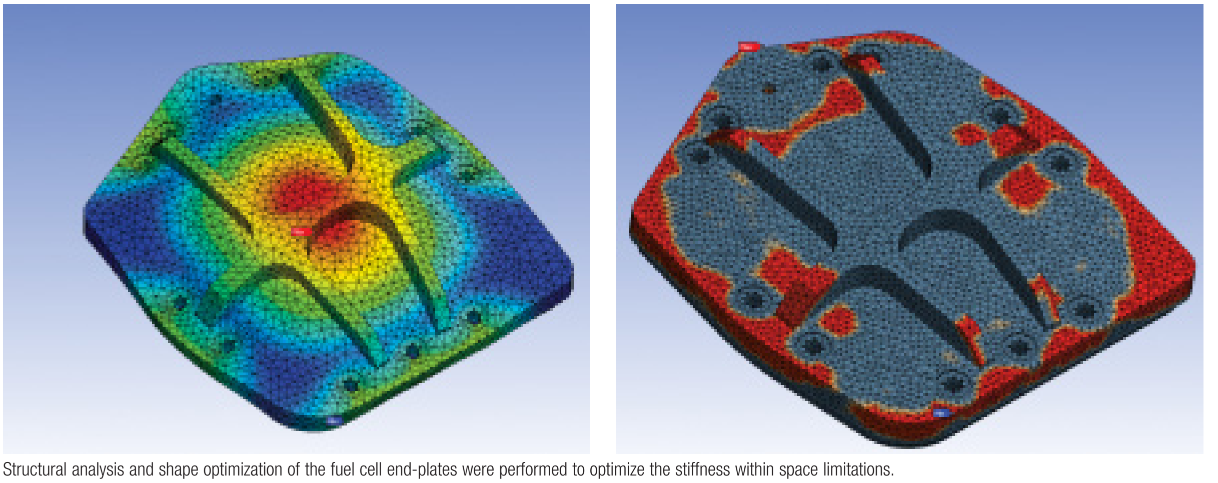
By Andreas Vlahinos, Advanced Engineering Solutions, Colorado, U.S.A.



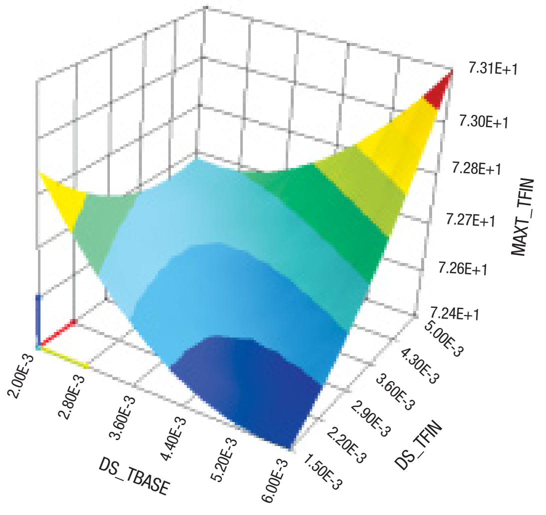
Thermal model of a four-cell stack was created with coolant flow and convective heat transfer modeled with pipe elements. Pipes also were used to model thermal contribution of air and hydrogen flow.

With pure water as the only byproduct, fuel cells are one of the most environmentally safe alternatives for providing power for vehicles and stationary applications: Stacks of the devices generate electricity directly from hydrogen and oxygen. One major concern in designing fuel cell stacks is dissipating heat created during the electrochemical conversion process. Thermal hot spots within the fuel cell stack may degrade performance, induce thermomechanical stresses and shorten the useful life expectancy of the stack.

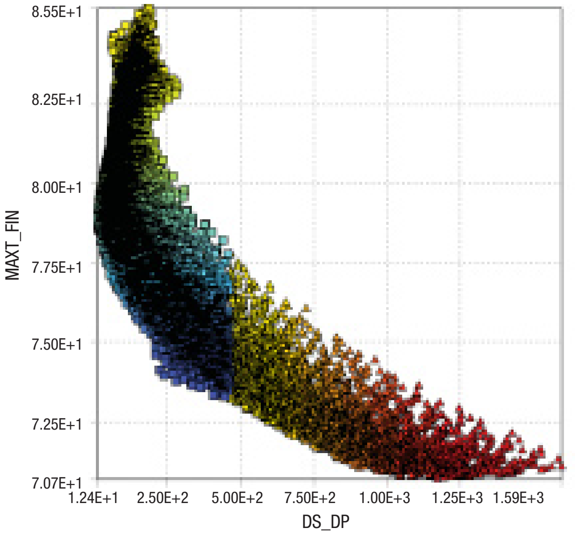
Temperature distributions within the stack depend on many variables, including non-uniform heat generation, fluid properties and flow quality, fuel cell geometry, and the configuration of cooling plates between the cells. To arrive at a suitable design, engineers may resort to numerous prototype build-and-test cycles that are lengthy and costly — not to mention how they stifle innovation — because of the prohibitive time and expense of evaluating new ideas and what-if scenarios. These limitations can be alleviated somewhat with “deterministic” computer-aided engineering (CAE) methods that perform a series of individual analyses. Even in this scenario, engineers must run hundreds or even thousands of individual simulations to arrive at a satisfactory design.



Structural analysis and shape optimization of the fuel cell end-plates were performed to optimize the stiffness within space limitations.



Response surfaces show the relationship between multiple variables, in this case visualizing the impact of fin thickness and base thickness on the maximum temperature of a cell stack.



After generating 10,000 virtual experiments, engineers create a scatter plot of performance requirements showing maximum temperature versus pressure drop. Dark blue squares represent data points that meet all design requirements and have minimal temperatures.

A more efficient way to optimize a design with many variables and uncertainty is to account for variation using advanced computational and probabilistic tools early in the design process. This approach is being used extensively on research for market-viable alternative energy solutions. In some of this leading-edge work, the ANSYS Workbench platform and ANSYS DesignXplorer software have been implemented for performing design of experiments in accounting for uncertainty and variation in materials, manufacturing and load conditions. Simulation tools also are used to streamline laboratory experiments by numerically evaluating the design space to assess and determine which variables have the largest impact on results. Laboratory tests validate the results and are fed back into the model to improve its predictive capabilities.

In one project studying fuel cell design, the engineering consulting firm Advanced Engineering Solutions, based in the United States, used an approach that was aimed at establishing optimal design methodologies for fuel cells. The company also was charged with improving product development time and costs by reducing the number of physical prototypes and laboratory tests required. In one case in particular, the research team used tools from ANSYS to develop a fuel cell stack thermal modeling process to assess design sensitivity on fuel cell thermal performance. The models were used to evaluate new cooling plate flow paths and to assist in the development of improved heat transfer characteristics.

The thermal modeling process incorporated an ANSYS Mechanical 3-D multi-cell stack thermal model that reflected real-world stack geometry and non-uniform heat generation in the membrane. ANSYS DesignXplorer technology was used for design space exploration and probabilistic design methods. Classical design of experiments techniques integrated with the model were used to define response surfaces and perform sensitivity and trade-off studies on heat generation rates, heat-sink fin geometry, fluid flow, bipolar plate channel geometry, fluid properties and plate thermal material properties. A Taguchi screening study was used to identify the most sensitive input parameters; robust design was used to understand the impact of variation on thermal performance.

Researchers at Advanced Engineering Solutions then used the ANSYS thermal model to develop an alternative coolant flow path design that yielded improved thermal performance. The team found that this approach shaved months off the development process and led to innovative designs through improved understanding of fuel cell behavior, especially the impact of a wide range of design variables. ■

References

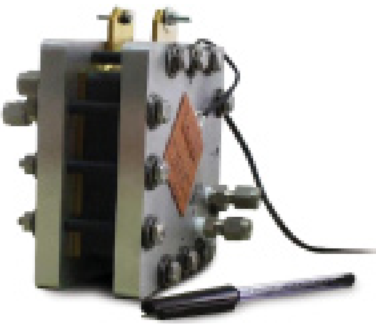
[1] Vlahinos, A.; Kelly, K.; Mease, K.; Stathopoulos J., “Shape Optimization of Fuel Cell Molded-On Gaskets for Robust Sealing,” ASME paper Fuelcell2006-97106, 2006 International Conference on Fuel Cell Science, Engineering and Technology, Irvine, CA June 19–21, 2006.

[2] Kelly, K.; Pacifico, G.; Penev, M.; Vlahinos A., “Robust Design Techniques for Evaluating Fuel Cell Thermal Performance,” ASME paper Fuelcell2006-97011, 2006 International Conference on Fuel Cell Science, Engineering and Technology, Irvine, CA June 19–21, 2006.

Making Electricity through Chemistry

Analysis helps power fuel cell design.

By Laura Ambit and Esther Chacón, Instituto Nacional de Tecnica Aeroespacial, Madrid, Spain
Monica Pardo and Eva Novillo, Compañía Española de Sistemas Aeronáuticos, Madrid, Spain



PEM fuel cell

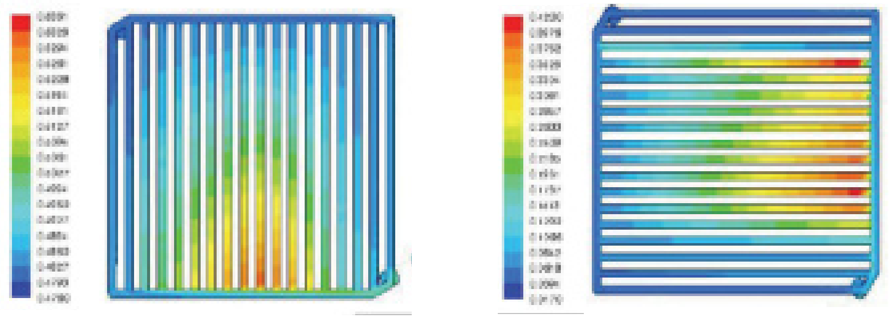
Fuel cells are electrochemical devices that produce electricity from an external supply of fuel and oxidant. Many combinations of fuel and oxidants are possible; however, the fuels most often used are hydrogen, hydrocarbons and alcohols, while oxygen typically is the oxidant. The conversion of the fuel to energy takes place via an electrochemical reaction in which the only byproducts are water (when hydrogen is the fuel) and heat. The process is clean, quiet and highly efficient. For these reasons, fuel cells are highly regarded in the search for sustainable energy sources.

There are several types of fuel cells, and their differences are dependent on the nature of the electrolyte. Polymer electrolyte membrane (PEM) fuel cells operate at lower temperatures than other types, can supply up to 10 W of power per cell and can be stacked to handle higher power loads. The principal applications of PEM fuel cells are in transportation — some experts believe fuel cells will revolutionize the automotive industry — and decentralized stationary electrical applications, which range from powering home co-generation systems to vacuum cleaners and notebook computers.

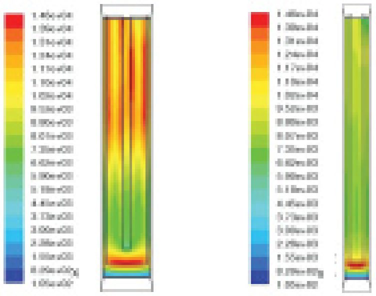
In work done by the Instituto Nacional de Tecnica Aeroespacial (INTA) and Compañia Española de Sistemas Aeronáuticos (CESA) in Madrid, Spain, researchers chose to model a single 7 W PEM fuel cell with the intent of understanding how different geometric configurations and operating conditions affect the cell's performance. The performance depends on a variety of structural and functional parameters, such as the geometry of the flow paths in the bipolar plates, along with the humidity, temperature and operating pressure. To improve the performance of a PEM fuel cell, it is necessary to understand the behavior of variables such as velocity, flow distribution, condensation of water and current distribution. Numerical simulation thus becomes an important tool for understanding the physical phenomena that take place.

The work at INTA and CESA set out to utilize the fuel cell module of FLUENT computational fluid dynamics (CFD) software to capture the fundamental processes of the fuel cell and to optimize the flow path design of the bipolar plates to improve efficiency. To achieve these objectives, a number of simulations were carried out, ranging from the simplest models of fluid flow analysis to more complex ones that included modeling electrochemistry and multiphase flow.

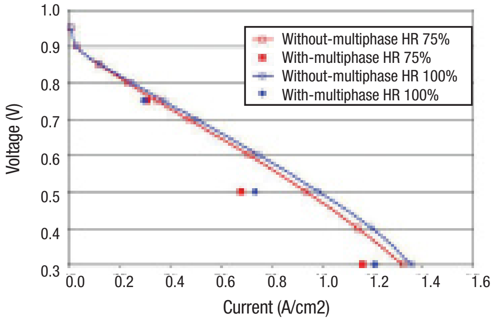
The simulations were conducted using both a commercial geometry with parallel channels and a prototype geometry with two serpentine path flow channels. The effect of operating conditions such as inlet flow humidity, mass flow rate and the influence of geometric parameters such as channel width also were studied in a simplified model of a single serpentine channel.



Contours of the mass fraction of water within the anode (left) and cathode (right) channels for a commercial parallel geometry PEM fuel cell



Contours of current density are plotted in order to compare the effect of varying channel widths on fuel cell design.



Polarization curves based on the FLUENT results showing the influence of relative humidity at the inlets both with and without multiphase flow simulation

Within the parallel geometry, the researchers simulated a laminar, incompressible single-phase (gas only) flow using FLUENT 6.2 technology. In these simulations, the electrochemical phenomena were ignored, and only an analysis of the fluid flow was carried out. The computational domain was restricted to the flow channels of the anode and cathode. Different mass flow rates were simulated, from a high excess of both reactants to the minimum flow that guarantees the electrochemical reaction will occur. The results showed a non-uniform distribution of the flow in all simulations, meaning that a large part of the membrane surface was being wasted. Similar fluid-flow analysis of the serpentine geometry concluded that it allowed a more uniform distribution of flow, and, thus, better electrical conductivity, than the parallel geometry.

In addition, a simulation of the simplified single serpentine channel model was used to better understand all variables involved in the electrochemical reactions and all transport phenomena inside the gas diffusion layer and the membrane electrode assembly (MEA). The FLUENT fuel cell module accounts for reacting flows in contact with the MEA, heat transmission between reactants and bipolar plates, diffusion of reactants through porous media, and liquid water formation via multiphase flow. Using this module, complete polarization curves of the single serpentine channel were calculated. The simulations allowed INTA and CESA to observe the influence of such parameters as the humidity of the mass flow inlet, the mass flow rate and the channel width over these polarization curves, and, subsequently, the electrical current density of the cell. The CFD findings demonstrated that the change of flow direction results in an increase of current density in the local region.

From both the full electrochemical simulations of the simplified serpentine geometry and the fluid flow simulations of the complete serpentine geometry, it is apparent that a serpentine design can improve fuel cell performance when compared to conventional parallel geometry. Building on this research, the current intention is to achieve a full electrochemical simulation of the complete serpentine design using FLUENT software. ■

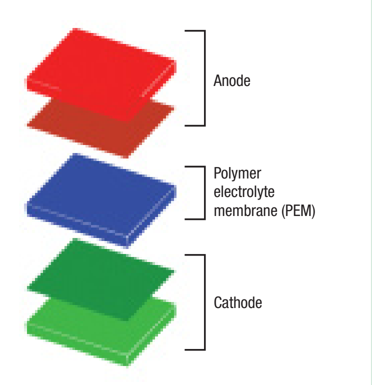
The authors would like to acknowledge Carla Vico for her assistance with this article.

How PEM Fuel Cells Work

The PEM fuel cell consists of an anode and a cathode, separated by a polymer electrolyte membrane (PEM). Simply put, the fuel cell works as follows: Hydrogen and oxygen molecules enter into the device, the hydrogen is broken down in order to produce electricity — and water is created as the byproduct. A catalyst layer is placed between the anode (or cathode) and the PEM.

Oxygen enters the fuel cell on the cathode side of the device. Hydrogen enters the anode side of the device, and, as it comes into contact with the catalyst layer, it splits into two hydrogen ions and two electrons. The hydrogen ions are conducted through the PEM. When the hydrogen ions come into contact with the catalyst layer on the cathode, they join together with the oxygen atoms and recombine with the electrons that have driven the energy-producing current, forming water as the only byproduct for the entire process.

To ensure efficiency in the process, the channels through which the oxygen and hydrogen pass in the anode and cathode should be designed to create as much contact area as possible between the gas molecules and the catalyst layers.



Basic design for a PEM fuel cell. Hydrogen enters the fuel cell at the anode while oxygen enters the fuel cell at the cathode.

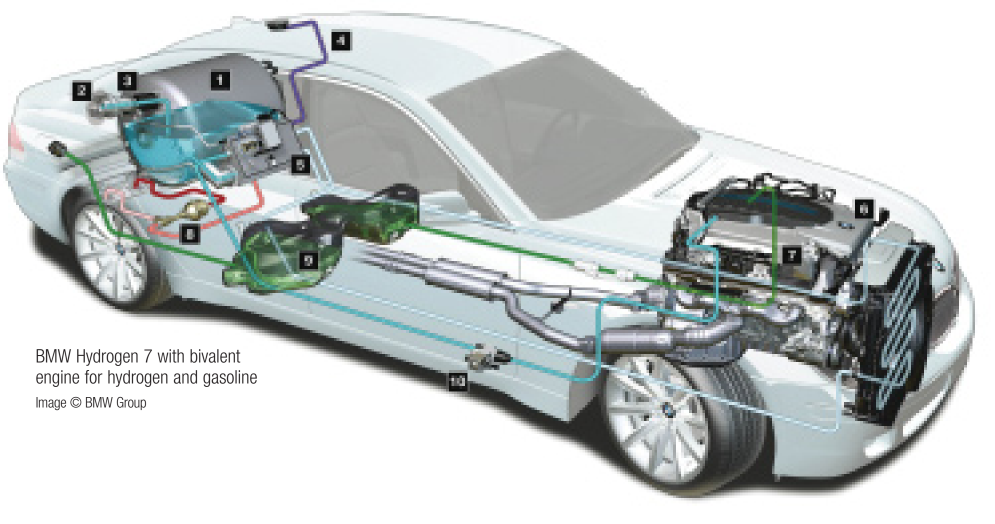
The **Future** of **Fuel**

A European research project is developing internal combustion engines powered by hydrogen.

By Jorge Ferreira, ANSYS, Inc.

Commercially available reserves of fossil fuels are fast running out, and the influence of harmful automobile emissions on the global climate is an ongoing debate. For these and other reasons, researchers and developers have been involved in investigating alternative fuels for the automotive industry. Fuel cells and electric cars are possible alternatives to today's cars and trucks, which are powered by fossil fuels; however, these technologies face some disadvantages, such as limited power dynamics and unsatisfactory power–weight ratios. As another alternative, the internal combustion (IC) engine itself offers many promising solutions if it is fueled by hydrogen. One benefit of hydrogen is that it can be produced from water and a renewable energy source, such as solar power. The main emission that results from this type of process is water vapor, making hydrogen a positive alternative fuel that has the potential for reducing carbon dioxide emissions.

Using trial runs and miniature models, many truck and automobile manufacturers, including BMW, MAN and Ford, have examined what can be done with alternative fuels. The research to date has been conducted mostly on bivalent systems — engines that can use two types of fuels — with most experiments using fossil fuels and hydrogen. The BMW Hydrogen 7™ luxury performance automobile is capable of running on either hydrogen or gasoline. Such mixed use of fossil fuels and hydrogen has the advantage of extending the range of today's cars as compared to pure hydrogen-fueled cars. However, a number of disadvantages arise when using one engine design for multiple and very different fuels. The engine is not optimized for hydrogen nor for gasoline or diesel consumption, meaning that efficiency cannot be optimized. When compared to gasoline and diesel, hydrogen has a good deal of variation in physical attributes, such as density, evaporative characteristics and combustion behavior. As these types of factors have a direct effect on engine performance, it is clear that if hydrogen were



BMW Hydrogen 7 with bivalent engine for hydrogen and gasoline
Image © BMW Group

1 LH_2 Fuel Tank
2 LH_2 Tank Cover
3 LH_2 Tank Coupling
4 Safety Line to Blow Valve
5 Auxiliary Units Capsule containing Heat Exchanger for H_2 and Control Unit of the Tank
6 Bivalent Internal Combustion Engine (H_2/Gasoline)
7 Intake Manifold with H_2-Rail
8 Boil-Off-Management-System (BMS)
9 Gasoline Tank
10 Pressure Control Valve

- GH_2 Feed Line
- Boil-Off Pipe
- Safety Blow-Valve Feed Line
- Exhaust Pipe BMS
- Air Inlet BMS
- Water Cooling Cycle
- Gasoline Pipe

used to replace gasoline or diesel fuel in an engine designed for those fuels, there would be a loss of fuel efficiency and engine effectiveness.

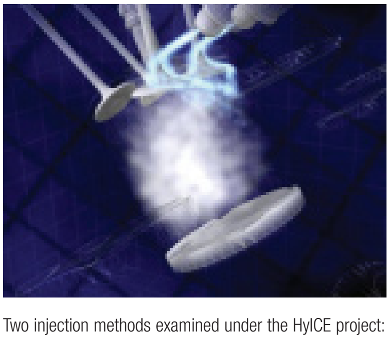
In order to properly take advantage of the characteristics of hydrogen as a fuel, a hydrogen-powered engine must be built from the ground up. This was the goal of the European Commission-funded Hydrogen Internal Combustion Engine (HyICE) research project, a three-year effort aimed at designing a clean automobile engine. This initiative led to the development of a hydrogen-powered IC engine that offers significant advantages in terms of cost and power as compared with other systems. The project team, coordinated by the BMW Research and Technology Group, comprised automobile manufacturers, automotive suppliers and two universities. Already, the group has shared its results and experiences with partners in the United States; in 2003, the United States and the European Union agreed to collaborate on speeding up the development of the hydrogen economy.

An important consideration for the project was the customization and improvement of appropriate simulation tools for the hydrogen-based combustion process, in order to support the future mass production of engines. ANSYS CFX computational fluid dynamics (CFD) software was selected as the main commercial CFD platform, because it already was employed by many of the project participants and because the software could be customized for the specific needs of the effort.

When designing the simulation model, special attention was paid to the specifics of hydrogen combustion. HyICE studied two different methods for fuel injection and, therefore, developed two different simulation models. For the cryogenic method, already in use as a bivalent solution, hydrogen was mixed with oxygen in the inlet port before it entered the cylinder, at which point it was compressed and ignited. In the direct injection method, hydrogen was injected directly and at high pressure into the cylinder and subsequently ignited. Hydrogen combustion is much faster than that of fossil fuels and occurs under higher pressures. Experts from ANSYS, Inc. implemented and tested different sets of models for their numerical stability and accuracy. These were compared with experimental data.

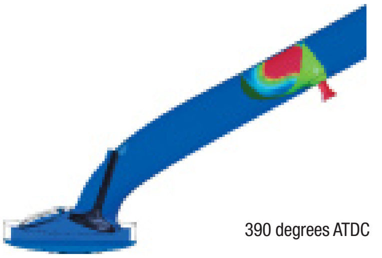
Because the ignition process takes only a few milliseconds to occur, the team developed a quasi-one-dimensional combustion/ignition model to simulate this behavior. A full 3-D simulation of the combustion process then was developed based on the solution provided by the ignition model.

The goal behind using CFD for the HyICE project was to accelerate the development process, though another goal focused on the design of a reliable and validated simulation solution for future development projects. These goals were achieved. The agreement of the simulation results with experimental data, especially in the areas of temperature and pressure distribution, was excellent. The extensions and modifications to the software made in the course of the HyICE project can be applied to conventional engines as well. ■

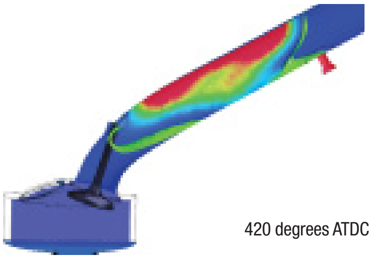


Two injection methods examined under the HyICE project: (top) direct injection of high-pressure hydrogen and (bottom) port injection using cryogenic hydrogen

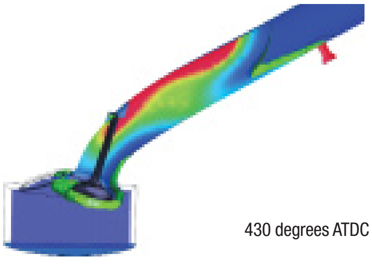
Image © BMW Group



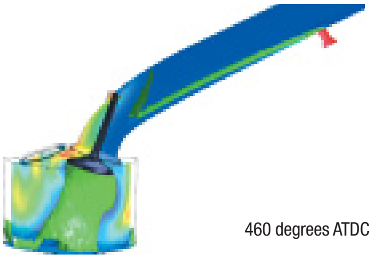
390 degrees ATDC



420 degrees ATDC



430 degrees ATDC



460 degrees ATDC

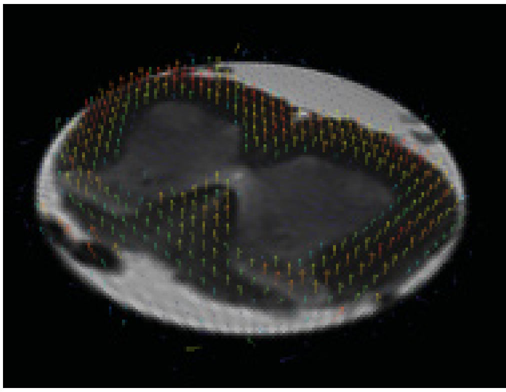
Cryogenic port injection plots show hydrogen mass fraction and motion of hydrogen as it flows into the cylinder during the intake stroke of the engine. As the intake valve opens and the piston drops down away from the valves, the piston motion draws the fuel air mixture in through the intake port.


Enter a new era with 3D
Why should sophisticated technologies be for experts only? At Dassault Systèmes we want to break with this tradition and establish 3D as a new universal language. Our passion for 3D inspires us to create software that is changing the way people see the world. www.3ds.com
DASSAULT SYSTEMES
See what you mean

Special Delivery

Researchers use simulation and medical imaging to explore new options for managing pain.

By Malisa Sarntinoranont, Xiaoming Chen, Jianbing Zhao and Thomas Mareci, University of Florida, U.S.A.



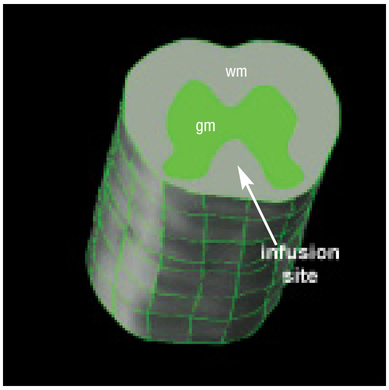
Maximum eigenvectors (identified by arrows in the image) identify the locations of maximum water diffusivity and preferred tissue transport for a fixed rat spinal cord. The red arrows represent aligned eigenvectors.

The latest therapeutic agents for chronic pain, spinal injury and other neurodegenerative diseases are characterized as macromolecular (large) proteins. Delivering these drugs at the site of action is gaining popularity. However, the transport environment in the spinal cord and other nervous tissue must be taken into account when designing direct infusion therapies.

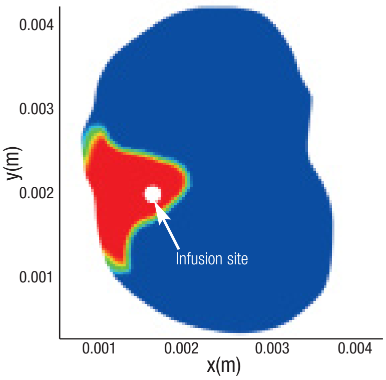
Since macromolecular drugs diffuse relatively slowly, transport factors affect the effectiveness of delivery greatly. The delivery of drugs through the spinal cord is dependent on a variety of factors, including variations in material properties and flow regions within the cord itself. Specialized analysis methods that correctly predict the related transport behaviors are required before one can develop general, and possibly patient-specific, delivery protocols.

By coupling medical imaging with computational fluid dynamics (CFD) analysis, a research group at the University of Florida in the United States recently developed methods for predicting the distribution of a drug tracer injected directly into the rat spinal cord [1, 2]. Traditional magnetic resonance imaging (MRI) was used to determine the geometry and structure of the spinal cord. Diffusion-tensor MRI (DT-MRI), which provides information on how water molecules spread through tissue, was used to determine the preferred and most likely transport directions in the cord.

Analyses of interstitial pressure, velocity and tracer distribution within the porous media in the spinal cord were performed using FLUENT software. An anisotropic hydraulic conductivity (K) was applied in the white matter, a transport region located at the periphery of the spinal cord, to model the flow through it. The magnitude of K was assigned based on experimental data [3]. DT-MRI technology was used to identify the direction of maximum water diffusivity, which, since it was assumed to be parallel to the local fiber orientation, was used to determine fiber tract directions. This alignment data was used to assign behavior properties to the model.



MRI-derived three-dimensional geometry of the rat spinal cord. White matter (wm) is in grey and grey matter (gm) is in green. Drug is delivered into the white matter, near the boundary with the grey matter.



Predicted albumin tracer distribution in the spinal cord 20 minutes after a 2.0 μl infusion. Red areas represent sites of higher concentrations.

Using FLUENT technology, the distribution of a small volume infusion of the tracer then was predicted. Convection-dominated transport along white matter tracts was found, and the preferred distribution was identified along the cord axis with little penetration into adjacent gray matter zones. These results correspond well with small volume distribution trends found experimentally [3].

A model of this type could be used further to analyze the effectiveness of different injection protocols, such as continuous versus discontinuous injections or the effect of injection site on drug distribution. Eventually, this type of image-based modeling effort may allow customized medical care that inherently factors in patient-specific physiological differences. ■

References

[1] Sarntinoranont, M.; Banerjee, R.K.; Lonser, R.R.; Morrison, P.F., A Computational Model of Direct Interstitial Infusion of Macromolecules into the Spinal Cord, *Annals of Biomedical Engineering*, 2003, 31, pp. 448–461.

[2] Sarntinoranont, M.; Chen, X.; Zhao, J.; Mareci, T.M., Computational Model of Interstitial Transport in the Spinal Cord Using Diffusion Tensor Imaging, *Annals of Biomedical Engineering*, 2006, 34, pp.1304–1321.

[3] Wood, J.D.; Lonser, R.R.; Gogate, N.; Morrison, P.F.; Oldfield, E.H., Convective Delivery of Macromolecules into the Naive and Traumatized Spinal Cords of Rats, *Journal of Neurosurgery (Spine 1)*, 1999, 90, pp. 115–120.

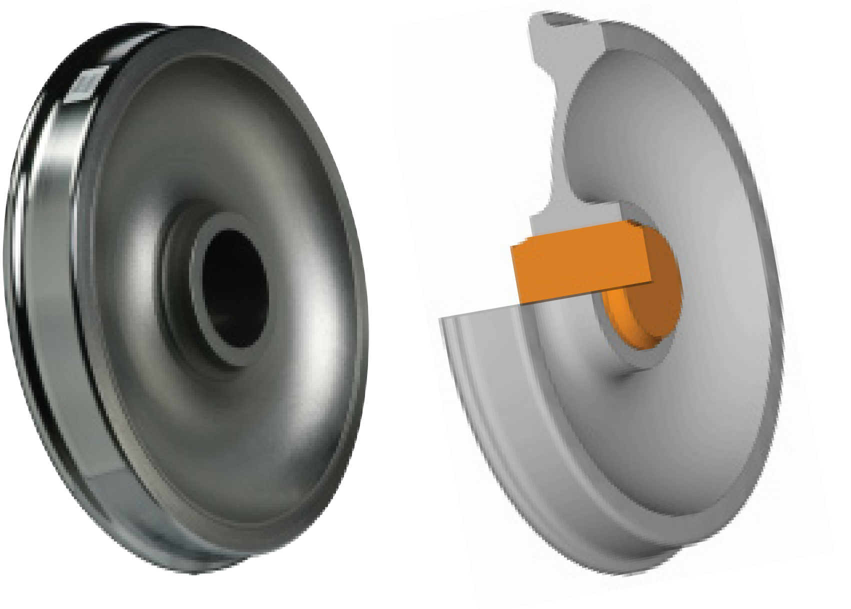
More Certainty by Using Uncertainties

Engineers apply probabilistic methods to historically deterministic problems.

By Kexiu Wang, Griffin Wheel Company, Illinois, U.S.A.

Rail cars carry enormous loads, often triple that of the largest 18-wheeler trucks. These loads pass entirely through the wheels, which not only bear the weight but are subject to a number of other structural, thermal and fatigue loads. Griffin Wheel Company, a division of AMSTED Industries in the United States, produces 90 percent of rail wheels for the North American railroad industry. The company recently applied probabilistic tools from ANSYS, Inc. to their wheel design process.

A basic freight car wheel is relatively simple, yet the wheels are subjected to extreme forces and, therefore, must withstand tremendous amounts of abuse. The wheel not only bears the load of the car, but also its tread surface is used as a brake drum, absorbing varying loads under constantly changing thermal conditions. In addition, the flange guides the train on the track, conveying lateral loads throughout the wheel. Although deceptively simple in construction, the multi-faceted character of the freight car's load environment makes for an extremely complex analysis.



Computer-aided design (CAD) model (right) of railroad freight car wheel (left)

Freight car wheels are solid steel castings. Heat-treating strengthens them, improves wear resistance and induces circumferential residual compressive stresses in the upper rim to prevent fatigue crack formation. Heat-treating, however, generates axial tensile stresses in the lower part of the rim, causing vertical split rim, a rare but catastrophic failure mode. Understanding the factors that can affect these types of stresses is essential in effectively optimizing wheel design.

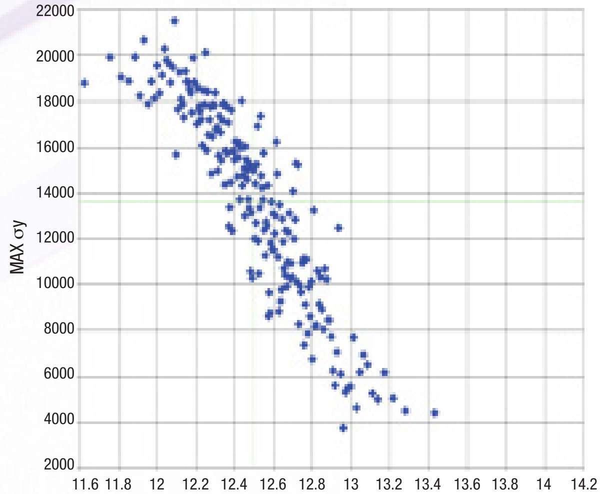
For an engineering analysis, many features are inherently variable and uncertain: operational loads, geometry, manufacturing processes, material properties and operational environments, as well as testing. These uncertainties lead to uncertainty in product development and manufacturing. The traditional deterministic design approach accounts for variations by using safety factors. But this approach does not account for the random nature of design parameters. Treating the various parameters as singly determined values decreases predictable reliability. Without measuring this reliability, performance levels become inconsistent. Moreover, since common practice assumes the worst-case scenario for each singly determined value, the resulting design is often less than optimal, and subsequent changes produce undetermined effects in other areas.

The probabilistic method makes use of statistical tools as a more reliable means to account for these multi-faceted uncertainties. During an analysis, parametric uncertainties are characterized statistically in terms of probability density functions (PDFs). These PDFs quantify the inherent risks in a system and allow evaluation of input parameter variations in relation to changes in output performances. Probabilistic analysis yields a more comprehensive understanding of the entire system, allowing engineers to develop a better understanding of product behavior in, and responses to, real-life conditions.

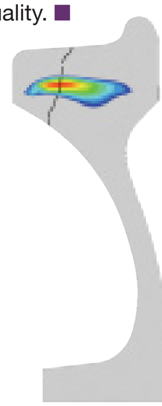
When used in simulation, once the random variations of boundary conditions, geometry and material properties are specified for a specific analysis case, the input variables are studied simultaneously by using statistical sampling methods. The parametric finite element analysis (FEA) model then is invoked repeatedly, performing deterministic analyses over the resulting input parameters.

Deterministic approaches have shown that the residual stress (from heat-treating) varies in relation to many parameters. To investigate the effects of different parameters in the heat-treatment process and to identify parameters that have the greatest impact on residual stress, Griffin engineers analyzed a CJ36 freight car wheel using the probabilistic tools from ANSYS. After performing a deterministic de-coupled thermo-mechanical analysis on a baseline model, engineers performed a Latin Hypercube sampling probabilistic analysis. This determined the variations in the residual stresses when given the uncertainty of the manufacturing process parameters, boundary conditions and material properties. The scatter plot showed that the residual stress was especially sensitive to creep.

The probabilistic analysis is being used to identify future steps needed for further optimization and eventually will lead to optimizing the Griffin wheel's residual stress field, thereby improving wheel reliability. The process illustrates how simulation technology from ANSYS can be used to understand production process uncertainties and related parameter variations in a manufacturing process, leading to increased product reliability and quality. ■



The scatter plot compares maximum axial stress to the stress exponent. Correlation coefficients show strong sensitivity of the stresses to creep.



Vertical split rim and contours showing the residual tensile stress

Probabilistic Analysis with ANSYS Workbench

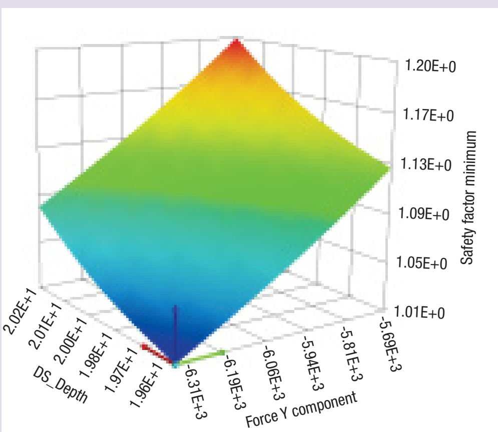
By Pierre Thieffry, ANSYS, Inc.

As a complement to the ANSYS Workbench environment, ANSYS DesignXplorer software provides a number of probabilistic analysis tools. Engineers use these tools to describe a parametric model in terms of statistical distribution functions for variations in the input parameters.

The technology uses two methods to estimate analysis variations. The Direct Sampling method, based on Monte Carlo sampling, requires a large number of simulations and can benefit from the parallelization techniques offered by products from ANSYS, Inc. The other method, called Design of Experiments (DOE), is based on response surfaces. DOE requires fewer simulations than Direct Sampling and builds an approximation of the system response from which probabilistic results are drawn.

Both methods analyze results variability and allow standard statistical analysis techniques (mean, standard deviation, kurtosis, etc.) as well as statistical sensitivity measures, with the latter actually identifying critical parameters driving the design.

ANSYS DesignXplorer software also provides information about the probability to reach a given performance. This data helps assess the risk of failure for a given design at a given target value, such as maximum stress, maximum displacement and minimum eigenfrequency. An equivalent Sigma level also is given, based on the Six Sigma quality criteria.



Variation of the simulation results with respect to design parameters

Out of **Harm's Way**

Engineers used simulation to design an innovative military gun turret.

By Eric Hobson, Precision Remotes, Inc., California, U.S.A.

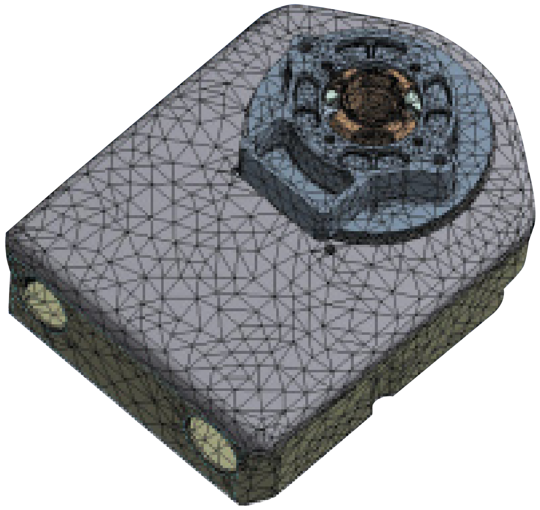
It's a chilling fact but a reality of modern warfare: Snipers attack military vehicles. This is particularly true of high-mobility multipurpose wheeled vehicles (HMMWVs or Humvees) that are in widespread use by the U.S. Army. To protect soldiers from such attacks, the Army is equipping some of these vehicles with roof-mounted rotating gun turrets that are remotely controlled by personnel from inside the vehicle. This design strategy reduces the risk to soldiers by taking them out of the direct line of fire, enabling personnel to more safely assess and deal with the situation.

One of the leaders in developing this technology is Precision Remotes, Inc. in the United States, which designs and manufactures proprietary products for the military, government and industrial security/surveillance industries. The engineering firm offers the telepresent rapid aiming platform (TRAP) for armed applications as well as security monitoring of military installations, nuclear power plants, oil pipelines and other facilities that could be vulnerable to terrorist threats.

The core weapon platform from Precision Remotes is the T-250, which is designed to handle either a sniper rifle or a machine gun. A complementary sniper detection system identifies incoming muzzle fire and sends a signal that automatically rotates the turret in the appropriate direction for faster aiming by the operator. Originally developed for operation in a static location, the design of the T-250 was later adapted for use on mobile vehicles and named the M-360.

Modifying the system to become a vehicle-mounted product involved

The roof-mounted M-360 weapon system was designed using finite element analysis (FEA) to maximize the assembly's stiffness and strength while also minimizing weight.



ANSYS Mechanical modeling features and bonded contact elements were used to efficiently generate an accurate mesh representation of the M-360 housing.

Deformation of the M-360 unit under extreme loading conditions, represented by the FEA results shown here

developing a drive system to give the M-360 unit full 360-degree rotational capability, rather than the static application specification for an azimuth (left–right) movement range of 60 degrees. The new dual-drive used two motors operating in opposite directions in order to provide high-speed changes in orientation and virtually no delay when decelerating to a stop, which resulted in a highly accurate position for aiming.

Within a limited time frame, engineers had to satisfy stringent technical requirements for the M-360 system and its new drive. The frame and housing were required to have the strength to effectively support the entire structure — including the weapon and ammunition — while undergoing the extreme loading conditions encountered when the vehicle to which it would be mounted maneuvered rapidly on bumpy terrain in a hostile environment. Engineers were particularly concerned about deformation and vibration of the unit under these loading conditions, since deflection adversely affects weapon accuracy. In addition, the unit was required to be light enough so it could be supported by the vehicle roof structure and compact enough to mate with a pre-existing receptacle on the HMMWV.

Using ANSYS Mechanical technology, the engineering team evaluated the structural behavior of the M-360, particularly the frame and housing, which were expected to experience the greatest load. Computer-aided design (CAD) geometry from Autodesk® Inventor™ was imported directly into the ANSYS Mechanical application. Having the CAD assembly features and properties in place on import was a major advantage for the design process: It assured accuracy for both the application of loads and the representation of the relative positions, connection points and different material properties of individual components, including the aluminum rotating turntable and housing as well as the stainless steel turret receptacle and dual-drive system.

The bonded contact element capability within ANSYS Mechanical software was useful in modeling the structure and accurately representing the ways many interconnected parts interact with one another. This software automatically recognizes bonded contact between the welds joining individual parts and allows for different material properties and dissimilar meshes of contacting parts, thus avoiding the difficult and time-consuming task of manually adjusting mesh densities and element types. Using these features, engineers efficiently proceeded through an iterative process of distributing material in the structure to maximize the stiffness and strength of the frame and housing without appreciably increasing the weight of the system.

In this way, the ANSYS Mechanical solution enabled the Precision Remotes team to arrive at a refined design of the M-360 in only three weeks — sufficient time to build and test a physical prototype, verify the integrity of the final system configuration and deliver the product within the tight time frame required. In the end, product quality for the delivered unit met all expectations. An added bonus was that the newly designed M-360 weighed less than 20 pounds, far lighter than competing systems.

In order to maintain an edge in the defense industry, customer responsiveness and rapid time-to-market are critical. Companies can succeed only by delivering excellent products that meet real-time needs faster and more cost-effectively than the competition. Simulation software from ANSYS is a key element in Precision Remotes' ability to meet those goals and strengthen their leadership position. ■

Designing Out the Weakest Link

Engineering simulation validates the design of a mooring system component, a critical wheel/chain assembly that holds ships in place during oil and gas operations in the North Sea.

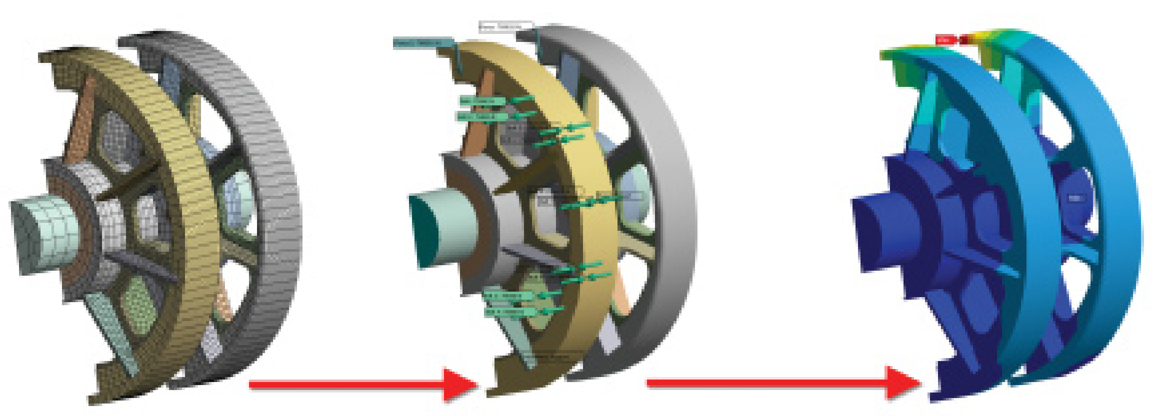
By Joel Thakker, Integrated Design & Analysis Consultants Ltd., Croydon, U.K.

Floating production, storage and offloading (FPSO) vessels take oil or gas from deepwater offshore petroleum platforms, process it and store the material until it can be offloaded onto waiting tankers or sent through a pipeline. To maintain a stable, fixed location — even in rough waters — these huge ships have on-board winch systems that handle mooring chains, which can be hundreds of feet in length and weigh thousands of tons. Critical to the winch system is a central assembly called a gypsy wheel that, together with hydraulic chocks, grippers and interlocks, controls the release, retraction and tensioning of the mooring chain.

To validate the structural integrity of a newly designed gypsy wheel, Whittaker Engineering in Scotland, a company that provides engineering design to the offshore oil and gas industry, approached the engineering consulting firm Integrated Design & Analysis Consultants (IDAC) Ltd. The primary reason for developing a new design stemmed from an earlier design failure that resulted in a mooring chain dislodging from the assembly and consequently sinking to the ocean bed. IDAC was tasked to analyze stresses and deformation associated with the gypsy wheel under both preload and operating load conditions. The cause for the failure fell outside the scope of this analysis.

The challenge in this project was in evaluating whether the wheel and chain locker components of the gypsy wheel, which hold the retracted chain, could withstand various loading conditions as well as whether the size and weight of the assembly could be minimized without compromising structural integrity. The design would have been impossible to load-test safely and difficult to analyze precisely by conventional hand calculations. For these reasons, engineering simulation provided a useful evaluation pathway. Upon validation of the design, the new gypsy wheel assembly was to be installed on board the *FPSO Captain*, a vessel operated by Chevron Texaco in the North Sea.

The simulation process began with the finite element analysis (FEA), in which the geometric model of the new gypsy wheel was meshed using



Overview of the mechanical analysis of a gypsy wheel (left to right): The geometric model of the gypsy wheel was meshed, loading of the wheel was defined and a mechanical simulation was executed in order to validate the structural integrity of the assembly.

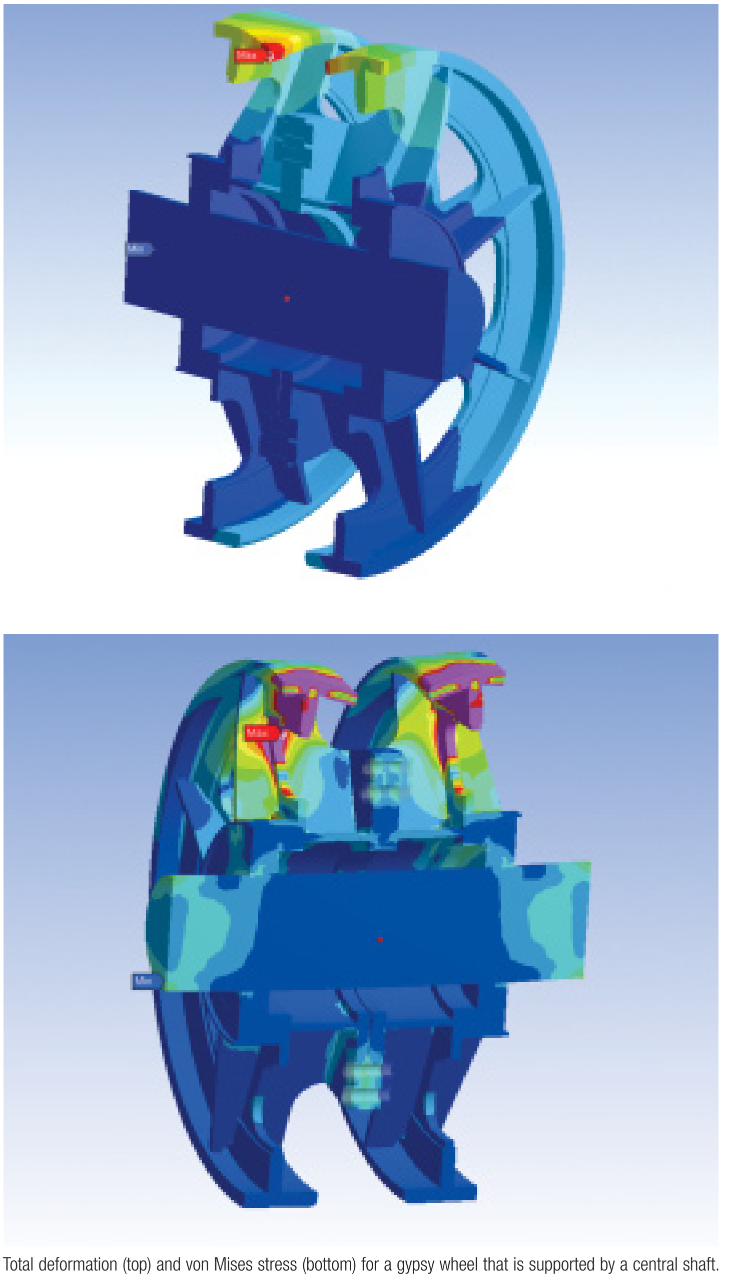
ANSYS Mechanical software. Nonlinear contact elements were generated between the shaft and the wheel, and bonded contact elements were generated between the bolts and the wheel to simulate the bolt preload. As part of the meshing capabilities, contact elements automatically detect contact points and allow for dissimilar meshes between contacting parts. In addition, the mesh is configured to account for joined parts, thus avoiding the task of manually adjusting mesh densities and selecting element types — a process that can be time-consuming.

In the new gypsy wheel design, the mooring chain was guided into the chain locker, thereby relieving the assembly of excessive loads during the mooring process and intrinsically reducing the chances of failure. Per design mandates, the frame of the gypsy wheel was designed to withstand transient and impulse loads for a small period of time. The transient loading cases were conducted to mimic operating conditions, while the impulse loading was done to design the structure for accident scenarios. (An accident had caused the system to fail in the first place, thus demanding FE structural evaluation.) In addition, due to the limitations associated with access and mechanical handling, effort was put into keeping the weight and size of the assembly at a minimum without compromising the structural integrity of the assembly.

Three separate load cases were studied as part of the investigation. The simplest load case was used to evaluate whether the assembly suffered damage under a pre-loading scenario. The other two cases analyzed a normal and angular downward force independently as well as in combination with an out-of-plane force. In each case, the simulations confirmed that the new gypsy wheel design could undergo the pretension and operational loads, with resulting deformation and stresses falling well within the design parameters.

In the end, structural analysis using ANSYS Mechanical software effectively evaluated the new design of the gypsy wheel under the various loading conditions and failure modes. Simulation overcame the inability to perform load tests safely as well as the difficulty of time-consuming and less accurate manual calculations.

IDAC then worked with Whittaker Engineering to produce an engineering package that was acceptable to the ship's certifying authorities. Subsequently, Whittaker Engineering manufactured and installed the eight new chain gypsies, which currently are in service in the North Sea. ■



Total deformation (top) and von Mises stress (bottom) for a gypsy wheel that is supported by a central shaft. From the von Mises plot, it can be seen that the area that supports the chain on the top of the wheel experiences high stresses, as does the shaft that supports the wheel.

Image ©Camera 4/Thonfeld.

Going for the **Gold**

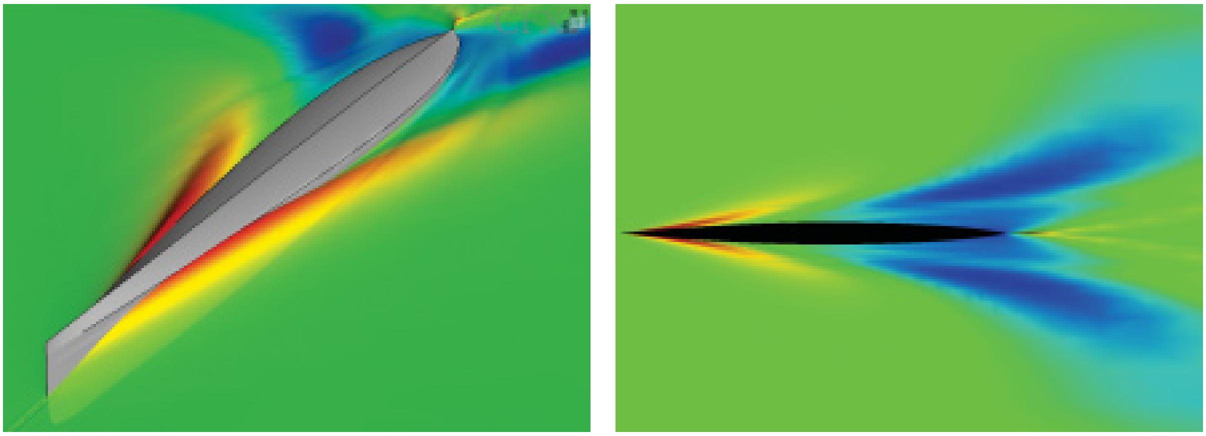
Simulation helps design low-drag canoes for Olympic-medal performance.

By Nicolas Warzecha
Institute for Research and Development of Sports Equipment, Berlin, Germany
Andreas Spille-Kohoff
CFX Berlin Software GmbH
Berlin, Germany

Competition among world-class athletes at the Olympics has become so intense that tiny variations in performance mark the difference between the gold medal winner and the also-rans. Relying heavily on computer simulation to reduce the air resistance of their bobsleds, the German national team leveraged their win in that sport to emerge triumphant in the 2006 Winter Olympics at Turin, Italy. Germany edged out the United States in overall gold medals, 11 to 9, and in total medals, 29 to 25.

The work in optimizing the performance of bobsleds was carried out by engineers at the Institute for Research and Development of Sports Equipment (known by its German acronym of FES) in Berlin, one of the world's leading centers for the development of sports equipment. Today, FES engineers are hard at work designing skiffs, canoes and sailboats that they hope will help produce a similar triumph at the 2008 Summer Olympics in Beijing, China.

To gain an edge in the canoe competition, FES engineers are using ANSYS CFX fluid simulation software to simulate the performance of various canoe designs. They selected this technology largely because it provides the powerful CFX Expression Language (CEL), which allows users to create their own physical models quickly from within the user interface, to add new variables, and to

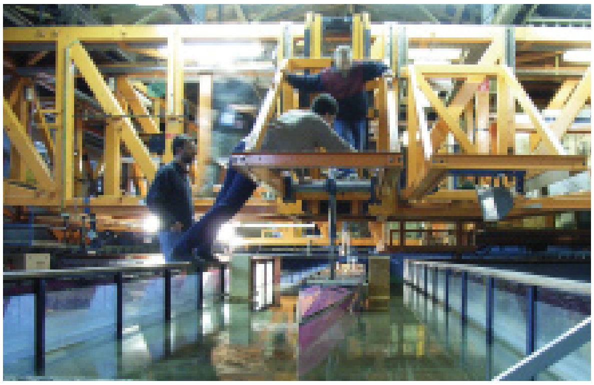


CFD results depict the wave pattern as it develops along a canoe body. Wave height is indicated by color, with blue denoting lowest areas and red indicating the highest areas.

define property relationships and boundary condition profiles. CEL goes beyond similar languages by allowing FORTRAN™ routines to be called, allowing other FORTRAN applications to be coupled to ANSYS CFX software.

The engineering team at FES began by simulating experiments involving towing a canoe, which eliminated the challenge of simulating the effect of the paddlers' strokes on the boat's motion. They used ANSYS ICEM CFD Hexa software to create a block-structured grid model with 3 million elements and used the free-surface multiphase model of the ANSYS CFX product to analyze both the motion of the water and the air trapped by the movement of the boat and the water. This simulation showed a very good correlation with the drag measured in the towing experiments. All cases were simulated in parallel on a 64-bit Linux® cluster, whose installation was supported by CFX Berlin. The best results were acquired by running the CFX solver on 10 to 20 mesh-partitions, depending on the size of the grid. Using this approach, a transient simulation representing a 10-second real-time interval for a moving boat could be performed in one to two days.

The wetted surface of the canoe in this model did not match the experiments, a conclusion that was expected since this simplified model did not account for the forces and moments acting on the boat. So FES engineers simulated the boat at its position with the ANSYS CFX solver to calculate these forces and moments and estimated the new position of the boat — that is, sink and trim values. They continued with a series of manual steps that slowly converged to a final position showing good agreement with experiments. Once they had determined that this approach provided realistic results, FES automated the analysis, with the assistance of CFX Berlin, by writing CEL expressions and some pieces of FORTRAN code that performed all of these steps in the same way but much faster and automatically within the ANSYS CFX solver. With this approach, it was possible to evaluate and compare the performance of several alternative designs.



Experimental testing area used to study water flow and wave formation around a boat as well as total drag

The analysis performed by FES provides the drag as well as complete information on the movement of the water around the boat, the position of the boat and the forces acting on the boat. In particular, computational fluid dynamics (CFD) makes it possible to measure the bow wave, aft wave and wake of the canoe to a high level of precision. Simulation provides engineers with good indications of what is causing drag in a particular design and what aspects of the design they should change to improve it.

In the past, designing these boats was based largely on trial-and-error prototyping. Using simulation, FES engineers were able to quickly design a new skiff that ANSYS CFX software predicts will provide a 3 percent improvement in drag. A prototype of this boat is currently under construction. After completion of the prototype, physical testing will be used to verify the simulation results. In the meantime, FES engineers are working to expand the scope of CFD analysis to analyze other effects that are difficult or impossible to measure, such as the effect of initiating paddling forces at different times and variations in the velocity of the boat. Including these effects in the analysis may make it possible to advance the performance of canoes and athletes to even higher levels. ■

Something **in the Mix**

Researchers use the Poincaré plane method to obtain quantitative time scale information from CFD simulations.

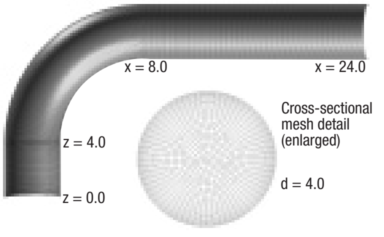
By M.N. Godo, Intelligent Light, New Jersey, U.S.A.

Mixing processes are involved at some level in nearly all chemical manufacturing processes. They are fundamental to the successful operation of combustion-driven systems. Today, many computational fluid dynamics (CFD) practitioners in the chemical process industry are able to use simulation to obtain detailed insight into overall performance of their process equipment. However, it is still difficult to relate CFD data to the effective management and control of a particular process. In addition, the cost of production delays due to sudden, unexpected changes in product quality provides strong motivation to understand the impact and relevance of CFD studies that are focused on these areas.

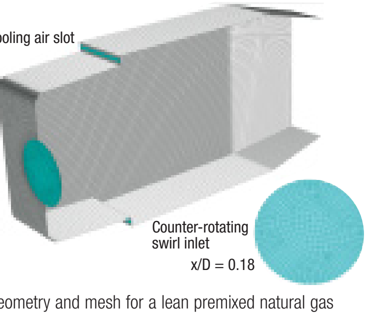
While CFD continues to be more accessible to analysts, managers and operators, problem complexity and sophistication also has increased. Relating flow data, such as mixing time scales to device performance, now is a major challenge. Flow visualization methods, which use iso-surfaces and cutting planes, can be used to help visualize flow topologies in an ad-hoc way. Streamlines and time-dependent streaklines also are effective at elucidating flow patterns. However, these approaches are limited in that they provide very little quantitative information on how flow patterns affect overall performance.

At Intelligent Light in the United States, engineers turn to the Poincaré plane method to obtain quantitative time scale information from CFD simulations. Poincaré planes, placed at various locations within a flow domain, display the time and locations at which streaklines cross these planes. Time scales obtained from these plots relate directly to how effective a mixing tank is or how efficiently a furnace or incinerator can be run. Being able to see time scales within mixers and combustion chambers offers much easier interpretation of the CFD data for everyone involved in the production process. For instance, Poincaré planes showing holes or concentric rings indicate flow regions that are strongly segregated, that is, poorly mixed. In general, this behavior is undesirable; knowing exactly where this occurs in a process vessel is a key step in resolving performance problems.

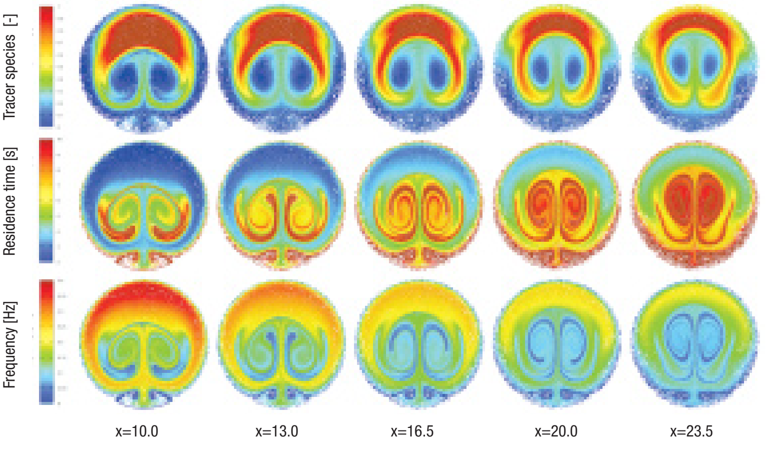
Since its introduction, the Poincaré plane method has been applied to mixing studies [1] and fundamental flow problems [2]. To create Poincaré planes, FIELDVIEW [3], a CFD post-processing tool from Intelligent Light, is used to interpret CFD simulation results that are generated by FLUENT



Geometry and mesh for a laminar flow case through a 90-degree bend



Geometry and mesh for a lean premixed natural gas power turbine CFD case, based on the General Electric Aircraft Engines LM6000 engine



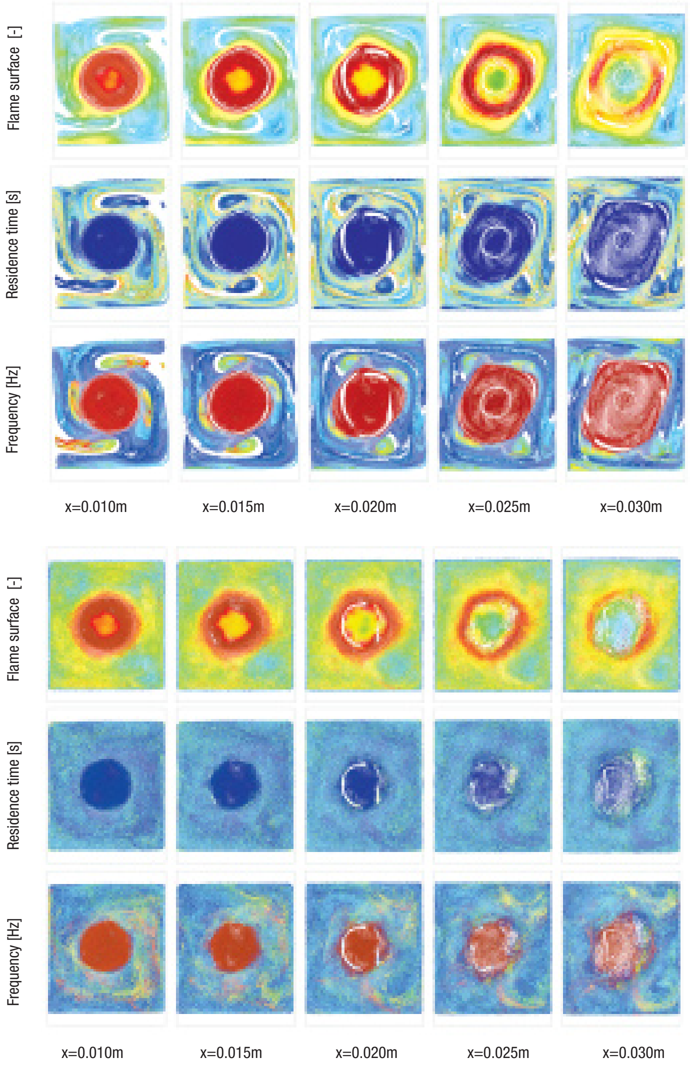
A series of Poincaré planes in the downstream section of a 90-degree tube bend; rows are colored by tracer species, residence time and frequency (1/residence time), respectively, and x represents location in the tube after the bend, which ends at x = 8.0.

software. FIELDVIEW is able to read data exported directly from FLUENT tools as well as from the ANSYS CFX product. Using the velocity field information from the CFD simulations, FIELDVIEW calculates the large number of streaklines necessary to obtain accurate results for Poincaré planes. Because of the repetitive, quantitative tasks needed, the FIELD-VIEW programming language, FVX™, was used to automate streakline trajectory calculations, identify streak-line intersections with the Poincaré planes and visualize the final results.

Of two cases studied, the first simulated simple laminar flow through a 90-degree bend. The second case was a fully validated flow calculation for a lean premixed natural gas power turbine, based on the General Electric Aircraft Engines (GEAE) LM6000 engine. A counter-rotating swirl inlet boundary condition was provided directly by GEAE. Both Reynolds-averaged Navier–Stokes (RANS) and large eddy simulation (LES) turbulence models were calculated using FLUENT tools, and GAMBIT software was used to create the meshes for both cases.

For the 90-degree bend case, it was observed that flow details based on either the residence time or frequency are highly structured, and they exhibit significant local differ-ences as the fluid is rolled up by the action of the vortices. Notably, the fluid in the center of the tube, which has a residence time roughly five times that of the flow near the upper section, has a significant impact on mixing effec-tiveness, as the flow has clearly become quite structured.

Within combustion chambers, a key goal in design assessment is to quantify mixing rates, particularly at time scales that are on the same order of magnitude as the chemical reaction and energy and mass transfer rates. For the RANS turbine case, areas of strong flow isolation are clearly seen near the inlet. In addition, the RANS solution exhibits significantly more structure than the LES solution. Time scales, observed in the Poincaré planes for the RANS case, cover a wide range. This strongly affects the extent of combustion predicted by this simulation. In contrast, Poincaré planes for the LES case show a very high level of chaotic mixing on a fine spatial scale. Apart from the region immediately downstream from the swirl inlet, there were no significant differences in either the residence time or frequency, and the central flame envelope is nearly gone at the farthest downstream plane for the LES case. Mixing time scales for the LES case are expected to provide more realistic predictions of the combustion physics in this particular case. ■



A series of Poincaré planes used in the analysis of a gas turbine case (specifically the GEAE LM6000) modeled using FLUENT software: (top) RANS turbulence model and (bottom) LES turbulence model

The author would like to express gratitude to Greg Stuckert and Graham Goldin of ANSYS, Inc. for providing the GEAE LM6000 combustion case and for sharing their considerable knowledge of best practices concerning the setup of the partially premixed combustion routines as well as the parameters for the large eddy scale calculations.

References

[1] Zalc, J.; Szalai, E.; Alvarez, M.; Muzzio, J., "Using CFD To Understand Chaotic Mixing in Laminar Stirred Tanks," *American Institute of Chemical Engineers Journal*, 48(10), 2002, pp. 2124–2134.

[2] Shariff, K.; Leonard, A.; Ferziger, J.H., "Dynamical Systems Analysis of Fluid Transport In Time-Periodic Vortex Ring Flows," *Physics of Fluids*, 18(4) , 2006, pp. 047104-1 – 047104-11.

[3] FIELDVIEW, CFD Postprocessor, Version 11, Intelligent Light, Rutherford, NJ, 2006.

Cluster Computing with Windows CCS

New clustering technology from Microsoft speeds up engineering simulation.

By Barbara Hutchings, ANSYS, Inc.

The disciplines of computer-aided engineering (CAE) and high-performance computing (HPC) have been closely aligned and interdependent since the 1970s, when ANSYS, Inc. was founded. As software and hardware technologies have evolved, engineers who conduct simulation analysis have been among the beneficiaries. Recent advances in HPC have been particularly valuable, bringing down the cost of entry for small workgroups in need of large-scale computing capacity. In particular, cluster-based computers — based on x86/64-bit processors from Intel® and AMD — now represent over 50 percent of HPC solutions and provide enormous computing capacity for a fraction of the cost of previous-generation solutions. Working with Microsoft® and other partners, ANSYS, Inc. now is making clusters a more viable solution for Windows®-based customers through support of the Microsoft Windows Compute Cluster Server 2003 (Windows CCS) operating system.

The Argument for Clusters

Engineers who perform simulations in support of product development are well versed in the business drivers that make clusters attractive. Simply put, more computing capacity increases productivity along with the value that simulation brings to the product development process. By reducing turnaround time, increased parallel computing capacity helps ensure that simulation results are available in a time frame that can impact engineering decisions. By enabling larger and more detailed simulations, computing systems with more memory (RAM) yield more accurate and more reliable results. Finally, by increasing throughput, a larger computing capacity enables the engineering team to simulate multiple design options while meeting schedule requirements. Clusters provide all three benefits — parallel speedup, large memory availability and capacity for high throughput — in a form that can be expanded over time as simulation needs expand.

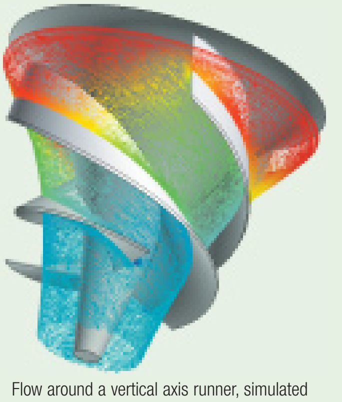
Given these benefits, it is not surprising that clusters are now the dominant platform for computational fluid dynamics (CFD) simulations using ANSYS CFX or FLUENT software, since these packages have, for many years, been designed for parallel speedup on clusters. More recently, with the release of version 11.0 technology from ANSYS, clusters have become a much stronger solution for finite element analysis (FEA) simulations as well. The new distributed memory solver in version 11.0 provides improved parallel scale-up. In addition, clusters are being used to increase throughput for parametric FEA analysis.

Early Adopters

Alden Research Laboratory, Inc., based in Massachusetts, U.S.A., is an acclaimed fluids flow engineering and environmental laboratory providing analytical, computational and physical flow modeling services. When the CFD team at Alden wanted to expand their analysis capacity, they turned to a cluster running FLUENT 6.3 software on Windows CCS. “We needed to increase our computing power in order to increase the number of FLUENT simulations we perform as well as to consider larger, more detailed models,” said Dan Gessler, Alden’s director of numeric modeling.

Using FLUENT software, Alden engineers simulate flow in advanced hydroturbine designs. For example, the Alden/Concepts NREC turbine team used flow modeling to maximize generating efficiency of the fish-friendly turbine. The unique turbine design has the lowest fish mortality for turbines in its class.

According to Charles Ulrich, Alden’s IT manager, “The ability to deploy a cluster using Windows CCS was very attractive for us, as it leverages our expertise and fits into our current computing environment. The deployment was quite smooth: We had our cluster up and running FLUENT software within two weeks. The integration of FLUENT with the Microsoft Job Scheduler is especially valuable, giving us the ability to manage and monitor multiple simulations on the cluster.”



Flow around a vertical axis runner, simulated using FLUENT software and Windows CCS
Image courtesy Alden Research Laboratory, Inc.

Windows Compute Cluster Server 2003

Windows CCS was released by Microsoft in 2006 to enable cluster computing within a Windows environment. Windows CCS is based on the Windows Server 2003 64-bit Standard Edition and leverages familiar Windows technologies, such as Active Directory, to provide authorization and authentication services on the cluster. In addition, Windows CCS provides cluster management utilities for deploying and administering the cluster as well as a built-in job scheduler to control and manage multiple tasks on the cluster. The combination of support for clustering and 64-bit memory addressing has made Windows CCS a very viable option for engineers using products from ANSYS who want to leverage their existing Windows infrastructure and expertise.

A typical cluster configuration involves one or more client systems (for example, desktop workstations) running the ANSYS Workbench platform. These clients submit compute tasks (solver jobs) to the cluster via the ANSYS Remote Solve Manager (RSM) and the Windows CCS Job Scheduler running on the cluster head node. For software not yet integrated within the ANSYS Workbench environment — such as the FLUENT 6.3 application — the process is very similar, with the FLUENT GUI running on the client systems and solve requests submitted to the Microsoft Job Scheduler via a new FLUENT launcher panel. Both version 11.0 from ANSYS and FLUENT 6.3 packages are fully integrated with the Windows CCS Job Scheduler, providing off-the-shelf management of jobs on the cluster.

Performance

The performance of ANSYS 11.0 and FLUENT 6.3 products on Windows CCS has been documented by Hewlett-Packard, and the results are very good. For FLUENT, parallel scaling is nearly linear with the number of processors on a correctly sized cluster and similar to performance on equivalent clusters running Linux®. Figure 1 shows performance of a typical FLUENT simulation involving 3.6M finite volume cells as the cluster size increases up to 16 processors (32 cores). As CPUs are added to the cluster, the simulation speed scales well — with a speedup of roughly 75 percent of ideal scaling on 32 cores. ANSYS 11.0 scaling on Windows CCS is shown in Figure 2, in which the benchmark suite yields a range of speedups depending on the solver and physics involved.

ANSYS and Microsoft

Microsoft and ANSYS have worked closely to ensure that Windows-based clusters are a strong solution for engineers using solutions from ANSYS, Inc. This engagement began and continues in the technical arena, with support from Microsoft for porting and tuning applications from ANSYS on Windows CCS. Feedback from ANSYS helps to define requirements and improve the combined offering. The resulting performance makes Windows CCS an excellent choice for expanding simulation capacity.

The two companies also are working together at customer sites, building a combined understanding of the details required to successfully deploy software from ANSYS on Windows CCS. Working with Microsoft, ANSYS has developed detailed guidance for customers on sizing the cluster, setting up the cluster and deploying engineering simulation software on the cluster with connection to desktop clients. Despite its very attractive price/performance ratio, cluster technology has a reputation for being a challenge to implement. By teaming up, ANSYS and Microsoft have improved their ability to respond promptly to questions and problems that customers encounter. Both organizations also are working with original equipment manufacturers (OEMs) — including Hewlett-Packard, IBM®, Dell®, SGI® and Sun®, as well as system integrators and resellers — to help streamline the delivery of complete Windows CCS solutions to engineering simulation customers.

For more information, email windowsccs@ansys.com.

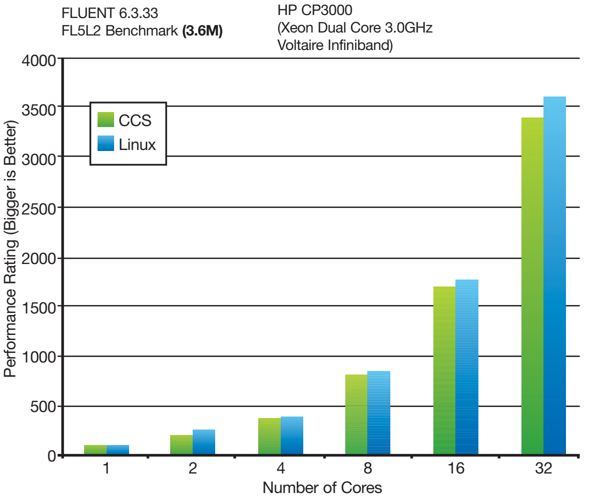


Figure 1. Performance of FLUENT 6.3 software using the HP-MPI (message passing interface) on a standard benchmark simulation of flow around a simple automotive sedan shape. The performance rating is comparable for Windows CCS and Linux. Results courtesy Hewlett-Packard.

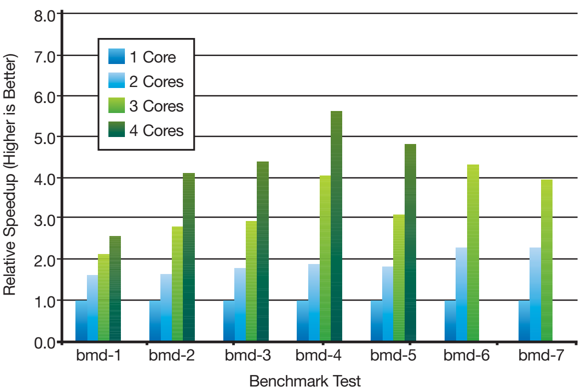


Figure 2. Performance of ANSYS 11.0 benchmarks as the processors count increases up to eight cores. The new Distributed ANSYS solver scales well on clusters. Results courtesy Hewlett-Packard.

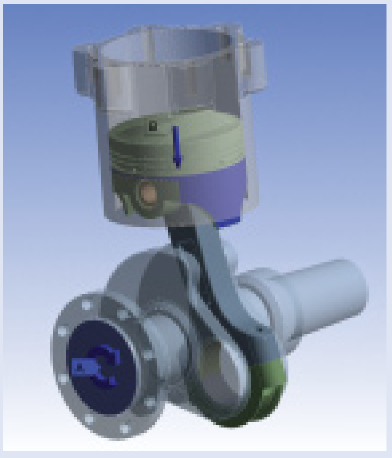


Figure 1. Flexible Dynamic analysis setup with all nine original assembly parts as rigid components

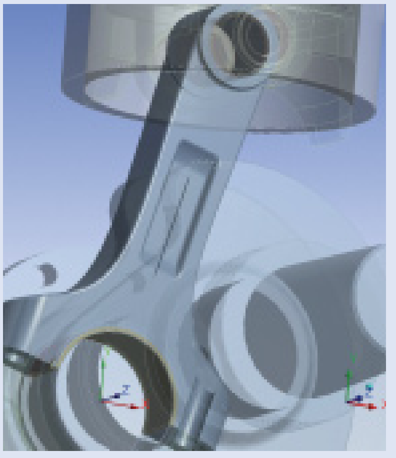


Figure 2. System-level analysis with joints defined

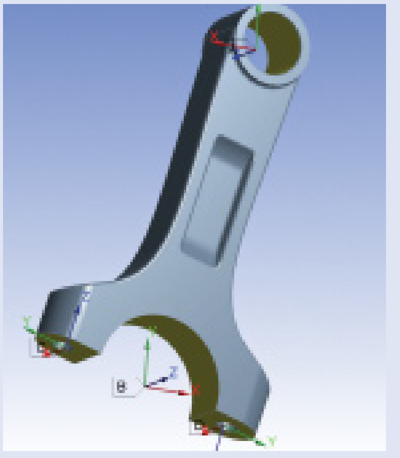


Figure 3. Reference Coordinate System location and orientation

Component Mode Synthesis in ANSYS Workbench Simulation

CMS superelements provide flexibility of simulation models while reducing the number of degrees of freedom for highly efficient solutions.

By Sheldon Imaoka, ANSYS, Inc.

At ANSYS Workbench 11.0, the ANSYS Rigid Dynamics add-on module enables users to define joint connections in complex kinematic assemblies.

This ability to model rigid and flexible parts in ANSYS Workbench Simulation via joints is beneficial since the rigid parts are modeled with mass and rigid links (that is, rigid contact elements), thus providing load transfer capabilities and allowing users to evaluate system-level performance with less computational cost. However, the solution time is dictated by the size of the flexible mesh, and in some cases the rigid assumption may not be accurate enough for specific applications.

A better approach in these cases is the use of Component Mode Synthesis (CMS) superelements, in which the flexibility of the model is retained yet the number of degrees of freedom (DOF) is reduced, thereby providing efficient solutions.

A superelement has a reduced number of DOF compared with the "full" model, yet it still can accurately model the flexibility of structures. Superelements can be created by regular substructuring or by component mode synthesis, in which DOF at interface nodes are retained while all other DOF are eliminated by condensation of the matrices. CMS appends a regular substructure with generalized coordinates, thereby improving the accuracy of the superelement response in dynamic applications.

Since CMS superelements can be used in large-deflection nonlinear analyses, they are especially attractive for nonlinear transient problems. This is because of the low number of DOF (that is, interface nodes and generalized coordinates) and the accurate dynamic representation. CMS superelements, however, also can be used in static, modal, harmonic and response spectrum analyses.

If contact is used with superelements (regular substructure or CMS), the number of interface nodes increases dramatically, depending on the size of the contacting region. Hence, CMS superelements should be used in Flexible Dynamic analyses in ANSYS Workbench Simulation with joints. Joints are defined at a single node, so the number of interface nodes used in CMS is reduced to a bare minimum.

Example Case

A sample Autodesk® Inventor® assembly was used for this example. A Flexible Dynamic analysis was set up with all nine parts as rigid, as shown in Figure 1. Interaction between the parts was defined by joints using rigid behavior for the associated surfaces, and two Joint Conditions were used for loading of the sample engine.

In a separate model, the connecting rod was set up in order to create a CMS superelement. Figure 2 shows the original system-level analysis with the joints. The individual model needs to reproduce the Joint Reference

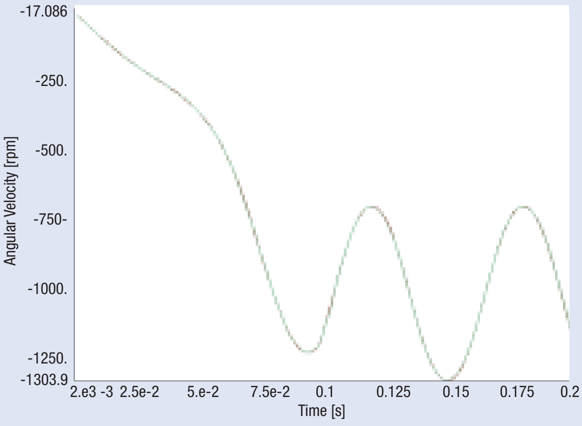


Figure 4. Comparison of simulation results for the rigid and CMS cases match for low loading. The green line indicates CMS results while the red line represents the rigid case results.

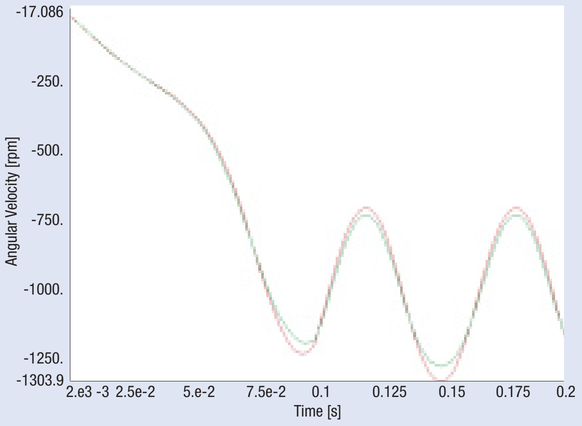


Figure 5. For higher loading, the rigid case and the CMS one differ due to the flexibility of CMS superelement components. The green line indicates the CMS results while the red line represents the rigid case results.

Coordinate System location and orientation with a Remote Displacement support, as shown in Figure 3. With that completed, the model can be meshed, and a superelement can be created.

To accomplish the generation of the superelement, a Commands object is inserted that selects constrained nodes (from the Remote Displacement support), deletes the constraints and designates those nodes as master DOF. A CMS generation pass is performed with a user-specified number of requested modes, and the resulting file.sub is the superelement file needed for the system-level analysis.

This procedure is repeated for each rigid part that will be converted to a superelement. Note that since multiple superelements may exist, each .sub superelement file should be renamed to have a unique name. Lastly, the user should ensure that the same unit system is selected as is used in the system-level assembly.

Once the superelements are generated, they can be incorporated into the system-level model. The author prefers doing this by adding two Commands objects, one under the part to be replaced and another under the Flexible Dynamic branch. The first has a single line, PART1_ = MATID, allowing the user to reference the element type ID a priori. The second Commands object would delete all rigid links of the rigid part except one, add the superelement and then couple coincident nodes together.

The rigid body is connected to joints via rigid links (that is, rigid contact elements). The reason the author prefers to delete all but one rigid link is so post-processing can be conducted as usual. In ANSYS Workbench Simulation, the original rigid mass is used to track the position of the part, so if only one of the rigid links is left, the mass element moves with the connected joint, and the user still can visualize the response of the system.

Figures 4 and 5 show relative angular velocity at the crankshaft–connecting rod joint for both the rigid assembly and the assembly that used CMS superelements to represent the piston, connecting rod and crankshaft, respectively. For relatively low loading, as illustrated in Figure 4, both results match well, as expected. For higher loading in which there is some relative deformation, as shown in Figure 5, the rigid-only and CMS cases start to show differences due to the flexibility of the parts.

It is important to note that the solution times for both system-level runs were relatively quick. The rigid-only case ran for 303 iterations, which required 19 seconds on a 3.2 GHz PC. The assembly with three CMS components ran for 1,340 iterations that required a total of 191 seconds. The CMS assembly case required more iterations to account for the flexibility behavior that was included in the analysis with the additions of the three CMS superelement components.

Once overall results (deformation, joint results, spring results) are reviewed, the deformation, stresses and strains for each part can be reviewed by expanding the superelement solution, then using "Tools menu > Read ANSYS Result Files." The user can expand either results at particular points in time or all of the results, then post-process as usual using the ANSYS Workbench platform, as shown in Figure 6. ■

Contact the author at sheldon.imaoka@ansys.com for the full article from which this was excerpted as well as demonstration input files.

Figure 6. Results, with CMS parts, are post-processed as they normally would be in ANSYS Workbench Simulation.

Accelerating to **Convergence**

ANSYS VT Accelerator technology can help solve nonlinear transient and static analyses faster.

By Ray Browell, ANSYS, Inc.

In this example from Florida Turbine Technologies, parameter modifications involved variations on film coefficients and bulk temperatures.

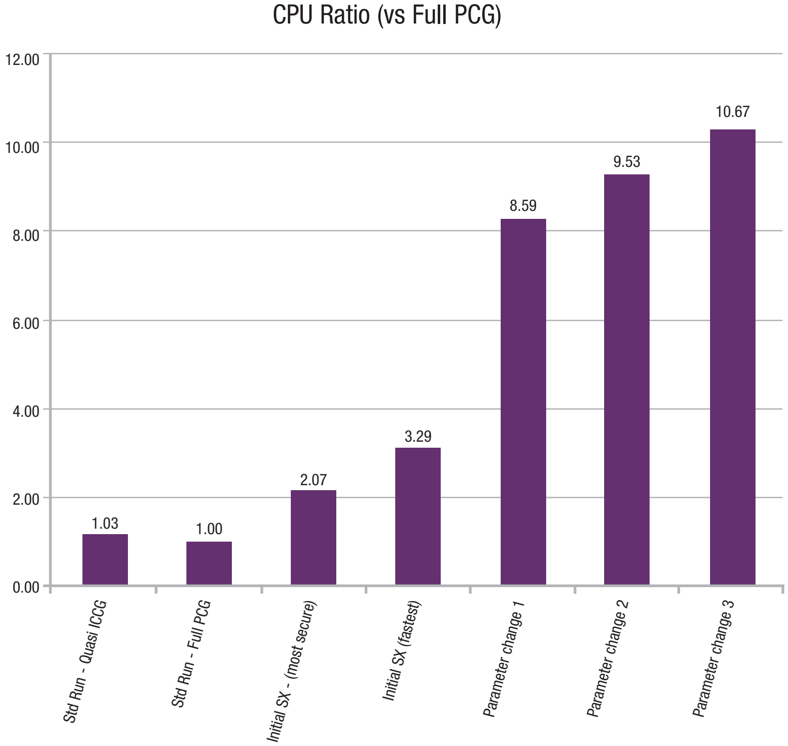
ANSYS, Inc. has offered leading-edge nonlinear solutions for a number of years. Today, by using a unique feature known as ANSYS VT Accelerator, engineers are uniquely positioned to solve their nonlinear analyses faster. Any nonlinear problem that is being solved numerically typically needs some method by which to iterate to a converged solution. One solution is the Newton–Raphson method, which is an iterative process of solving nonlinear equations — the method used by most computer-aided engineering (CAE) software tools including those from ANSYS, Inc. Usually, the solution used as the starting point is the previously converged solution.

Technology from ANSYS includes a predictor method that extrapolates the solution by using the previous history in order to get a better estimate of the solution. The ANSYS VT Accelerator feature greatly expands upon the predictor method by using a very advanced predictor–corrector algorithm. The algorithm is so powerful that sometimes no equilibrium iterations are needed. In other words, the solution is converged at the first equilibrium iteration, thereby greatly increasing the speed of nonlinear solutions. And as a feature unique to ANSYS, ANSYS VT Accelerator technology enables efficient Simulation Driven Product Development.

Solver Speedup

Variational Technology (VT) has been applied to two distinct types of mathematical problems: nonlinear solutions for structural and thermal analyses as well as harmonic analysis. These capabilities are referred to as ANSYS VT Accelerator. This feature provides a two times to five times speedup for the initial solution, depending on the hardware, model and type of analysis. ANSYS VT Accelerator technology makes re-solves three times to ten times faster for parameter changes, allowing for effective simulation-driven parametric studies of nonlinear and transient analyses in a cost-effective manner. Users can make the following types of changes to the model before an ANSYS VT Accelerator re-solve:

- Modify, add or remove loads (constraints may not be changed, although their value may be modified)
- Change materials and material properties
- Change section data and real constants
- Change geometry, although the mesh connectivity must remain the same (that is, the mesh must be morphed)



Turbine model speedup with full PCG solver time as a baseline. Initial ANSYS VT Accelerator solution was more than two times faster, and subsequent re-solves improved, with the third re-solve being 10 times greater.

Nonlinear Solution Speedup

ANSYS VT Accelerator technology for nonlinear solutions speeds up the solution of applicable nonlinear analysis types by reducing the total number of iterations. The technology supplies an advanced predictor–corrector algorithm based on Variational Technology to reduce the overall number of iterations for nonlinear static and transient analyses. It is applicable to analyses that include large deflection, hyperelasticity, viscoelasticity and creep nonlinearities. Rate-independent plasticity and nonlinear contact analyses may not show any initial improvement in convergence rates; however, users may choose this option with these nonlinearities if they wish to resolve the analysis with changes to the input parameters. In general, ANSYS VT Accelerator technology can be used for:

- Nonlinear structural static or transient analyses not involving contact or plasticity
- Nonlinear thermal static or transient analyses

Harmonic Analysis

The harmonic sweep feature of ANSYS VT Accelerator provides a high-performance solution for forced-frequency simulations in high-frequency electromagnetic problems and structural analysis. For a structural harmonic analysis, the material may have frequency-dependent elasticity or damping.

For a high-frequency electromagnetic harmonic analysis, ANSYS VT Accelerator technology computes S-parameters over the entire frequency range. In practice, the harmonic sweep feature completes one normal run in ANSYS Mechanical software at the mid-frequency of the specified frequency range. It then performs accurate approximations of the results across the frequency range (in user-specified steps). In addition to controlling the steps and the frequency range, users can specify the accuracy of the approximations. Two harmonic sweep solution methods are available: Variational Technology and Variational Technology Perfect Absorber. The Variational Technology Perfect Absorber method provides about a 20 percent faster solution, but it is somewhat less accurate. ■

> In order to provide “innovations that work,” Florida Turbine Technologies, Inc. — which executes all aspects of turbine engine design and development in the military and commercial aircraft industry — desires transient fidelity early in the design process. “Due to long run times, we usually reserve transient analyses for detailed final design,” says Joseph T. Metrisin, lead structures engineer at Florida Turbine Technologies, Inc. “Faster solution options will allow us to perform detailed transient analyses early on in the design process, resulting in more robust designs.”

Seeing is Believing

Tired of looking at your CFD images inside out? Developments in version 11.0 software from ANSYS allow inclusion of solid parts during pre- and post-processing, making for more intuitive problem setup and results visualization.

By Judd Kaiser, ANSYS, Inc.

When engineers are first introduced to computational fluid simulation, one of the challenges they face is learning to see the world inside out. In the real world, we're accustomed to focusing our eyes on the parts that fluid flows in and around. But unlike what we usually see in the real world, models created for computational fluid dynamics (CFD) simulations include only the regions in which the fluid flows — not the solid components that surround the fluid. This often makes it difficult to visually communicate the relationship between the CFD results and the related solid geometry under analysis, especially to the untrained eye. With some surprise, the author discovered that a confluence of recent developments that came along with the ANSYS Workbench 11.0 release gives an unexpected solution to this visual dilemma.

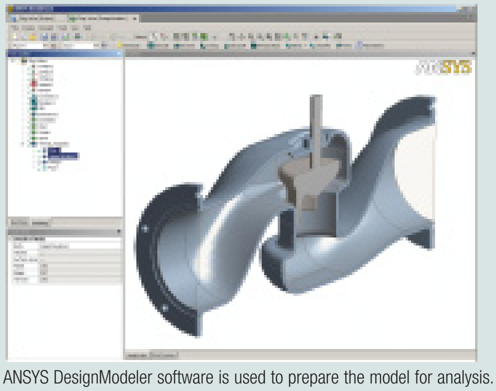
First, the ANSYS Workbench 11.0 platform brings a new meshing application that offers physics-based, part-by-part meshing with multiple methods. For CFD users, the primary methods of interest are swept meshing with inflation and tetrahedral meshing with inflation. The inflation layer allows for finer mesh resolution to better accommodate the near-wall physics details that are critical for accurate prediction of wall-bounded flows. This new application also offers highly automated, fault-tolerant meshing methods that were tuned for use in mechanical finite element analysis (FEA) simulation. These methods are extremely fast and produce meshes that resolve the geometry with a minimum number of elements. So why would they be of interest to the CFD user?

When preparing meshes for CFD simulation, engineers often work with computer-aided design (CAD) models that were created for manufacturing purposes. These models are usually complex assemblies that describe the solid components that bound the fluid region. For CFD analysis, these parts must be turned inside out to create a part that represents the fluid domain. Typically, the CFD engineer discards the CAD model's solid parts, since they are not of direct interest in the fluid simulation.

There is some additional "cost" to meshing the solid parts, carrying those meshes into the pre-processor and loading multiple files in the post-processor — so this may not be something a user will do for everyday design work. However, if you are generating results that can be used for marketing or to share with individuals who are not experienced in looking at CFD simulations, you may find that including the solid parts visually helps to convey your analysis and results more clearly.

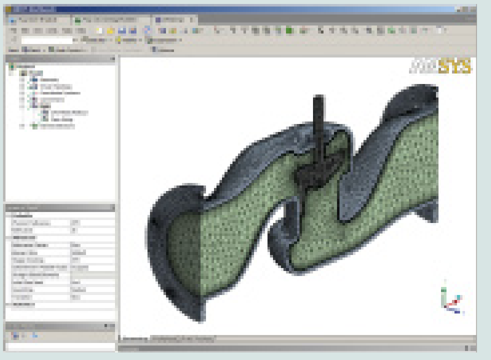
Because these solid parts can be meshed "cheaply," consider bringing them along for the ride. For your next CFD project using ANSYS Workbench 11.0 tools, give the following steps a try.

1) After importing the CAD assembly, use ANSYS DesignModeler software to prepare the model for analysis. For CFD simulation, this often involves the use of fill and/or enclosure operations to prepare the fluid domain. (This step also could be performed with CAD software.)



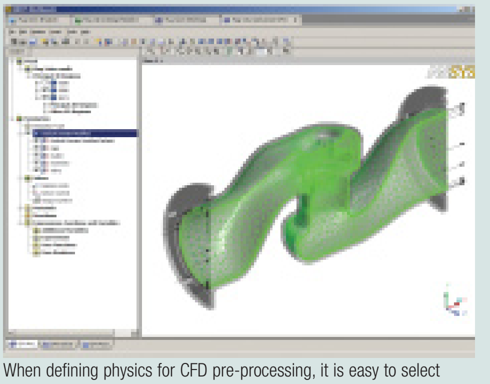
ANSYS DesignModeler software is used to prepare the model for analysis.

2) Bring the fluid domain and all solid parts from the original CAD assembly into the new ANSYS Workbench 11.0 Meshing Application. You can use your meshing method of choice for the CFD domain, but in general it should be a method that includes an inflation (prism) layer to resolve the near-wall physics. Next, instead of discarding the solid parts, mesh them using the automatic meshing method in the Meshing Application, which will result in a mesh that resolves the solid parts with a minimum number of elements. Meshing these parts typically is very rapid, so this comes at low computational cost.



Meshing occurs in the ANSYS Workbench 11.0 Meshing Application.

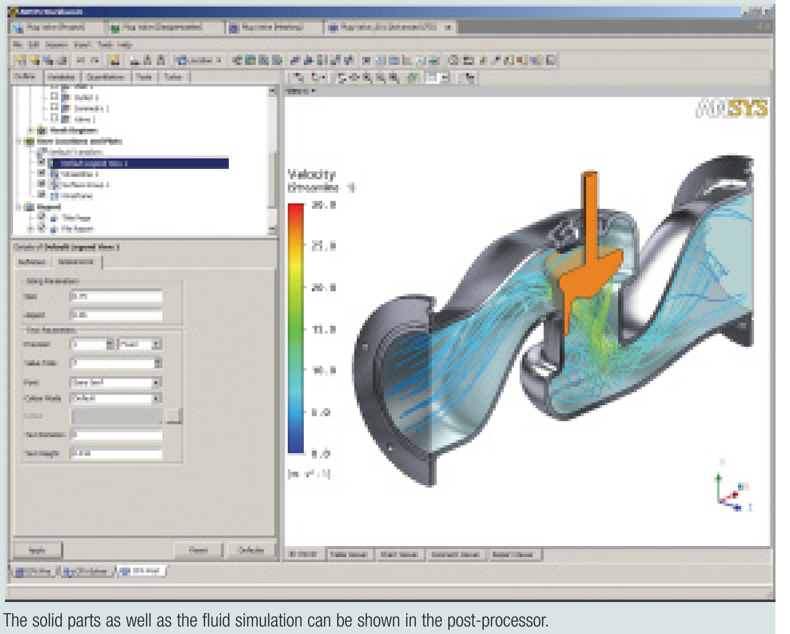
3) Import the mesh assembly into CFX-Pre (the Advanced CFD tab in the ANSYS Workbench platform). In CFX-Pre you may find that having these solid parts available helps you understand the problem visually. When defining physics in CFX-Pre, it is now easy to select only those parts that you want to include in your CFD domain. (In this case, include the fluid part but not the solid parts.) The solid portions of the mesh will be saved in a .cfx file but not in the .def file.



When defining physics for CFD pre-processing, it is easy to select only those parts that you want to include in your CFD domain.

4) Solve the CFD simulation as usual.

5) In ANSYS CFX 11.0 software, CFX-Post allows users to load multiple files — so load both the simulation result (the .res file) and the physics setup (the .cfx file). You then will be able to create graphics to illustrate the nature of the flow field as usual. In addition, you will be able to show the solid parts. Being able to render the solid parts should make it easier for the untrained eye to comprehend the nature of the simulations. ■

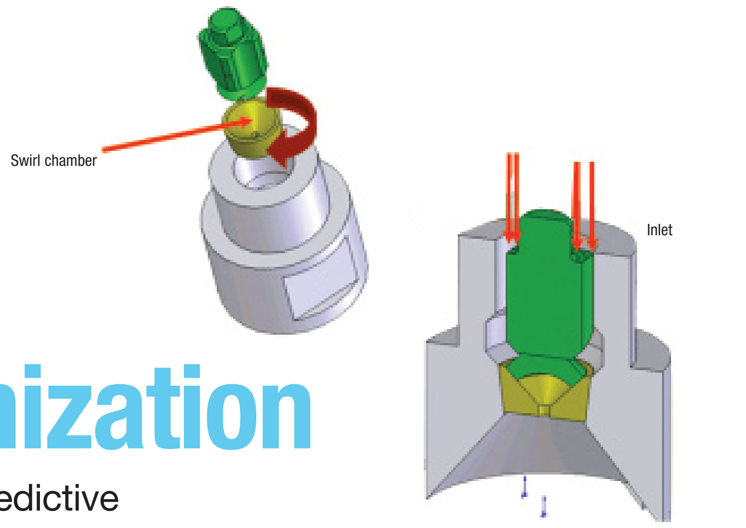


The solid parts as well as the fluid simulation can be shown in the post-processor.

Predicting Liquid Atomization

Simulation can be used as a predictive spray characterization tool.

By Lisa Graham and Kumar Dhanasekharan, Bend Research Inc., Oregon, U.S.A.
John Widmann and Birendra David, ANSYS, Inc.



CAD geometry of typical pressure-swirl atomizer

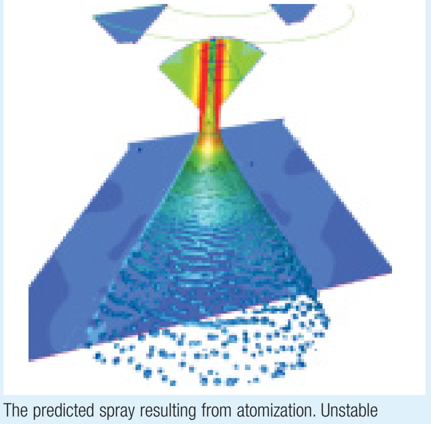
Pressure-swirl atomizers, also known as simplex atomizers, are used commonly in many industries, including aerospace, automotive, pharmaceuticals and others. These nozzles work by forcing a liquid under high pressure into a swirl chamber in which the fluid gains tangential momentum and exits through a small orifice or nozzle. The liquid exiting the nozzle forms a sheet that thins as it disperses radially outward. The thin sheet becomes unstable and breaks up to form ligaments and then discrete droplets. The ability to tailor spray characteristics is important, for example, in controlling the evaporation rate of fuel sprays in gas turbine combustors or the transport of drugs administered through inhalation. A fundamental understanding of spray formation can provide useful insight into the design and operation of the atomizer — in order to produce sprays with desired characteristics such as the droplet size and spray pattern.

Liquid atomization processes such as those associated with pressure-swirl atomizers can be simulated using the volume of fluid (VOF) multiphase model in FLUENT computational fluid dynamics software. This model is preferred when an engineer desires to predict the location of the interface between two immiscible phases or fluids. In the case of the atomization process occurring in a pressure-swirl atomizer, the VOF model predicts the gas–liquid interface location during the formation and disintegration of the liquid film, the formation and tearing of ligaments, and, ultimately, the formation and transport of droplets.

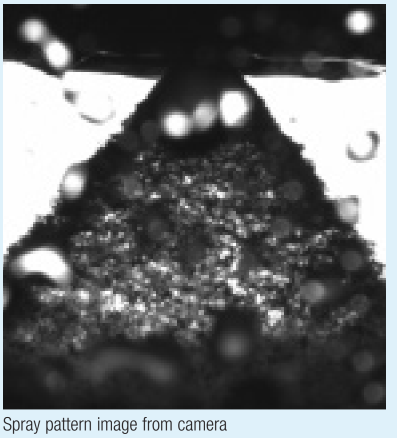
In this study, the research team developed a FLUENT model to predict the liquid atomization and spray formation. Additionally, the spray was characterized experimentally for model validation. A high-resolution camera combined with a laser flash was used to visually capture the spray. The model predicted a cone angle of 60.2 degrees, which compares favorably to the experimental values in the range of 69 to 75 degrees. For the liquid flow rates investigated, the model predicted the atomization pressure within 10 percent of published data (spraying systems water capacity data). Additionally, the model predicted all the salient features of the flow, including the air core that develops within the swirl chamber in response to the swirling liquid flow. As the liquid exits the nozzle orifice, the tangential momentum of the swirling liquid causes the sheet to move radially outward, thin and, ultimately, disintegrate.

Under the conditions considered in the model, the disintegration of the liquid sheet is preceded by the formation of unstable waves, called Kelvin–Helmholtz waves, on the liquid surface. The model predictions demonstrated that the development of Kelvin–Helmholtz waves led to disintegration of the sheet and formation of ligaments. The ligaments experienced further breakup until surface tension forces exceeded aerodynamic forces, resulting in spherical drop formation.

The research team used the validated model to study liquid atomization under varying operating conditions, including solution viscosity, surface tension and flow rate. This enabled engineers to map out a design space for successful operation of the nozzle. Such simulations provide a fundamental understanding of how operating parameters affect spray characteristics and help in tailoring nozzle design and operation to obtain sprays of desired characteristics. ■



The predicted spray resulting from atomization. Unstable waves, called Kelvin–Helmholtz waves, are apparent on the surface of the liquid. Downstream, the sheet disintegrates into ligaments and further into droplets. This image corresponds to a surface of constant liquid volume fraction (a = 0.05) and is colored by the magnitude of the tangential (swirl) component of velocity.



Spray pattern image from camera