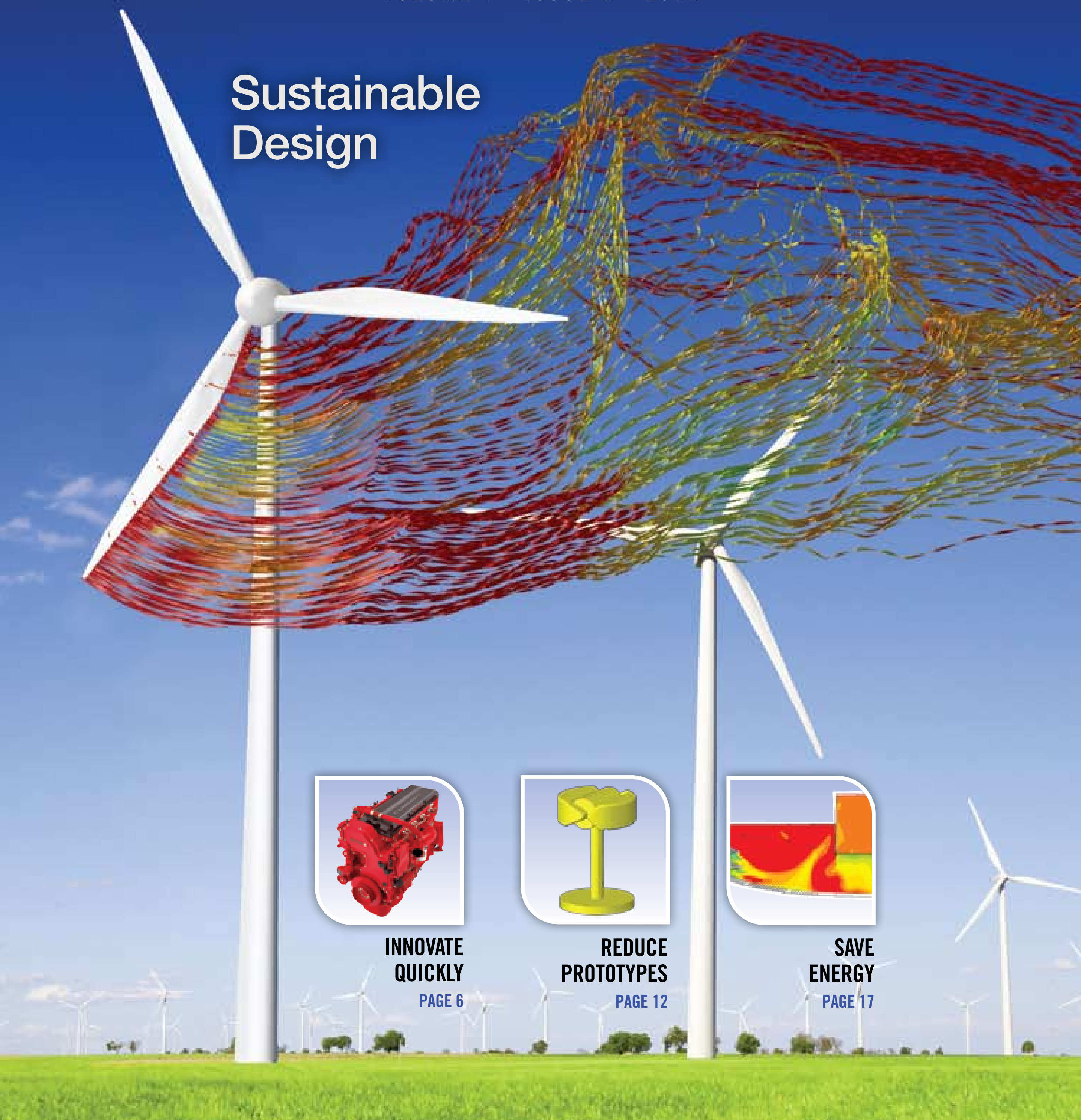
ANSYS

ADVANTAGE™

EXCELLENCE IN ENGINEERING SIMULATION

VOLUME V ISSUE 1 2011



Sustainable Design

INNOVATE QUICKLY

PAGE 6

REDUCE PROTOTYPES

PAGE 12

SAVE ENERGY

PAGE 17

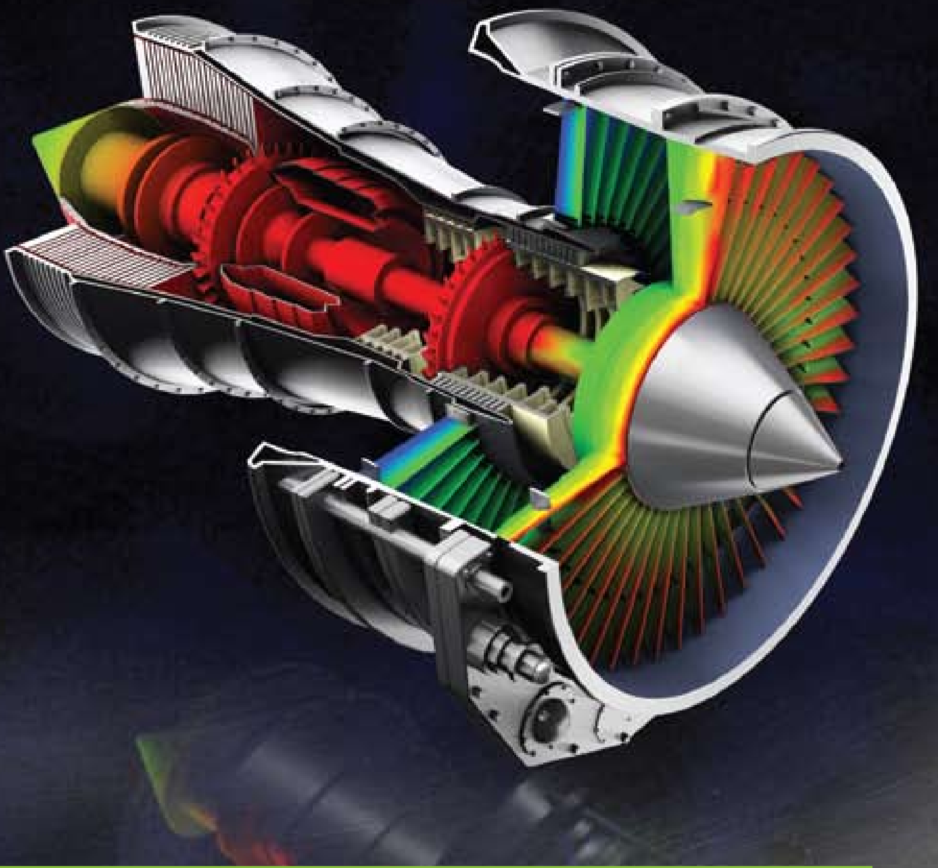
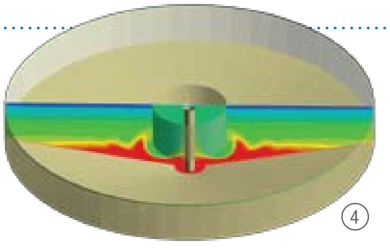


Table of Contents

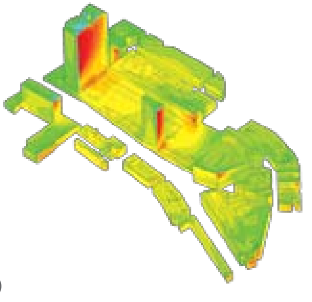
SUSTAINABLE DESIGN



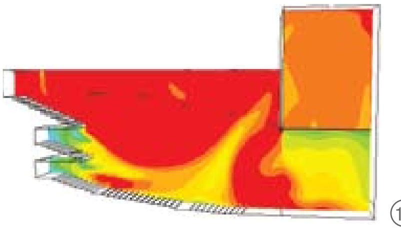
4

6

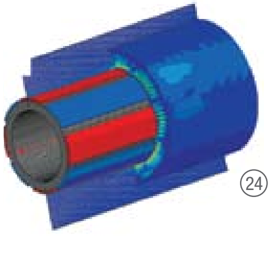
12



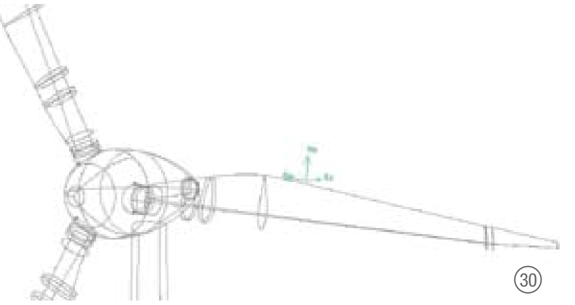
9



17

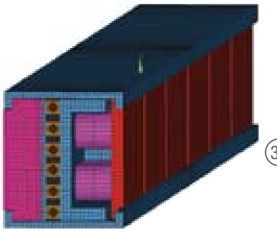


24

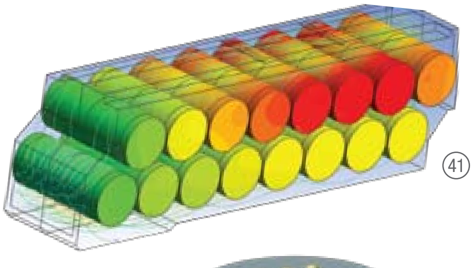

(30)

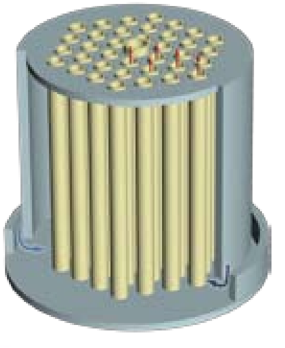
SIMULATION@WORK

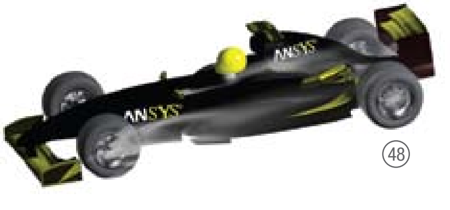

(32)


(38)

DEPARTMENTS

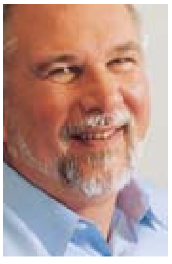

(41)


(46)



(48)

In Memoriam



John Krouse

John Krouse, senior editor and industry analyst for *ANSYS Advantage*, passed away on November 6, 2010. He was editorial director and one of the originators of *ANSYS Solutions*, the corporate publication that preceded *ANSYS Advantage*. His credits included being editor and publisher with Penton Media for 18 years, where he contributed to such publications as *Machine Design* and *Computer Aided Engineering*. Krouse founded the communications consulting firm Krouse Associates in 1994. His understanding of CAD/CAM/CAE was extensive, and he authored many books and articles on the topic.

ANSYS Advantage readers and collaborators alike will miss his ability to explain complex engineering concepts in an entertaining manner, as well as the keen perception he brought to light in his editorials.

For ANSYS, Inc. sales information, call **1.866.267.9724**

Email the editorial staff at **ansys-advantage@ansys.com**.

Executive Editor
Fran Hensler

Managing Editor
Chris Reeves

Art Director
Dan Hart

Editors
Erik Ferguson
Shane Moeykens
Mark Ravenstahl

Ad Sales Manager
Helen Renshaw

Editorial Contributor
ANSYS North America Support and Services

Editorial Advisor
Tom Smithyman

Designer
Miller Creative Group

Circulation Manager
Sharon Everts

About the Cover
Turbulence can greatly affect the placement of wind turbines on a farm. This and other *ANSYS Advantage* features focus on solutions for sustainable design.

Simulation image courtesy Fluid and Energy Engineering GmbH & Co. KG.
Photo © iStockphoto.com/Mienny.

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

Green Design without Compromise

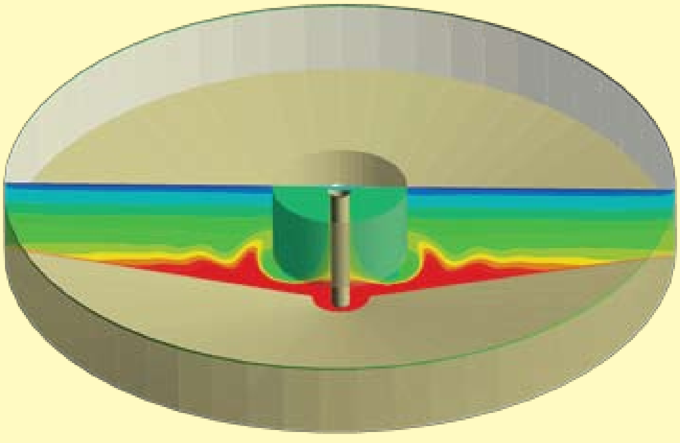
Companies can gain a sustainable competitive advantage and be environmental stewards — through robust design and optimization.

By Thierry Marchal, Industry Director, ANSYS, Inc.

The Need for Sustainability

Throughout much of the world, people expect that life will become better, easier and more comfortable for succeeding generations. Over the past century, privileges once limited to the wealthy have become commonplace; indeed, many are now considered a necessity. Owning a car, flying around the world, accessing a computer, connecting to the internet from anywhere, and using a cell phone are routine activities, even in emerging countries. However, the world can't sustain such exponential growth forever. The planet has received a wake-up call about addressing overconsumption. Many scientists state emphatically that we are putting our planet, and the human species, in danger.

The population is rapidly draining the world's fossil energy sources; at the same time, emissions from these fuels are negatively impacting the environment and its natural evolution. Huge amounts of waste material are being stored with the hope that the earth can cope with it. Most people are aware that the globe cannot sustain these practices forever. While a growing number of people are urging the world's population to do something about it, many individuals are unwilling to sacrifice their comfortable lifestyle.



Engineering for sustainability includes designing more efficient ways to treat waste, such as studying settling in a wastewater treatment tank.
Courtesy MMI Engineeering.

Reluctance to Sacrifice Current Standards

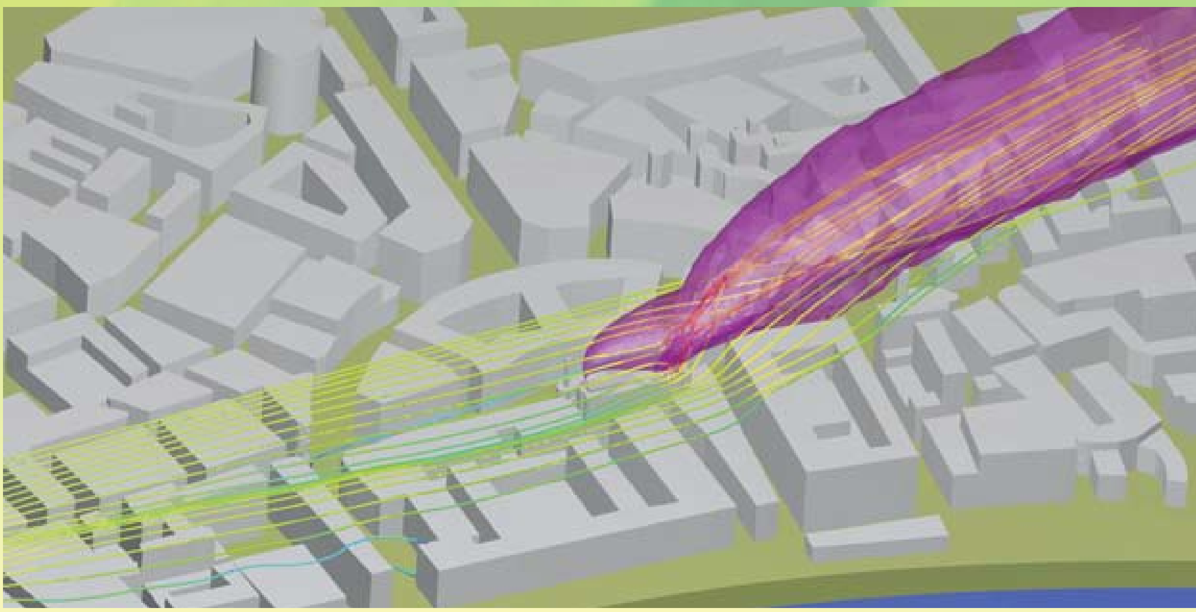
Without a doubt, cars could be more environmentally friendly, but if you surveyed the planet, would you find people willing to pay more to achieve this objective? Most would be reluctant to accept significant reductions in performance, speed or acceleration. Green building is a common aspiration, but will the occupants tolerate being a bit less warm in winter, and a bit more warm in summer? Renewable energy is a "must do" only if the resulting power is less expensive and at least as stable. An admirable goal: Can we reach it without personal compromise?

Common sense dictates that we can't meet such expectations using traditional approaches and technologies. Real breakthroughs may be necessary to achieve sustainability goals in these areas:

- **Better energy management**: Fossil fuels remain an important energy source, but they could be used more effectively and selectively. Sources of renewable energies should be exploited. Transportation, construction and manufacturing must become more energy efficient.
- **Pollution reduction**: Twenty-first century lifestyles result in the release of huge quantities of various materials into the environment, making absorption and recycling of these substances difficult without a direct impact on the planet.
- **Product lifecycle**: After a lifetime of use, products quickly become waste material. Shortened product life increases the amount of waste. Processing recycled materials into new products in a cost-effective manner can benefit greatly from modern simulation technology.

Virtual Experimentation, Virtual Testing, Virtual Prototyping

If sustainable products and processes today are merely "nice to have," it is likely green design will become mandatory in the future, legislated and forced by market pressures to develop better solutions. Pioneers and leading companies are already preparing for this evolution by designing sustainable solutions. But the challenge is



Engineering simulation can help organizations meet regulations and standards. This study was used to assess the impact of fume cupboard discharges on surrounding buildings and the environment.
Courtesy BDP Engineering.

Background photo © iStockphoto.com/ooyoo

daunting. Environmentally friendly products may require major innovations that could impact both cost and robustness. Innovation usually requires intense experimentation and optimization. Real breakthrough solutions need systematic testing to ensure that new designs behave properly throughout the entire lifecycle. Experimentation and testing processes are typically time intensive, costly and difficult to manage in a highly competitive world. Best-in-class companies are addressing this issue by switching to the virtual world.

Numerous reports and industry studies, some found in this issue of *ANSYS Advantage*, suggest that an order of magnitude more experimentation can be done virtually for a smaller cost. Engineering teams can vary numerous parameters to identify the best combination of sustainable design, minimal cost and maximal performance. Leading companies are systematically testing their virtual product prototypes against conditions that would be experienced throughout the lifecycle to ensure that product behavior will satisfy end users. Continually increasing computational power combined with high-efficiency solvers can contribute to the effort: They might enable the number or complexity of virtual tests to double every 18 months without impact on cost or time to market.

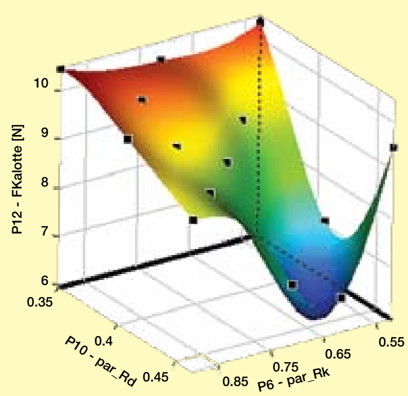
Robust Design and Optimization

Sustainability adds a new dimension that is difficult to handle with standard technology. Designing smart, green products to a high level of product integrity requires investigating a growing number of parameters — such as dimension, materials and operating conditions — to provide designers with enough freedom to meet numerous constraints. Robust design includes identification of influential parameters and evaluation of design sensitivity based on performing variations.

The combination of parametric studies and sensitivity analysis for a large number of parameters using advanced tools (robust design and optimization) opens the door to a new era of design.

Engineering Simulation: A Profitable Green Technology

Engineering simulation has proven its cost-effectiveness in developing innovative products. This same technology can be used to design greener products and processes. Pioneering companies are already creating a cleaner and more profitable future by changing their design processes and widely adopting simulation to transform the green challenge into a major business opportunity. This issue of *ANSYS Advantage* illustrates how companies are using comprehensive integrated multiphysics from ANSYS to maintain corporate responsibility while increasing profit. ■



Designing smart green products to a high level of product integrity requires advanced tools that investigate a growing number of parameters.

Image © iStockphoto.com/PeskyMonkey

Cummins ISX15 heavy-duty engine

Cummins ISF3.8 light commercial vehicle engine

Cleaner, Greener Engine Design

Cummins uses simulation to reduce weight, improve fuel economy and reduce emissions of engines.

By Bob Tickel, Director of Structural and Dynamic Analysis, Cummins Inc., Columbus, U.S.A.

The phrase “environmentally responsible” doesn’t seem to fit into a sentence with the terms 18-wheel tractor trailer and heavy-duty truck. As a world-leading manufacturer of commercial engines and related systems, Cummins Inc. is working to change that perception one design at a time, developing next-generation technologies that are revolutionizing the international trucking industry.

Using software from ANSYS, Cummins is developing and testing radical improvements in engine design, including the use of alternative materials and smaller engine footprints that reduce weight, improve fuel economy and reduce emissions — while also boosting performance. The work of the corporate research and technology organization focuses on developing new, environmentally responsible technologies for the company’s core engine business.

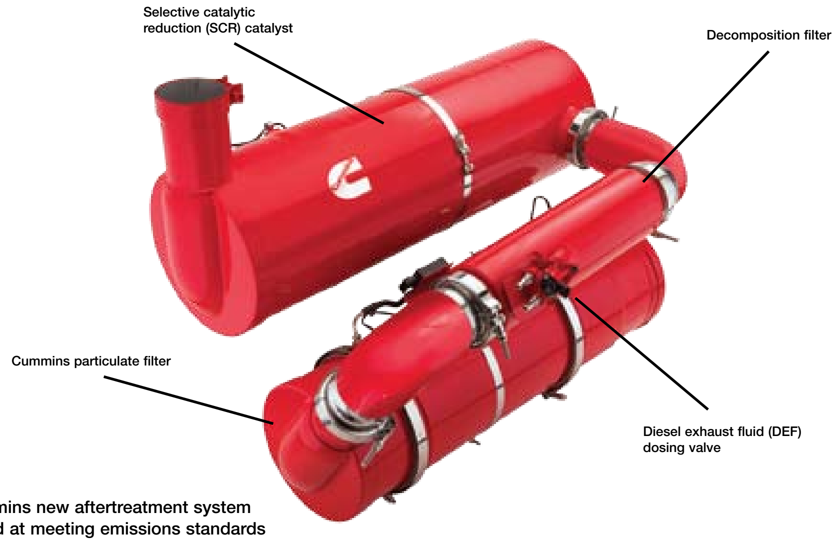
Cummins’ recent product development efforts target a great deal of attention on fuel economy and emissions — and with good reason. Government environmental standards grow more challenging every year. Because commercial trucking is a low-margin business, every improvement that Cummins makes in fuel economy adds to its customers’ successes. Beyond these practical considerations, Cummins invests in environmentally responsible engine technologies because it is the right thing to do. The company wants the trucking industry to be viewed as an environmental steward and champion, not one of the “bad guys.”

Building the Truck of the Future

In recognition of its environmental technology leadership, Cummins recently received nearly $54 million in funding from the U.S. Department of Energy (DOE) to support two projects aimed at improving fuel efficiency in both heavy- and light-duty vehicles.

About $39 million will fund Cummins’ development of a new “supertruck” — a highly efficient and clean diesel-fueled Class 8 (heavy-duty) truck that is expected to set a new industry standard for green technology. Another $15 million in funding will support the development of advanced-technology powertrains for light-duty vehicles. The resulting improvements in engine system efficiency will mean significantly lower fuel and petroleum consumption by these vehicles as well as a significant reduction in greenhouse gas emissions.

Though details of this work are proprietary, Cummins is using software from ANSYS to simulate engine performance and achieve the ambitious goals defined by the DOE: Improve Class 8 vehicle freight efficiency by 50 percent and achieve a 40 percent improvement in fuel economy in light-duty vehicles.



Cummins new aftertreatment system aimed at meeting emissions standards

To illustrate how Cummins leverages ANSYS software on a daily basis, consider this recent product introduction: the next-generation ISX15 engine design, developed specifically to meet stringent new emissions standards from the U.S. Environmental Protection Agency (EPA). These regulations, which call for near-zero emission levels of nitrogen oxide (NOx) and particulate matter (PM), place new demands on truck manufacturers and fleet managers.

Cummins used ANSYS technology to develop or enhance a number of features in the ISX15 design, including the new Cummins aftertreatment system that incorporates a revolutionary diesel particulate filter (DPF) targeted at meeting the new emissions standards.

The DPF removes diesel particulate matter or soot from the exhaust gas of a diesel engine. Cummins used ANSYS software to simulate typical operating loads and make predictions about the new DPF's performance and reliability. Engineering simulation predicted both peak temperatures and temperature distribution inside this component under a range of operating conditions. These thermal analyses were critical as they revealed the peak temperatures and temperature gradients within the filter, which ultimately determine the thermal fatigue and life of the component.

Varying temperatures within the DPF result in thermal stresses; if this component is not designed properly, temperature variations can lead to component failure. While it might take months or years for failure to occur in real-world use, software from ANSYS enabled Cummins to quickly simulate the effects of years of field usage and make predictions about how the DPF would hold up over time. This gave the engineering team a high degree of confidence as they designed the component and installed it into customer vehicles.

Subsequent field and bench tests confirmed the simulation predictions, as did actual road results. Engineers can rarely rely on virtual testing alone, but software from ANSYS confirmed that the Cummins design process was moving in the right direction — and that the results would be as team members expected.

Using Simulation to Rev Up Ongoing Engine Improvements

Environmental standards and customer needs are an ever-evolving target. So the Cummins team uses engineering simulation to improve engine features and boost performance. Design changes that reduce emissions and fuel consumption often result in higher temperatures and pressures within the engine, which require Cummins engineers to continually test the limits of conventional engine designs.

Most materials used in diesel engines exhibit reduced strength as temperature rises. The combination of high pressure/temperature and reduced strength places stress on components such as the cylinder head, a geometrically complex casting that serves many functions including transferring engine oil and coolant, intaking air and exhausting gas. The head also houses the fuel injector and valve train components; it must contain hot combustion gases during the cylinder firing event. To address the multiple forces that come into play, Cummins engineers use multiphysics software from ANSYS to combine thermal and structural analyses in their work on this sophisticated engine component.

Simulation tools from ANSYS enable the Cummins engineering team to evaluate the use of new materials across the entire topological surface of cylinder heads and other engine components. ANSYS allows the Cummins team to quickly and easily answer questions

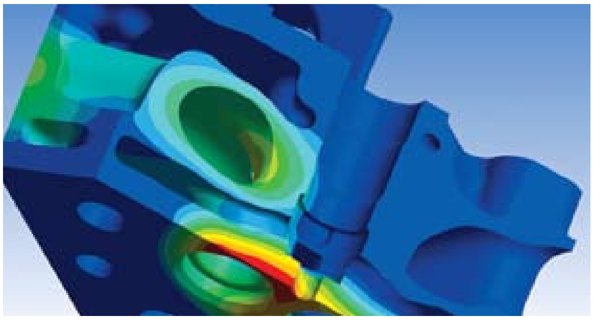
By using software from ANSYS to predict maximum temperatures and temperature distribution within its new diesel particulate filter under diverse conditions, Cummins engineers can ensure that the manufactured component will have acceptable durability.

such as, "What happens if we manufacture the cylinder head out of this new material? What are the performance gains?" without the need to actually machine and test new components. Working in a simulated environment gives these engineers the freedom to tweak existing engine designs — as well as to arrive at some "clean-sheet" designs that may have the power to revolutionize engine performance.

The team recently used software from ANSYS to predict temperature profiles in cylinder heads under various operating conditions. The thermal results were used to determine how this component would perform in the real world, as well as its threshold for both low-cycle and high-cycle fatigue.

Since cylinder heads are long-lead-time, expensive components, the Cummins team must be sure that a new design is right before moving forward. Using simulation tools from ANSYS, Cummins engineers can not only create more exacting test results but also pursue increased productivity by incorporating capabilities such as conjugate heat transfer modeling in their designs.

Without analysis-led design to introduce new materials and new part configurations, Cummins would have to rely instead on very expensive, time-consuming endurance tests to verify the engineering team's designs. ANSYS is a key enabler to Cummins, reducing the company's overall cost of development by minimizing its investment in physical engine testing. This approach allows the design of a more environmentally friendly engine without compromising cost or performance. ■



Cummins engineers can easily test the effectiveness of new materials, new designs and other innovations — and predict their long-term effect on overall engine performance. For example, Cummins engineers used software from ANSYS to simulate the impact of new materials on a cylinder head design.

Shifting Gears: ANSYS Creates a Cultural Change

With so many engine components and performance aspects to consider, the Cummins engineering team must perform a wide range of structural simulation and analysis. According to Bob Tickel, director of structural and dynamic analysis at Cummins, software from ANSYS is the only single-vendor tool with the technical breadth and depth to meet this challenge.

Recently, the Cummins corporate research and technology team began the switch from the traditional interface for ANSYS software to the ANSYS Workbench platform, a decision based primarily on the product's improved geometry import, cleanup and meshing capabilities. Tickel anticipates a 50 percent reduction in throughput time based upon the move to ANSYS Workbench.

"The ANSYS Workbench environment provides access to the best multiphysics tools we need to conduct many types of simulation and analysis," said Tickel. "Whether our need is thermal, structural, dynamic or static engineering analysis, ANSYS Workbench provides the flexibility and versatility to accommodate our needs — as well as the multiphysics capabilities to link the results of our various simulations."

Tickel noted that the efficiency and cost effectiveness of engineering simulation has resulted in a complete cultural change at Cummins. "The ease of using simulation tools from ANSYS has helped to transform our organization from a test-centric culture to an analysis-centric one," said Tickel.

"When investigating a new material or other design enhancement, traditionally we would build new parts and conduct physical tests as a first step — which represented a time- and cost-intensive approach," he said. "Today, we focus more attention on upfront analysis, only moving to part-building and testing for those design improvements that we can verify first using ANSYS tools. This new cultural approach has not only saved us time and money, but allowed us to selectively focus our attention on those design enhancements that are shown to hold the greatest promise for revolutionizing future engine designs."

The State Administrative City in Brussels was completed in 1983 and is now largely unused.
Photo courtesy Michael Uyttersprot.

Brussels'tainable

Simulation helps to determine potential environmental impact in preparation for a massive renovation in Brussels.

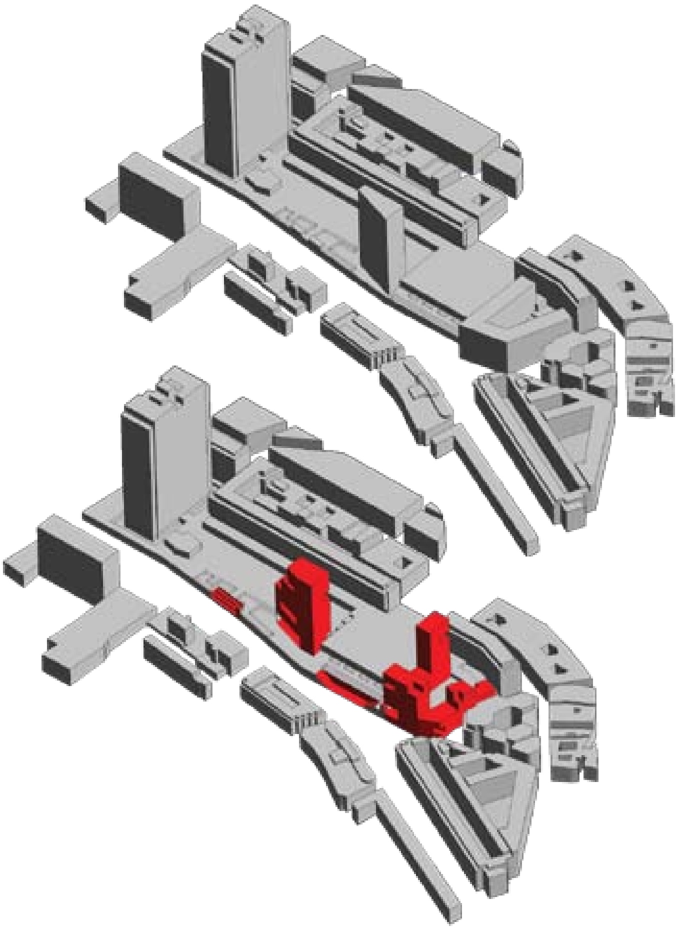
By Tin Meylemans, Architectural Engineer, and Jean-Pierre Demeure, Head of Urban Planning, City of Brussels, Belgium

Arnaud Boland and Quentin Hamoir, Support Engineers, ANSYS, Inc.

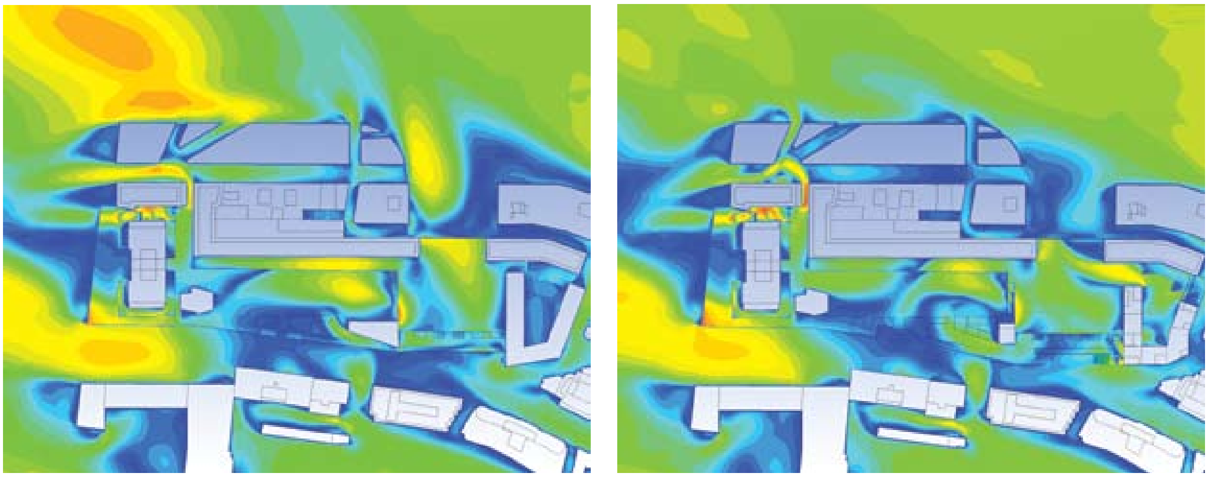
In 1992 during the Earth Summit in Rio de Janeiro, 200 countries — including Belgium — adopted Agenda 21, an action program for the 21st century to enable globally sustainable development on the planet. Among the program's main objectives is responsible management of natural resources and harmonious urban development. As the federal capital of Belgium and an administrative center of the European Union, the City of Brussels has been a leader by example. It is also a prime contributor to generating awareness of how large cities can naturally evolve to environmentally friendly urban growth.

Guided by Agenda 21 and targeting urban revitalization, the city undertook renovation of the State Administrative City, or CAE (Cité Administrative de l'Etat), an important site in downtown Brussels. Often recognized as the finest architectural achievement in Brussels since World War II, CAE is nevertheless an urban planning defeat, as it eliminated an entire district and broke the urban fabric with its 140,000 square meters of office space. As the Belgian federal government evolved, the administrative city has not been used since the early 2000s, leaving this huge site unoccupied. Its renovation had become a city priority. Today the site is undergoing a massive renovation project that combines housing, offices, shops, parks and a school.

Before any major project is constructed, Belgian law requires an environmental impact study that includes economic and social factors, environmental assessments, mobility studies, health influences and noise level effects.



Initial (top) and modified (bottom) geometries of Brussels' CAE



Initial design

Recommended revision

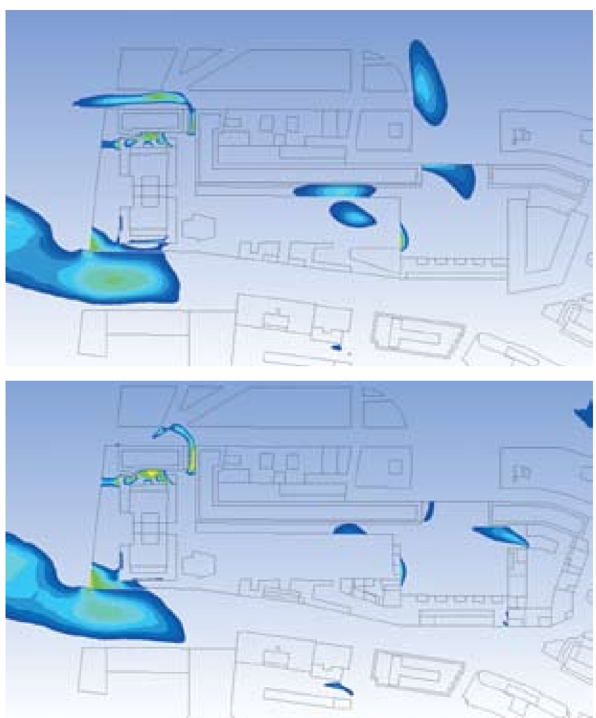
Engineering simulation was performed to determine pedestrian comfort levels (with yellow and red indicating discomfort).

The City of Brussels selected the ANSYS office in Belgium to investigate wind flow patterns for the new site and compare different options for building locations — via engineering simulation.

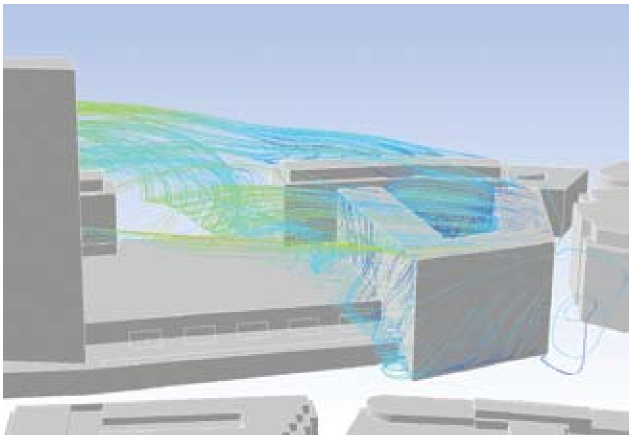
Wind analysis is useful in studying many parameters related to a site's user comfort, in this case including high-rise buildings and green spaces. In some instances, the building acts as a screen and improves comfort, but in others the wind pattern could cause significant discomfort — and might even be dangerous — to pedestrians and occupants. Several important phenomena can occur, such as channeling and wedge effects, that lead to local wind acceleration and increased turbulence. These phenomena can contribute to discomfort for people located at ground level, terraces and balconies. The wind effects can also damage vegetation and generate unwanted pressure effects on buildings that cause whistling or material damage.

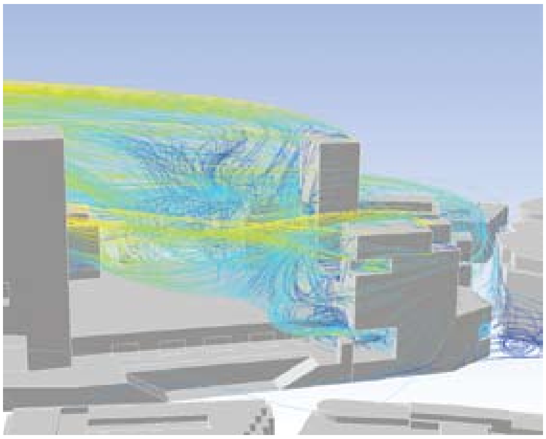
A typical comfort criteria from the TNO (an independent research organization in the Netherlands whose aim is to apply scientific knowledge to strengthen the innovative power of industry and government) requires that the wind speed in an area 1.75 meters above the ground should not exceed 5 meters/second for more than 220 hours per year. In identifying areas of discomfort, the standard must be expressed in terms of amplification factor (AF), the ratio between velocity without and with the building. A factor greater than 1 means the flow is accelerated by the buildings. Considering Brussels' meteorological data over the last 10 years, an AF of less than 0.88 satisfies the most-restrictive comfort criterion.

Project engineers investigated two configurations for environmental impact along with a possible need to minimize the negative impacts of wind. The first configuration, called initial state, contained the layout of the project (the volume envelope prescribed by the City of Brussels in which the building must be situated to satisfy local planning regulations). A second configuration, called the modified geometry, was studied to quantify the impact of architectural changes, such as towers being built on stilts, or architectural details, such as carvings or openings placed on different levels/stories of the structure. Because plans for reuse of the area were not complete at the time of analysis, this study would show where changes could have an impact.



Areas of potential discomfort in original (top) and modified (bottom) designs



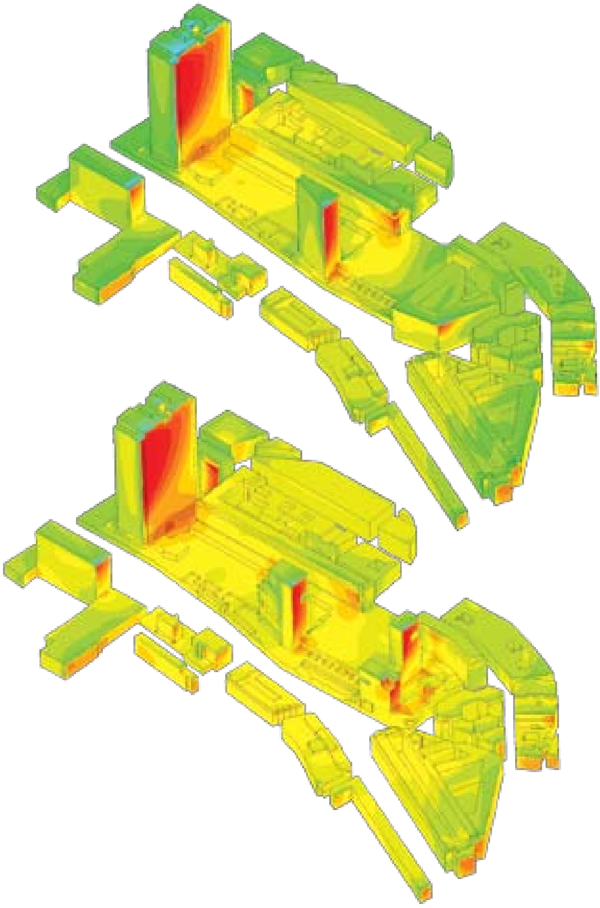


Pathlines showing wind trajectory colored by velocity magnitude for original (left) and new (right) designs provided increased understanding of wind behavior near buildings to determine areas of concern.

The ANSYS team determined the amplification factor at human height for both configurations, illustrating wind effects such as corner effects near tall buildings and channeling effects between closely spaced buildings. For the initial state, a map of the most-restrictive values (in which AF is greater than critical value) revealed areas where local wind speed was too high and would lead to pedestrian discomfort. The modified configuration reduced areas of discomfort by approximately a factor of three.

The team used ANSYS engineering simulation to determine airflow between the buildings, and these results were used to generate velocity vectors to identify specific points of interest, such as recirculation zones in which dust could accumulate. The details provided information that helped in adjusting the building design to avoid these types of zones. Pathlines of wind trajectory provided increased understanding of wind behavior near buildings, including turbulence, and identified areas where potential problems could occur. Finally, computing pressure contours on the buildings identified areas that would require special care during the design phase to avoid other concerns, including whistling problems due to insufficient insulation or damage to the facade.

This project demonstrates the use of engineering simulation for smart urbanization. Such analysis can provide comprehensive maps of airflow patterns, local wind velocities and local turbulence intensities — all useful in planning urban development projects. Simulation-driven building design provides information about how structures affect the environment, which can be an important part of discussions and decisions from a project's first stages. ■



Pressure contours for original and new designs identified areas that required special care during design to avoid insulation whistling problems or potential façade damage.

Catch the Next Wave

Hydrodynamic simulation helps to deliver two- to three-times wave power efficiency improvement.

By Bradford S. Lamb, President, and Ken Rhinefrank, Vice President of Research and Development, Columbia Power Technologies, LLC, Corvalis, U.S.A.

If all the ocean's energy could be harnessed, it would produce more than 500 times the global energy consumption. The practical potential for wave energy worldwide is projected to be between 2 trillion and 4 trillion kilowatt hours per year. The World Energy Council estimates that about 10 percent of worldwide energy demand could realistically be met by harvesting ocean energy.

But wave power is a much less mature technology than solar or wind power or, especially, fossil fuel. A tremendous amount of work lies ahead in optimizing the design of wave power systems. Researchers must improve efficiency and reduce costs to the point that these systems can make a major contribution to meeting global energy requirements.

Columbia Power Technologies (COLUMBIA POWER), LLC, is attempting to harness this potential by developing commercially viable and scalable wave power generation systems. In conjunction with Oregon State University, the company is working to develop and commercialize innovative wave energy harvesting devices.

There are several key advantages of wave power:

- **Power density**: Wave power is much denser than other renewable energy systems, enabling wave parks to produce large amounts of power from a relatively small footprint.
- **Predictability**: The supply of energy from wave power can be accurately forecast several days in advance, enabling utilities to make precise sourcing plans.
- **Constancy**: Unlike solar power, which produces energy only when the sun is shining, ocean swells are available 24 hours per day.
- **Proximity to load centers**: Wave energy will not require substantial buildout of transmission capacity, since 37 percent of the world's population live within 60 miles of a shoreline, and 70 percent reside within 200 miles.

Preparing to test the wave power device

COLUMBIA POWER's wave power system: The wings and vertical spar react to the shape of the passing ocean swell. Each wing is coupled by a drive shaft to turn its own rotary generator.

The wave power industry, however, faces a major challenge since product developers have much less experience in the design of wave power devices relative to other renewable energy systems. Wave power companies need to rapidly advance efficiency and reduce costs of their designs to demonstrate viability to potential investors and customers. Other industries have taken decades or longer to develop technology to the point of commercial viability. But the wave power industry does not have that kind of time. To achieve its goals, it needs to rapidly improve designs while conserving limited capital.

COLUMBIA POWER is focusing on development of direct-drive systems, which avoid the use of pneumatic and hydraulic conversion steps and their associated losses. The company believes that direct-drive systems are the future of wave power because they are more efficient and reliable as well as easier to maintain. The number-one design challenge was to optimize the design of the buoy to maximize the proportion of wave power transferred to the buoy. Relative capture width is a dimensionless measure of the efficiency of the device in capturing the available energy of the wave. A relative capture width of 1 means that the buoy has captured 100 percent of available wave energy.

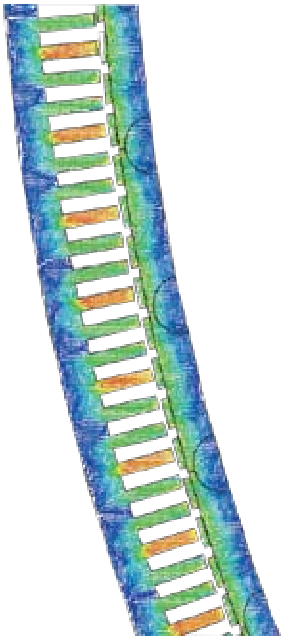
As COLUMBIA POWER set out to determine the optimal shape for the buoy, engineers looked at five different hydrodynamic simulation software packages. The company selected ANSYS AQWA software because of its ease of use, and tests showed that it provided a better match with physical experiments than did competitive software. COLUMBIA POWER also valued that ANSYS AQWA offers both frequency and time domain solutions. Frequency domain solutions are faster, which makes them ideal for quickly evaluating a large number of shapes, while time domain solutions provide the high level of accuracy needed to refine to the best shapes in the later stages of the design process.

COLUMBIA POWER engineers developed an initial concept design in SolidWorks®, built a prototype and tested it at 1/33 scale in the Tsunami Wave Basin at the Hinsdale Wave Laboratory at Oregon State University. The team used high-resolution cameras to track light-emitting diodes on the buoy, measuring its motion in the waves. Engineers exported the concept design to ANSYS AQWA software and performed a time domain simulation while using a wave climate with the same amplitude and frequency as that measured in the wave tank. There was a very good match between the measurements and predictions from ANSYS AQWA. Since then, engineers have used ANSYS AQWA as their primary design tool to optimize the shape of the fiber-reinforced plastic (FRP) buoy.

COLUMBIA POWER engineers doubled efficiency of the buoy by using ANSYS AQWA to optimize its geometry.

COLUMBIA POWER has since evaluated over 350 different geometries with ANSYS AQWA in an effort to maximize the relative capture width of the buoy. At the same time, the company worked closely with Ershigs Inc., its structural partner that produces the FRP floats, to explore the manufacturability of various shapes and to ensure that the final design can be produced at a low cost. The company also looked at the survivability and environmental impact of proposed buoy designs. COLUMBIA POWER engineers used a sinusoidal wave shape and a suite of wave frequencies ranging from 2 seconds to 20 seconds for frequency domain simulations. The response amplitude operators calculated by ANSYS AQWA software were used in a post-processing routine written by COLUMBIA POWER engineers that calculates the relative torque and speed of the buoy as well as the relative capture width.

Once they felt that they were close to an optimal shape for the buoy, COLUMBIA POWER engineers moved to time domain modeling, which makes it possible to evaluate the nonlinear effects of the waves. The team evaluated the shapes that had proven best in frequency domain modeling against a variety of wave climates, including those found at seven different coastal locations around the world. At the same time, engineers began optimizing the power takeoff system that converts mechanical energy into electrical energy. ANSYS AQWA model results from frequency domain models were post-processed in Matlab® Simulink® to incorporate the power takeoff reaction torque and to compute power output. The ANSYS AQWA time domain models were coupled to a DLL that simulated both linear and nonlinear power takeoff operation. The DLL for the power takeoff model was developed in Matlab Real Time®. Engineers used the output from ANSYS AQWA to drive a numerical model developed in Simulink that simulates the power takeoff system and control strategy. The control strategy tunes the power takeoff to the wave climate by changing the amount of current produced by the generator, which, in turn, changes the mechanical load placed on the system. This makes it possible to consider in a single model the effects of different buoy shapes, power takeoff system designs and control strategies; it also helps to determine the power that would be generated by each approach in a variety of different wave climates.



Maxwell software from ANSYS was used to optimize the generator design.

COLUMBIA POWER recently began using Maxwell electromagnetic simulation software from ANSYS to optimize the design of the generator. Engineers evaluated three different electromagnetic simulation software packages and concluded that Maxwell was the easiest to use and the most stable. Maxwell is being used to analyze the electromagnetic performance of the generator while varying the air gaps between the rotor and stator, different magnet geometries, different magnet types, and different types of steel. The overall goal is to maximize the generator's energy output while minimizing its cost.

As a technology startup with far-from-unlimited funding, COLUMBIA POWER must be capital efficient. By focusing its development efforts on simulation and using physical testing judiciously as a verification tool, COLUMBIA POWER is moving forward in the development process much faster than would be possible using traditional development methods. ANSYS AQWA and Maxwell simulation software enable the company to make its mistakes in the computer, where they are far less expensive than in the ocean. ANSYS AQWA technology, in particular, helped to more than double the efficiency of COLUMBIA POWER's wave power system. COLUMBIA POWER has benefitted from the excellent technical support and productive training sessions provided by ANSYS. As a result, the company is on track to soon deploy the first ocean demonstration of its technology in Puget Sound. ■

Reacting to Emissions

Fluids simulation helps to speed up research into chemical-looping combustion capable of reducing fossil fuel emissions.

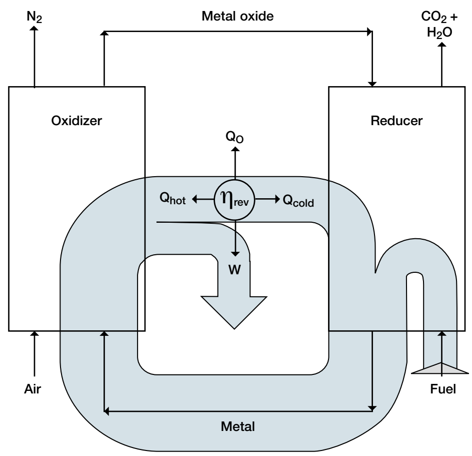
By Schalk Cloete and Shahriar Amini, Flow Technology Research Group, SINTEF Materials and Chemistry, Trondheim, Norway

The need to reduce CO_2 emissions to restrain climate change has never been more urgent. Anthropogenic CO_2 is mainly generated in the combustion of fossil fuels, and these fuels are expected to provide a very large percentage of overall world energy consumption for the next several decades. One thing is clear: Emissions must be lowered. An accepted solution is to separate and sequester the CO_2 emitted by fuel combustion. Chemical-looping combustion (CLC) is one of the most promising technologies to carry out CO_2 capture at a low cost.

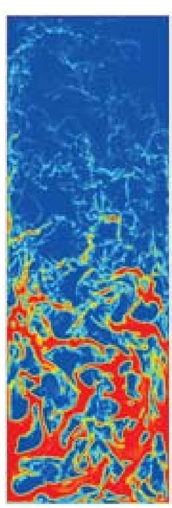
CLC is a combustion process that avoids direct mixing of fuel and combustion air. The method uses two fluidized bed reactors and circulating metal oxide that is oxidized in an air reactor and reduced in a fuel reactor to provide the oxygen required for the fuel. Pure CO_2 is obtained in the fuel reactor exhaust stream after condensation of water without the need for further gas separation. CLC will achieve significant CO_2 capture at a reduced cost when compared to other current technologies, including post-combustion amine scrubbing.

The delay in timely commercialization of CLC technology is primarily due to lack of understanding of reactive multiphase flows in the fluidized beds used in CLC systems. Design and scaleup of CLC reactors are very complex. Advanced modeling techniques are required to capture the intricate coupling between the complex reactor hydrodynamics, heterogeneous reaction kinetics and heat transfer.

With the advent of increased computational capabilities, the fundamental modeling framework of computational fluid dynamics (CFD) is emerging as a very promising additional tool for modeling reactive multiphase transport. Once the model has been validated, CFD can be used for accurate design and scaleup of the process. A team at the Flow Technology group at SINTEF, the largest independent research organization in Scandinavia, has been using ANSYS FLUENT software for this purpose in a project led by Shahriar Amini. The well-established granular flow modeling framework available within the software has been central to the progress made in this project. Fundamental model development also was made much easier by full access to all-important models by employing user-defined functions (UDF) compiled within the stable, parallelized ANSYS FLUENT solver. Heterogeneous reactions, alternate drag laws and wall functions for dilute granular flows were implemented through UDF along with several other variables, such as adjusting pressure drop over a periodic section to maintain a constant superficial gas velocity. Studies were conducted on coarse graining, implementing filtered drag, solids viscosity and solids pressure formulations by means of UDF.



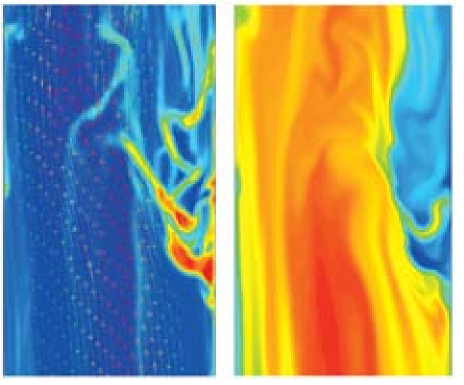
Chemical-looping combustion process



Simulation shows both the bubble (bottom half) and cluster (top half) structures occurring in fluidized beds.

One requirement for modeling the reactive gas–solids flows in CLC processes is that the software predicts formation of the particle substructures (clusters) that often occur in these systems. These structures appear as gas bubbles in dense regions and particle clusters in dilute regions of the fluidized bed reactor, and they influence everything happening in the reactor. Fine meshes and small time steps are required to resolve particle structures, but a price in computational time has to be paid to capture the physics of the system.

These particle structures increase the slip between particles and fluidizing gas. This effect is analogous to the clustering of tiny mist droplets into larger raindrops. A mist of microscopic droplets does not fall and is very easily blown away by the wind, whereas a big raindrop certainly falls with considerable force. The same happens in a fluidized bed reactor: The tiny particles conglomerate into larger clusters and, therefore, fall much more readily. If this clustering is not modeled correctly, the simulation will predict a bed behaving like a mist, but it should actually behave like a thundershower. Clearly, this will lead to substantial errors.

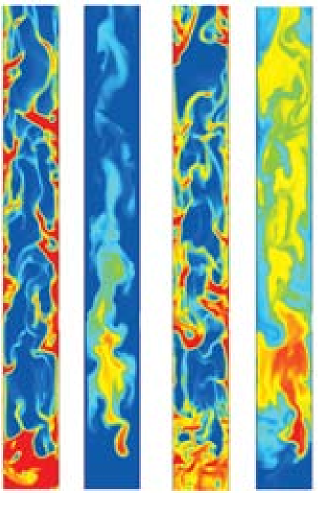


A particle cluster (left) influences the concentration of the reacting gas (right).

In addition, it is believed that particle structures influence reactions. High reaction rates are observed in regions with high volume fractions of particles and reacting gas concentrations. A dense particle cluster represents a region in which lots of surface area is available for reaction, and this significantly increases the reaction rate. However, the consequence is that the reacting gas inside the cluster is used up much faster, thereby slowing down the reaction. Clustering is actually an unwanted phenomenon in fluidized bed reactors because it slows down the reaction rate overall by concentrating all the solids in areas with a low amount of reacting gas. Incorrect cluster modeling, therefore, will predict better, but unfortunately very wrong, reactor performance.

The team at SINTEF has accepted the challenge to contribute fundamental knowledge in the field of simulation of structure resolution on reactor performance. Results have shown that the degree to which the clusters have to be resolved (and therefore the computational cost of the simulation) depends largely on the reactivity of the particle used in the reactor. When a highly reactive particle is employed, the reaction between the fuel gas and a dense cluster is almost instantaneous and occurs almost exclusively on the surface of the cluster. In this case, the most important phenomena to be correctly modeled are the area of the cluster on which the reaction can occur and the transport of the reacting gas species to this interface. Any slight error in the position or sharpness of the cluster surface will lead to significant modeling errors.

When a less reactive particle is used in CFD simulations, however, the resolution of the cluster interface becomes less important. In this case, the reaction rate is slow enough to allow reacting gas to penetrate into the particle clusters and react throughout the reactor domain. Since the reaction occurring on the cluster surface contributes only a small amount to the global reaction rate, the correct description of this interface becomes less important.



The interaction between particle clusters and reacting gas using a particle with high reactivity (two images on the left) and low reactivity (two images on the right). In each pair of images, the one on the left shows the particle volume fraction, and the one on the right shows the reacting gas concentration.

Proper resolution of particle structures along with achieving hydrodynamic and reaction kinetic grid independence require engineers to pay particular attention to grid size compared to particle diameter. To achieve hydrodynamic grid independence for denser beds, the grid size can be 25 particle diameters or even higher, while the fine structures formed in risers need grid sizes of a maximum of 15 particle diameters, provided high-order spatial and temporal discretization is used. Reaction kinetic grid independence works the opposite way. It is very sensitive in dense beds since the volume fraction gradient on a bubble is so large. For very high reaction rates, the grid size can be 10 particle diameters or lower, whereas for 50-times lower reaction rate the grid size can be 25 particle diameters. Reaction kinetic grid independence in risers is slightly more forgiving, since the clusters can be more dilute and normally coincide with hydrodynamic grid independence.

Interestingly, simulation results show that the use of a highly reactive particle will not improve reactor performance as much as might be expected. In the highly reactive case, no reacting gas species is allowed to penetrate into the cluster. A large percentage of the particles available for reaction, therefore, is being wasted inside the dense clusters where no reacting gas species is present. When the particle with a low reactivity is used, fuel gas is available throughout the reactor, and all particles are involved in the reaction. To quantify this concept, simulations have been used to show that a 50-times decrease in particle reactivity decreases the overall reaction rate by a factor of two to three, depending on the fluidization velocity.

Information gained from this study is currently being used at SINTEF to perform fluid simulation as a design and optimization tool for CLC systems. Fundamental insights offered by these models make substantial contributions toward identifying and optimizing important design parameters to accelerate the development of this important CO_2 capture technology. ■

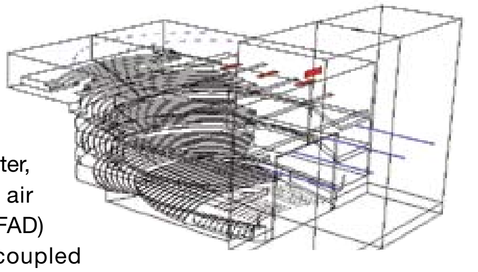
Meeting Green Building Design Goals with Engineering Simulation

Simulation is driving innovation in HVAC design for an assembly hall in Saudi Arabia.

By Sharbel Haber, Senior Mechanical Engineer, Balsam Nehme, Mechanical Engineering and Adnan Akhdar, Mechanical Engineer, Dar Al-Handasah, Beirut, Lebanon

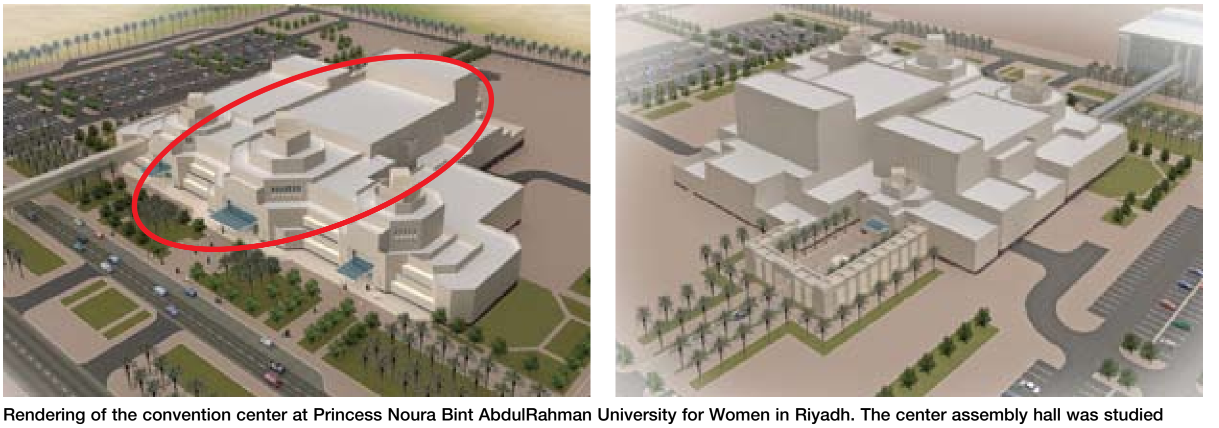
With the ever-increasing demands for sustainable buildings, engineers are developing more complex and diversified designs to reduce loads, boost efficiency and utilize renewable resources. Fluid dynamics simulations have proven to be a powerful and effective tool, providing flexible solutions in increasingly complex and demanding projects. At engineering firm Dar Al-Handasah, these simulations are extensively used as an optimization and validation tool at an early phase in the design process, since simulation supports implementation of innovative designs and energy-saving measures geared toward decreasing the overall facility's energy costs while maintaining or improving occupant comfort.

Dar Al-Handasah (Shair and Partners) has been a pioneering force in the planning, design and implementation of development projects in the Middle East, Africa and Asia since it was founded in 1956. Today, Dar Al-Handasah is one of the largest engineering and design firms in the world. A typical example of building for energy efficiency is the company's recent design for the 31,000-square-meter convention center at Princess Noura Bint AbdulRahman University for Women in Riyadh, Saudi Arabia.



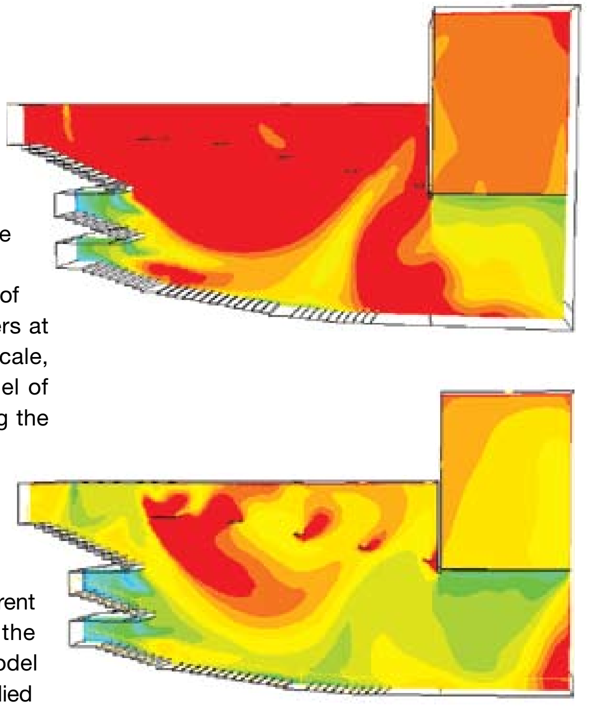
Full-scale 3-D model of the convention center assembly hall

In the assembly hall of the convention center, an underfloor air distribution (UFAD) system was coupled with a conventional ceiling air supply. The assembly hall that includes the stage and seating area consists of four levels (basement, ground, mezzanine and first floor) interconnected via one air continuum, with all levels occupied. Dar Al-Handasah was charged with reducing the HVAC system energy consumption while ensuring audience comfort: The total airflow supplied by the hybrid cooling system should cool the occupied zones but only temper the upper regions where maintenance catwalks are located. The distribution of air supply outlets needed to be optimized to ensure proper air delivery in the 24-meter-high assembly hall while avoiding disturbance of the thermal stratification of the air, a key energy-saving measure. Fluid dynamics



Rendering of the convention center at Princess Noura Bint AbdulRahman University for Women in Riyadh. The center assembly hall was studied to reduce energy costs while maintaining comfort.

simulations with ANSYS FLUENT software were fundamental in optimizing the UFAD system to maximize energy savings and reduce energy costs.



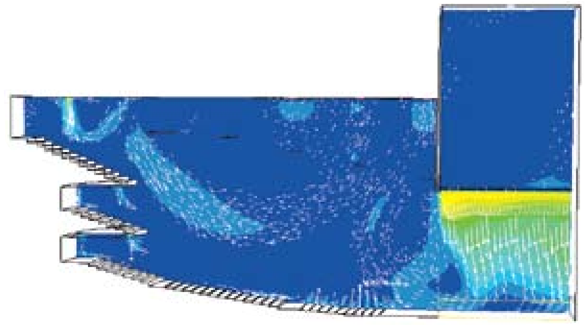
Original design scenario (top) and optimized design model (bottom) for temperature distribution across the convention center assembly hall

To analyze the performance of the proposed system, engineers at Dar Al-Handasah built a full-scale, three-dimensional virtual model of the assembly hall, representing the complex geometry of the building. They accurately modeled 902 air diffusers beneath the audience seats in the hall and 62 ceiling flow bars distributed throughout the different levels. For greater accuracy, the team then meshed the 3-D model with local mesh refinements applied in the occupied zones and near the air outlets. Engineers conducted steady-state simulations using the ANSYS FLUENT solver to optimize the performance of the proposed hybrid air conditioning system to produce an environment that complies with the required comfort conditions. In particular, velocity and temperature distributions were generated and air distribution refined so that no disturbances occurred in the hot stratified region.

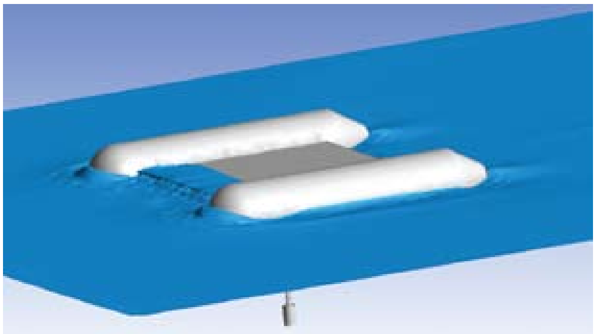
Initially, the flow simulation indicated a potential for improving the temperature distribution at various levels. In fact, high temperatures were observed at the first-floor-level seating area (around 31 degrees C) and at the stage area (around 25 degrees C), implying insufficient supply air flow to these zones. The team observed very high temperature (around 40 degrees C) in the core volume mainly due to high heat loads from the equipment on the catwalk. In addition, the mezzanine-level seating area was overcooled, with average temperature around 21 degrees C, indicating that the supply air flow delivered to that zone could be reduced.

Using the results from the initial fluid dynamics simulation, the engineers were able to visualize the airflow behavior inside the high-ceilinged hall and devise improved airflow delivery parameters, which were validated in a second simulation. Specifically, the supply air flow in the first-floor seating area was increased and directed at predefined angles, leading to better air delivery and accordingly lower temperature. Additionally, the team found the side flow bar diffusers at the balconies were inducing significant disturbances to the hot stratified air layer and, thus, engineers removed those diffusers from the design. Spot cooling was used to balance the high heat emitted by the catwalk's equipment so that disturbances would not be introduced into the upper hot stratified air layer. To capture the rising movement of buoyant hot air from the occupied zones, the team added air returns above the catwalks with an exhaust fan installed at the highest elevation of the stage. This fan also assisted in exhausting any contaminants.

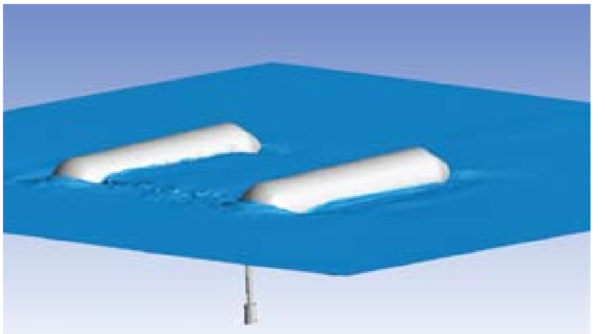
Fluid flow simulations have become an instrumental tool in supporting the company's design process through accurate prediction of thermal comfort conditions, design validation enabling design optimization, and energy reduction for the HVAC building systems. Furthermore, the proven breadth and depth of advanced fluid dynamics modeling capabilities from ANSYS have allowed the company to tackle a wide range of complex problems ranging from HVAC and smoke simulations to dispersion modeling and pumping stations simulations. Dar Al-Handasah, with the support of ANSYS channel partner Fluid Codes Ltd for the Middle East, continues to explore opportunities to incorporate state-of-the-art tools in building design to continuously improve design quality while exceeding client expectations. Fluid dynamics software from ANSYS helps engineers to optimize HVAC designs and meet the ongoing challenge of developing models that are energy efficient, sustainable and compliant with standards. ■



Optimized design model showing flow pattern and velocity distribution across the assembly hall. Simulation assisted the designers in meeting energy efficiency goals.



Sensor float fluid–structure interaction (FSI) transient response to current flow



Sensor float bow nosing down due to flow on sensor below float, deck awash

Don’t Rock the Float!

Fluid–structure interaction allows designers to assess impact of waves on freshwater and offshore systems.

By Richard Grant, President, Grantec Engineering Consultants, Inc., Halifax, Canada

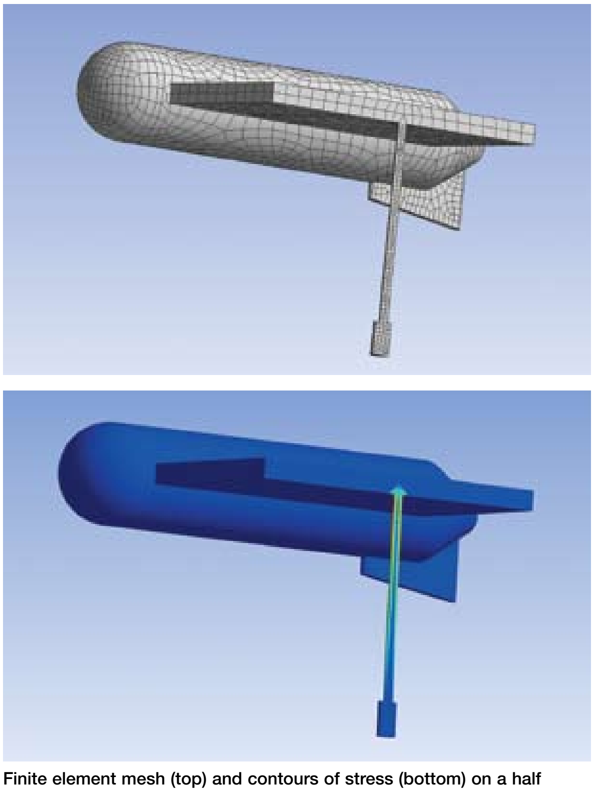
On a recent project commissioned by Environment Canada, Grantec Engineering Consultants, Inc. was tasked with developing a water quality monitoring float designed to carry a sensor for capturing environmental data. The float plays a role similar to — and looks somewhat like — a catamaran, though it is designed to be moored rather than driven by an engine or sails. The goal of the analysis was to minimize drag and ensure stability of the float as well as to develop specifications for the mooring system and structure. To meet this goal, Grantec used multiphysics simulation software from ANSYS to determine the fluid–structure interaction (FSI) by modeling the float and sensor under a wide range of water current and wave conditions.

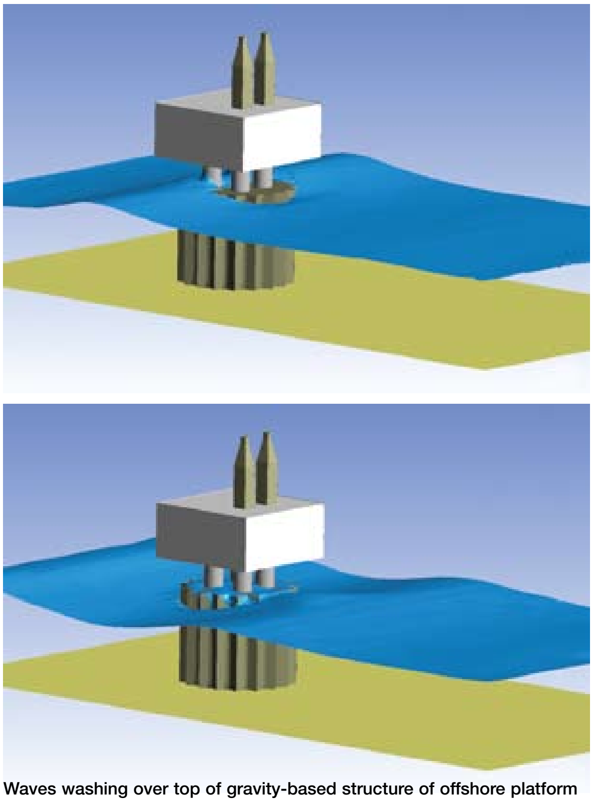
Based in the maritime province of Nova Scotia on the east coast of Canada, Grantec and its engineers have an extensive background in both structural and fluids analysis helping customers in the defense, offshore, marine, manufacturing, energy and aquaculture fields advance new designs and systems. More recently, however, Grantec has often faced the challenge of how to combine these two analyses that have historically been performed separately. Previously, when the interaction between fluid and structure was critical, Grantec’s engineers needed to enter the results from the fluid dynamics software manually into the structural analysis software and vice versa. In contrast, ANSYS offers a solution integrating several of its most powerful and trusted fluids and structures simulation tools. With its multi-field solver, the ANSYS FSI solution provided Grantec’s team with a bidirectional capability for time-transient or steady-state analysis with moving or deforming geometry. Using ANSYS Multiphysics software, the Grantec engineers were thus able to evaluate both the structural part of the analysis and the fluid flow solution with just a single tool.

In the original float design, the team modeled the float and sensor as a flow obstruction, which accounted for the flow currents and wave loading on the float as well as buoyancy forces. They then evaluated the development of bow and stern waves that result from the resistance of the hull to fluid flow, just as with the hull of a ship. The software duplicated the vertical heaving and angular pitching of the float in response to different wave and current conditions. The impact forces from the waves calculated in the fluid simulation were automatically passed back to the structural model to more accurately simulate the stresses and deformations on the hull. Though they have little effect on fluid flow, the stresses are important because they make it possible to optimize the design of the hull to a much higher level than would be possible without them.

With the FSI solution from ANSYS, Grantec evaluated the performance of a wide range of hull profiles and mass distributions under different flow conditions, and it took advantage of parallel processing to accommodate larger models more efficiently than using a single-machine environment. In the initial series of designs studied, the sensor was fixed to the stern of the float and extended vertically into the water. The FSI results for these designs showed the force exerted by water currents on the sensor combined with the bow wave tended to push the bow of the float underwater in faster currents. It was not practical to solve this problem by simply changing the hull design, so the team tried a hinged connection between the sensor and the float to reduce the load transmitted from the sensor to the float. The hinged sensor, however, greatly increased the complexity of the simulation analysis.

Grantec addressed the new challenge of the hinged-sensor design by modeling the float with the sensor fixed in different hinge positions using the immersed pipe element in the structural portion of ANSYS Multiphysics software. Unlike the extensive approach used for the non-hinge designs, this new method provided a more simplified way to perform FSI analysis. With the immersed pipe element, the team applied wave and current loading to the structural model without the computational load involved in coupling it to a full fluid dynamics analysis. In the future, Grantec plans to use a moving mesh to perform a more complete FSI analysis including full fluid dynamics simulation that will evaluate the motion of the hinge in response to hydrodynamic forces.

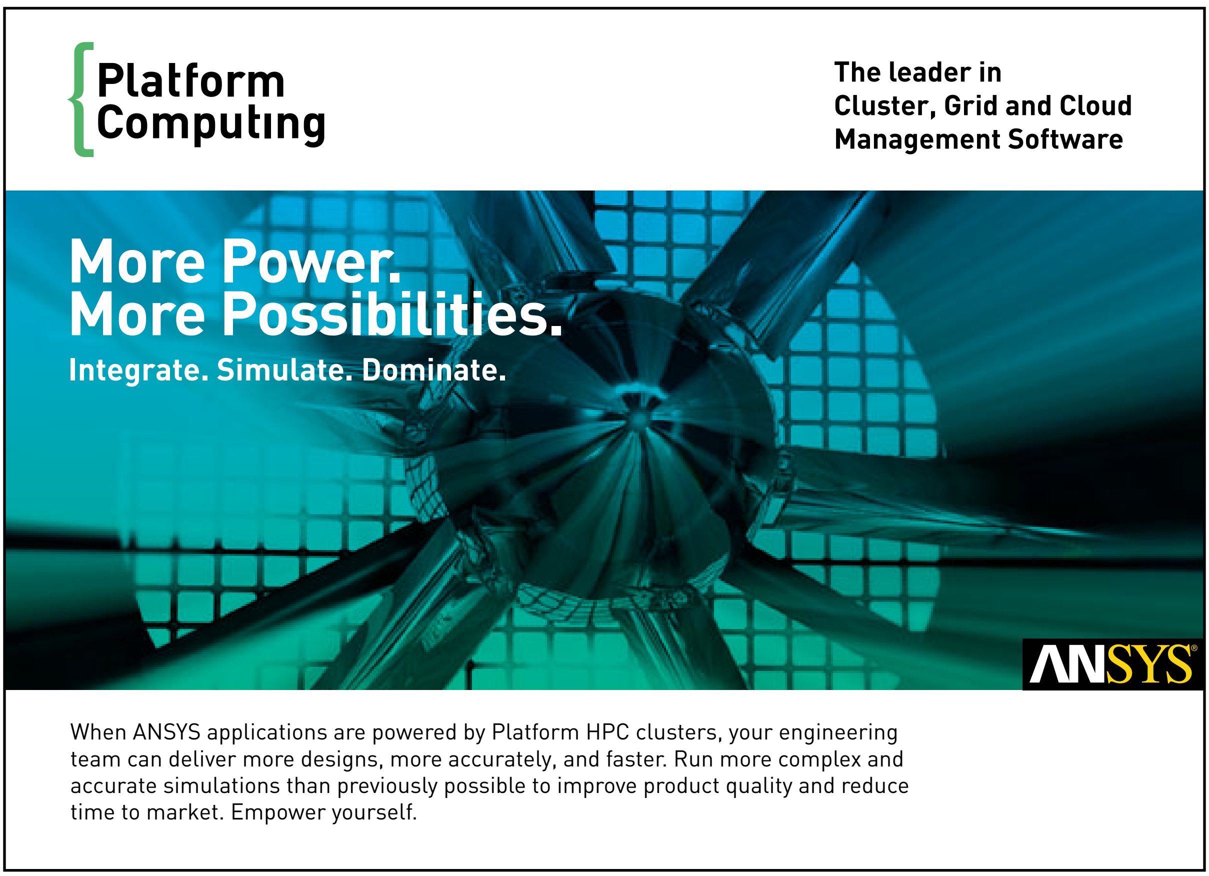

Finite element mesh (top) and contours of stress (bottom) on a half model of the sensor float. The FSI analysis was performed to look at the effect of a fixed flexible boom on the float.


Waves washing over top of gravity-based structure of offshore platform (waves traveling to the right)

Beyond its studies of water quality monitoring floats, the company has done extensive work with engineering simulation to help create safer and more structurally sound offshore structures and systems. Grantec's engineers have also used the ANSYS Multphysics solution to assess gravity-based structures (GBSs) used to protect offshore oil drilling and production platforms from icebergs. GBSs rely on weight to secure them to the seabed, which eliminates the need for pilings in hard seabeds. Concrete GBSs are typically built with huge ballast tanks so they can be floated to the site and, once in position, sunk by filling the tanks with water. The Grantec team used FSI from ANSYS to simulate wave loading a GBS including the effects of massive waves from storms — also known as green water — coming over its top.

The company believes that its investment in ANSYS Multiphysics software has made a significant addition to its analytical capabilities. Clients seek out Grantec because of its track record in performing advanced engineering to solve very complex problems. ANSYS technology has helped put another tool in the Grantec toolbox that makes it easier to address design challenges that just a few years ago would have been much more difficult. ■


Invented the Supercomputer, Reinvented the Workstation
The Cray CX1 brings high-return, low-risk benefits to design productivity and efficiency for ANSYS® FLUENT,® ANSYS® Mechanical,™ HFSS,™ and more.
Visit the Cray and ANSYS Partner Media Center at www.cray.com/reinvent
www.cray.com/reinvent
ANSYS®
CRAY


Platform Computing
The leader in Cluster, Grid and Cloud Management Software
More Power. More Possibilities.
Integrate. Simulate. Dominate.
ANSYS®
When ANSYS applications are powered by Platform HPC clusters, your engineering team can deliver more designs, more accurately, and faster. Run more complex and accurate simulations than previously possible to improve product quality and reduce time to market. Empower yourself.

Energizing the Wind Industry

Increased complexities require a system-level approach in designing and evaluating wind turbines.

By Ahmad Haidari, Global Industry Director for Process, Energy and Power, ANSYS, Inc.

Wind energy projects around the globe — from small installations to very large wind farms — have a common goal: to reduce unit energy cost while improving reliability. From a business perspective, technology contributes to viability by influencing efficient wind turbine design, manufacture, deployment and operation. Whether the application is an onshore, offshore or far-shore installation, advancements in science and engineering will contribute to the industry's success, especially through capabilities related to aerodynamic design, material science, structural design, electronic mechanical control, site selection and farm layout.

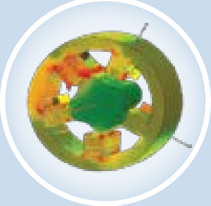
Wind turbines and wind energy projects are becoming increasingly more complex, so they must operate dependably at levels unimaginable a few years ago. Installations of very large wind turbines in offshore and floating configurations are a major technological achievement. Energy companies hope to design, install, and efficiently and reliably operate superstructures whose wind blade spans are over 50 meters and subject to wave and wind loading at different angles of attack.

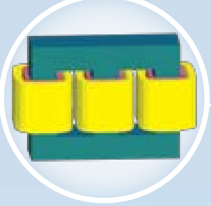
Historically, wind energy companies have used engineering simulation software as a point solution, used only to simulate a specific design aspect or analyze a component. Successful application of ANSYS solutions ranges across the wind energy industry, including:

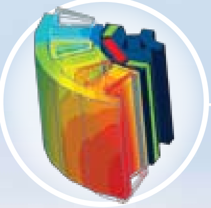
- **Aerodynamic design**: thrust coefficient, blade structural integrity, ultimate loads and fatigue, noise predication, wind gust fluid–structure interaction, bird strike, icing, boundary layer transition, near-wake and far-field studies
- **Structural design**: tower and rotor structural integrity/safety, power conversion efficiency, installation cost and maintenance, offshore transport and installation
- **Component design**: blades, gearboxes and bearings, generators, nacelles, rotors, drivers, motors, electronics cooling
- **Site selection and farm layout**: maximum project potential, power output (both peak and average), wind loads, fatigue
- **Turbine placement**: variable terrain, roughness, forestry, multiple wake effects, buildings and setbacks
- **Electromechanical system**: electrical machines, variable-speed control systems, transformers, power electronics, power distribution, sensor and actuator design
- **Blade manufacturing**

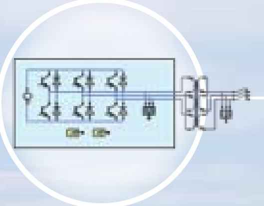
Today's increased complexities require a system-level approach in designing wind turbines and evaluating performance based on real-world conditions. Advances in engineering simulation software increasingly make this possible: For example, the ANSYS Workbench environment is designed with capabilities that enable modeling entire wind turbine systems. Its value is further enhanced through advanced solver functionality including turbulence transition models, advanced contact models, multiphysics capabilities, composites tools, high-performance computing and the flexibility to connect to third-party software for wind turbine blade manufacturing or aero-elasticity calculations.

Engineers can perform electromechanical system-level analysis using Simplorer software, electromagnetic analysis on electric machines and drives with Maxwell, wind power analysis via ANSYS CFD, and stress and modal analysis using ANSYS Mechanical. By leveraging high-level integration and advanced capabilities,


Speed sensor design

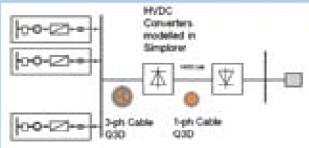

Transformer design


Electric machine analysis

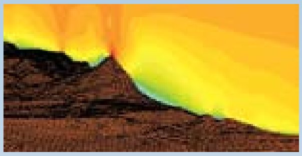

Power electronics design

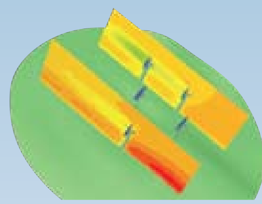

Generator and shaft design

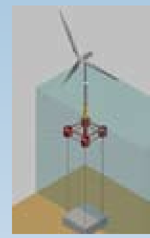
Wind Farm and Power Distribution



Power distribution analysis


Site selection, land and sea


Wind farm configuration for optimal power generation


Design of floating offshore wind turbine

engineers over time are extending their once-simplified simulations to include additional details in overall wind turbine design — enabling small efficiency gains, important in an industry in which a minute efficiency/performance gain can translate into much larger electricity production, reduced downtime and greater project profitability. Such details can also improve reliability and enable better wind energy project operation.

Already, there are many exciting examples of the expanding use of engineering simulation throughout the wind energy supply chain. Three separate wind energy applications are presented in this issue of *ANSYS Advantage* — and each highlights the breadth of ANSYS solutions. Without simulation capabilities, the projects presented may not have been as successful.

With increased demand for wind energy, engineers will face additional complexities, such as even-larger turbine blades that will be installed farther offshore and in harsher environments. Wind farm site selection must continue to reduce risk and overcome proximity and environmental concerns. New powertrains, lighter towers, multi-access turbines, floating platforms and quieter machines will be developed. The industry will innovate to meet the challenges of increased safety and reliability, improved remote monitoring, reduced system maintenance and regulatory concerns. ANSYS is keeping pace by providing high-fidelity integrated, advanced capabilities that meet single-physics needs as well as system-level and multi-disciplinary requirements of the wind energy industry. ■

Software from ANSYS meets the challenges of individual applications as well as of complete systems in the wind energy industry.

Some images courtesy CENER, IMPSA, REpower Systems AG, Trane Inc., Xanthus Energy Ltd. and iStockphoto.com/Tore Johannesen.

More Power to You

Simulation helps Indar to design one of the world's highest-efficiency permanent magnet wind turbine generators.

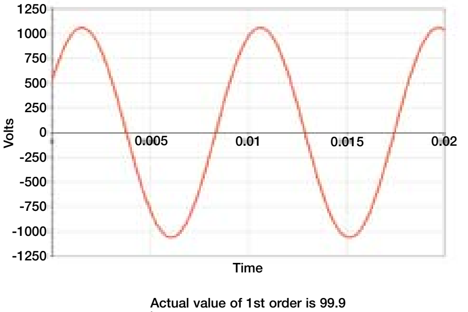
By Jon Vaquerizo, Project Manager, and Xabier Calvo, Technical Manager, Indar Electric, S.L., Beasain Guipuzkoa, Spain

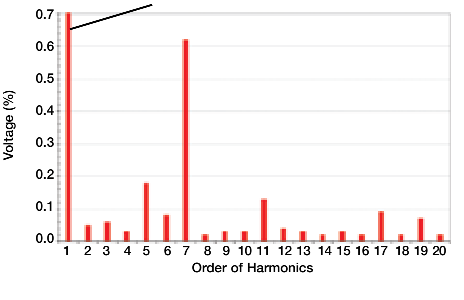
Indar high-speed permanent magnet generator with air–water cooling option

Wind power is the world's fastest-growing energy source, with 37.5 gigawatts of installed capacity added in 2009. The Global Wind Energy Council expects this resource to grow by 160 percent over the five-year period ending 2014. One rising trend is permanent magnet generators (PMGs), as they offer higher efficiency and design flexibility. Indar Electric, S.L., set out to develop a 2.5 MW PMG for wind power applications with the ambitious target of achieving an unprecedented 97.7 percent level of efficiency at rated load in converting mechanical to electrical energy using a permanent magnet generator. Another goal was to increase performance and efficiency at partial loads, because wind turbines often run at partial load. Traditional build-and-test methods could not achieve these goals in a reasonable amount of time. Thus, Indar applied electromagnetic field and fluid flow simulation to facilitate the process.

Indar Electric was founded in 1940 as a manufacturer of small electric motors. In 1997, it became part of Ingeteam, a Spanish renewable energy company that currently holds about a 15 percent global market share for wind power components. Indar produces a wide range of generator concepts, including more-traditional double-fed induction generators (DFIG) and newer PMGs. PMGs generally offer higher efficiency at rated load and even more at partial loads, since the permanent magnets eliminate the need for rotor windings that, in turn, remove rotor ohmic losses. PMGs also eliminate the need for brushes, which reduces possible problems and maintenance needs.

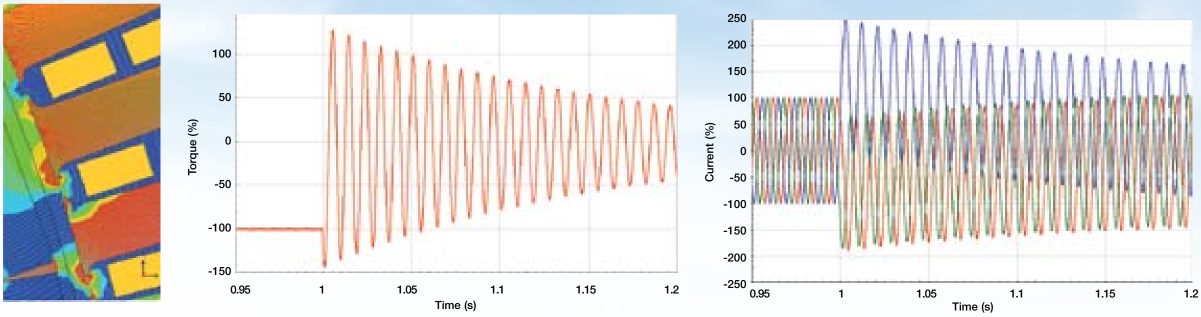
The Indar design team faced several major challenges in developing its newest PMG. Achieving high efficiency was the overarching goal, but there were a number of other targets that had to be simultaneously achieved for reliable operation. Cogging torque, caused by the interaction between the rotor's permanent magnets and the slots on the stator core, had to be reduced to 0.1 percent of overall torque. Voltage harmonics (THD) in the output had to be kept below 0.5 percent. For the cooling system, a demanding goal was set to maintain magnet temperature below 100 C to assure good performance of the magnets over a 20-year lifetime.





Magnetic flux density level in the stator (left) identifies areas of high losses. The generated voltage (upper right) and fast Fourier transform (FFT) (lower right) results for a no-load condition are also shown.

Indar engineers used Maxwell low-frequency electromagnetic field simulation software from ANSYS to evaluate the effect of different geometries and magnet properties on the electromagnetic performance of the generator. Well-known basic equations were used to develop a preliminary generator design.

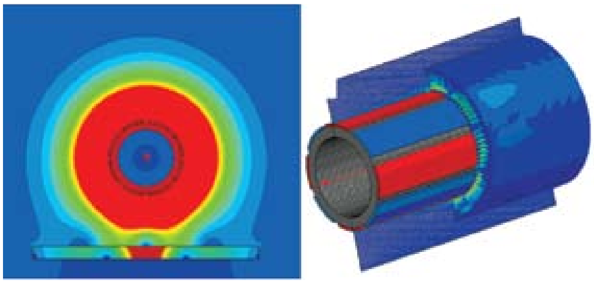


Magnet behavior in a three-phase short circuit as predicted by Maxwell

Engineers first created a 2-D and, later, a 3-D model of the generator, relying primarily on manufacturing drawings to reproduce the geometry and material properties of rotor and stator laminations and coils. The time step of the simulation was adjusted to match the rotating speed of the generator and the number of poles in the permanent magnet. Engineers simulated the performance of the proposed design under no-load, full-load and short-circuit conditions.

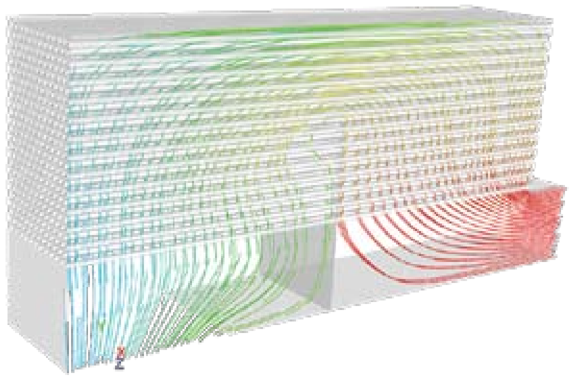
The results of the simulation included the voltage wave form produced by the generator, a measurement that was compared to the design requirement so that harmonic levels could be evaluated. The voltage fast Fourier transform output capability in Maxwell provides the voltage at different frequencies, making it straightforward to calculate the harmonic levels for a particular design.

The behavior of the design in the event of a short circuit was another important consideration. Short circuits may be caused by mechanical failure in the generator, insulation breakdown or power converter malfunction. Engineers studied the magnetic field generated in each area of the permanent magnet in a short circuit, expecting to ensure that the magnet had the right properties to avoid any damage. Generally when designing a PMG, Indar engineers consider a wide range of factors, including the contribution of magnet temperature, rotational speed, switching frequency and short-circuit (two-phase and three-phase) performance to achieve ideal magnet behavior for the entire lifetime of the generator circuit.



Electromagnetic simulation of the balancing operation and inserting the rotor in the stator with a crane

In the full-load simulation, engineers looked at the input required by Ingeteam frequency converters for the available switching frequencies to achieve nominal torque, high current and low losses. The team examined induction levels in the stator because of their important effect on efficiency. Though high induction levels make it possible to reduce the size of the generator, they also increase iron losses. Maxwell results showed the distribution of losses over the geometry of the stator, providing guidance for design changes to improve efficiency. Indar engineers continually modified the design, attempting to reduce losses in the stator copper, mechanical losses, and losses created due to switching frequency when working with frequency converters — all while achieving the other design requirements.



Flow speed and temperatures through the tubes in the air–air cooling system

Because of the magnet field's high strength even when the PMG is not rotating, Indar engineers simulated the process of assembling and balancing the rotor, ensuring it could be safely accomplished. They determined the level of magnetic forces generated while inserting the rotor, which made it possible to specify assembly tools that could withstand these forces. The generator rotor is balanced by placing it on pedestals instrumented with accelerators that detect forces generated by imbalance in the rotor. The electromagnetic field generated by the rotor during this process was simulated with Maxwell to ensure it did not interfere with the cables carrying the accelerometer signals.

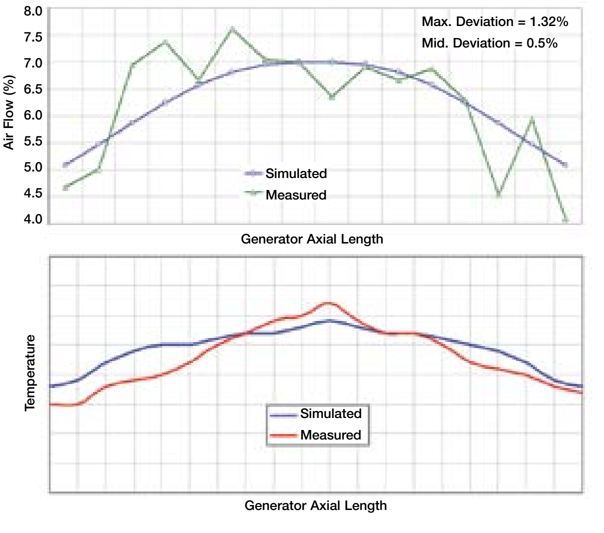
Fluid dynamics results show air flow variations over the length of the generator using three different cooling system options. The options involve modifying the geometry of the slot, windings and magnets.

Indar engineers simultaneously studied the generator's cooling system because of the interaction between electrical and thermal performance. The temperature of the magnet plays an important role in its ability to resist demagnetization, so improvements in cooling performance can increase the magnet's ability to handle a short circuit. Optimization of the cooling circuit helps to improve efficiency by reducing mechanical and cooling losses.

To optimize the cooling circuit, engineers used ANSYS FLUENT fluid dynamics software to perform a detailed study of fluid flow and heat transfer in and around the generator. Meshing was a challenge because of the difference in scale between the small 5 mm to 10 mm air gap between the rotor and stator, where accuracy was critical, and the large overall 1 meter length of the generator and cooling system. To minimize computational time, 3-D steady-state analysis was used during the majority of the design process, and the model size was reduced by using axial symmetry and periodic conditions. Fluid dynamics results included the local heat transfer coefficients, air flow velocity at every point in the machine circuit, pressure drop of the air circuit through the generator, generator temperature and thermal profile, and magnet working temperature. Engineers used these results as a guide in decreasing temperature hotspots by reducing variations in cooling over the length of the generator.

Using software from ANSYS helped Indar to easily explore multiple automated parametric design variations. The electromagnetic and fluid flow simulations provided far more diagnostic information than was available from physical testing. Simulation provides results for any output at any point in the computational domain, while physical testing provides results only at locations where it is practical to locate sensors. Engineers were able to iterate to a design that met all their specifications long before a prototype was built. Pervasive parameter management was performed quickly and easily by changing design parameters. These parameters were propagated throughout the entire design system, from CAD model through meshing and boundary conditions to generation of updated results.

The next step was building and testing a real-scale prototype to verify the simulation results and to ensure the generator's functionality and life expectancy. Two types of testing were performed at the test bench: generator full-load homologation testing to certify generator performance and durability testing to verify its reliability over time. The electrical and thermal measurements of the physical prototype matched up very well with the simulation results. For example, the maximum deviation from the voltage shape prediction to the measured values was 0.1 percent. The measured efficiency of the new generator was 97.86 percent, higher than the design target of 97.7 percent, and nearly exactly what was predicted by the simulation. The rating is one of the highest levels of efficiency for any permanent magnet generator on the market. Simulation made it possible to achieve this challenging performance goal in less than half the time that would have been required using conventional build-and-test methods. The simulation predictions correlated well with physical testing, providing confidence that Indar can use simulation to optimize its products to deliver high performance under the most demanding conditions. ■



The difference between the simulated and measured air flow rates across the length of the generator was a maximum of 1.32 percent. The maximum difference in calculated versus measured temperatures was only +/– 3 degrees C for the stator and +/– 3 degrees C for the rotor magnets. Software from ANSYS provided very good accuracy, which is essential for these applications and for Indar engineers to trust the new design variations.

Where the Wind Blows

Engineering simulation plays a role in getting the most power from wind farms by predicting the best available locations.

By Ian Jones, ANSYS Fellow, and Christiane Montavon, Senior Technical Services Professional, ANSYS, Inc., and Daniel Cabezón, Wind Specialist, National Renewable Energy Centre (CENER), Sarriguren, Spain

Driven by the need to change the power generation energy mix, the use of wind turbines is increasing significantly around the world. The Global Wind Energy Council (GWEC) predicts the global market for wind turbines will grow from 94 GW in 2007 to 296 GW of total installed capacity by 2012. Depending on the increase in electricity demand, wind power could supply 11.5 percent to 12.3 percent of global electricity demand in 2020, according to GWEC, and between 18.8 percent and 21.8 percent in 2030.

However, the uptake for land-based installations is not as great as it could be. Some of this is explained by planning considerations: People do not want to have wind turbines sited near their homes. Typical objections include visual intrusion, noise, the effect on nature, and disturbances caused by construction and operation.

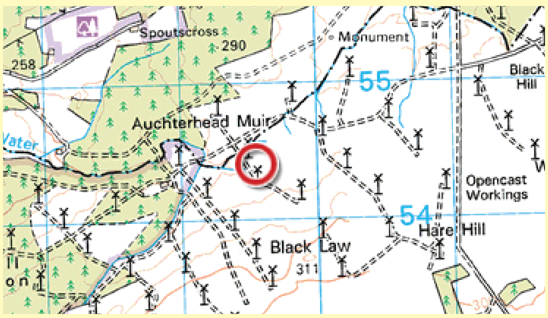
Furthermore, wind farms affect radar signals, a condition that is forcing turbines away from exposed hilltops to locations that are out of sight of antenna installations. Consequently, wind farm developers are considering sites not necessarily ideal for wind power generation: locations away from hilltops, in regions where the flow can recirculate and turbulence can be high, and in heavily forested areas. Packing turbines into such constrained sites means that wake effects can have a significant impact on the performance of downstream turbines, reducing power production and considerably increasing fatigue loading.

Black Law site
Courtesy Siemens press.

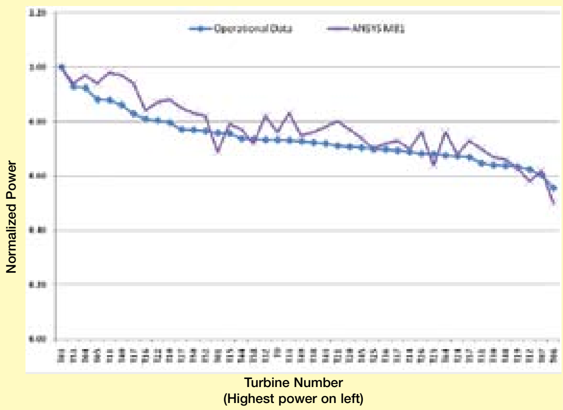
Black Law Wind Farm

A typical wind farm example is Black Law, in central Scotland, operated by ScottishPower Renewables. The site contains 54 2.3 MW Siemens turbines and occupies a former open-cast coal site. In January 2006, it was the largest operating wind farm in the United Kingdom.

With relatively small height variations across the wind farm (approximately 170 meters), the complexity on this site is less related to topography and more associated with significant forestry and wake effects. ScottishPower Renewables and ANSYS have carried out a collaborative study on this wind farm to understand the interactions between the turbines, the terrain and the forestry, to assess how well fluid dynamics would perform for this site, and to use the lessons learned in developing new sites. A comparison of ANSYS results and measurements of the annual average power for the Black Law site shows that the software predicts performance well. The results also indicate which turbines are underperforming, for example, because of their location in the forest.



Map of Black Law Ordinance Survey® Crown Copyright 2008, license number 100048580



Comparison of average wind power on Black Law, taken from operational data and ANSYS simulations, shows close agreement — giving ScottishPower Renewables increased confidence in using fluid dynamics simulation for future site application.

To overcome the shortage of land sites, interest is growing in offshore turbine sites, where low ambient turbulence is expected to result in reduced rates of wake recovery. To minimize wake losses and optimize energy production, it is vital to improve confidence in the ability of models to accurately capture these effects.

No matter the location, there is considerable reluctance to invest in expensive wind farms without a guaranteed return on investment (ROI). Using current prediction tools, developers are unable to provide sufficiently reliable estimates of projected wind power from challenging sites, failing to satisfy investment criteria. Developers are now looking for ways to make wind power estimation more reliable for non-ideal sites.

Turbine blades must withstand significant stresses caused by strong rotation and wind buffeting. There is considerable interest in extending turbine life and reducing operational costs. Understanding the wind regime and turbulence levels enables greater reliability in assessing suitable turbine types. Under the International Electrotechnical Commission (IEC) (the recognized international body for standards development activities) turbine classification system, the choice of turbine type is determined by the levels of turbulence intensity likely to be experienced at the site. For example a Class IIIB wind turbine, designed for lower wind speed (7.5 m/s at hub height) and lower turbulence intensity (16 percent), will be subject to lower loads than a Class IA or IIA wind turbine.

"This type of simulation is useful in highlighting features not picked up in other engineering models and in bringing to light the circumstances under which standard models fail to capture the detail required. Therefore, it has a key role to play in managing the risks associated with modeling complex wind farms."

– Callum Strachan, ScottishPower Renewables

A Class IIIB machine can therefore be larger both in terms of rotor diameter and hub height to capture a larger portion of the available energy which will translate in higher capacity factors compared to Class I and II wind turbines. However, they cannot be used at sites that experience a high level of turbulence intensity. Understanding turbulence allows a greater choice of turbine types and enables matching the type to the site conditions.

The use of ANSYS CFD solutions has increased significantly for modeling complex flows that arise in wind farms as a result of the terrain, forest and wake effects between turbines. Now a customized ANSYS tool helps designers to understand wind farm efficiency for complex flows and overcome technology limitations: WindModeller. The solution assists developers who don't have substantial CFD expertise to benefit from automated analysis of complex flows. Advanced users can further customize the setup and write post-processing scripts to capture their own expertise.

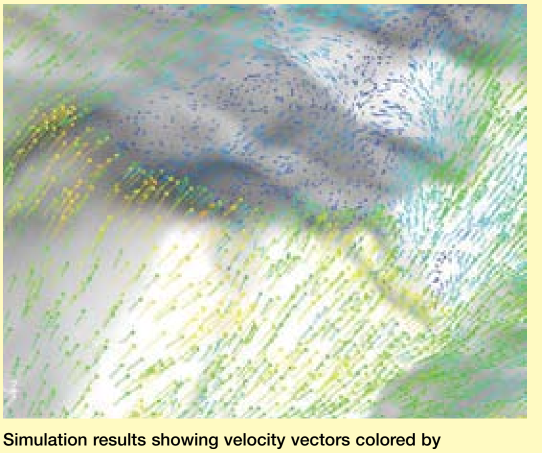
An add-on tool for ANSYS CFD — CFDWind — was created by El Centro Nacional de Energías Renovable (CENER, or National Renewable Energy Centre), a technology center in Spain that specializes in applied research, development and promotion of renewable energies. The tool contains advanced models for atmospheric boundary layers, enabling developers to take into account complex climatic conditions when developing projects. Fluid dynamics tools from ANSYS provide greater understanding of the wide variety of factors that need to be considered when designing wind farms. The flexible nature of the software provides an excellent platform for customization specific to this industry. ■

References

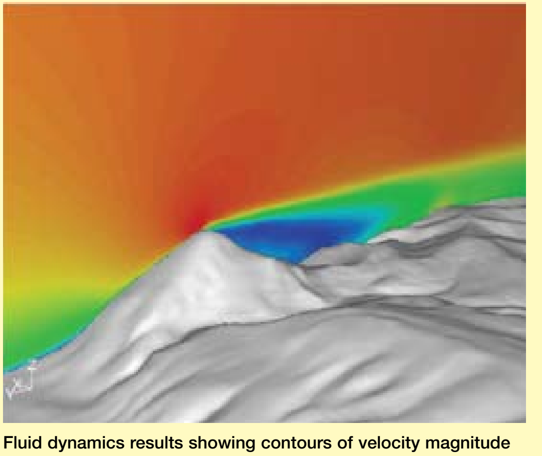
1. http://www.wind-energy-the-facts.org/en/factsheets.html.
2. Montavon, C.A.; Jones, I.P.; Staples, C.; Strachan, C.; Gutierrez, I. Practical Issues In the Use of CFD For Modelling Wind Farms. *Proc European Wind Energy Conference*, **2009**.
3. British Wind Energy Association: England's Regional Renewable Energy Targets: Progress Report, **2009**. http://www.bwea.com/pdf/publications/RRETProgressReport.pdf.
4. Turbine Classifications: IEC-1400-1. http://windwire.blogspot.com/2009/05/iec-classification-of-turbines.html.
5. BWEA: Calculations for Wind Energy Statistics – Emissions Reductions. http://www.bwea.com/edu/calcs.html (accessed 16 July, 2009).
6. UK Renewable Energy Strategy (June 2008) HM Government Department for Business Enterprise & Regulatory Reform (BERR).
7. BWEA UK Wind Energy Database.
8. Climate Change Act 2008 (Chapter 27).
9. Milborrow, D. Dispelling the Myths of Energy Payback Time. *Wind Stats*, **1998** (Spring), Vol. 11, No. 2.
10. Cabezon, D.; Iniesta, A.; Ferrer, E.; Marui, I. Comparing Linear and Nonlinear Flow Models, Proceedings of EWEC 2006.

Alaiz Wind Farm Test Site

The Alaiz hill, located in northern Spain, is situated in an east–west direction. The site is 4 kilometers long and approximately 1,050 meters high, and it comprises very rugged terrain. This hill has been used as a test site for gathering wind farm data and evaluating different simulation methods. Detailed calculations have been carried out at CENER employing ANSYS CFD software as well as using an industry-standard linearized model. Comparisons showed significantly lower errors in the predicted wind speed for the fluid flow simulation, on average about 1.75 percent, compared with about 5.4 percent for the linear models. This demonstrates the real benefits that fluid dynamics simulation can provide.



Simulation results showing velocity vectors colored by velocity magnitude for Alaiz wind farm test site



Fluid dynamics results showing contours of velocity magnitude for Alaiz site. Fluid dynamics software from ANSYS was able to accurately model this complex terrain.

Second Wind

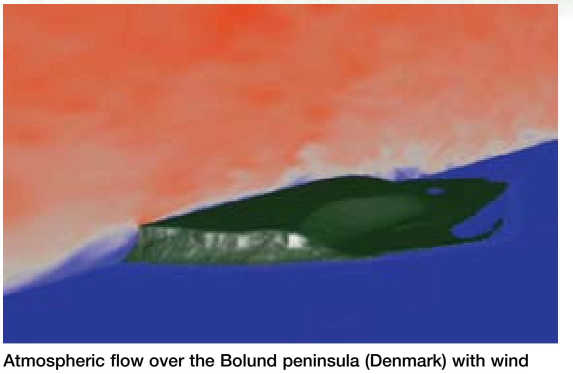
Advanced turbulence models lead to optimized wind turbine spacing.

By Thomas Hahm, Fluid & Energy Engineering GmbH & Co. KG, Hamburg, Germany

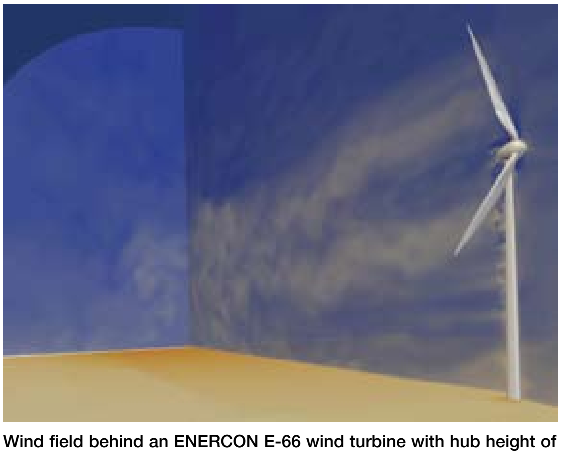
With the Global Wind Energy Council projecting that wind power installations will grow to 409 gigawatts in 2014, the amount of land occupied by wind farms is becoming a serious concern. The issue is already critical in Germany because of dense population and the country's leadership in deploying wind power. Typically the goal is to generate as much power as possible at the lowest cost from a given wind farm site. But there are drawbacks to spacing wind turbines too close together, including lower power output and wake effects.

Fluid & Energy Engineering (F2E) GmbH & Co. KG is pioneering the use of advanced turbulence models such as large eddy simulation (LES) to increase the accuracy with which turbines can be sited within a wind farm. The end result will be the ability to generate more energy from a given volume of land.

The wake of a turbine has significant effects — including reduced power output and shorter turbine life — on any turbine in its path. Extracting energy from incoming wind causes a loss in the kinetic energy and velocity of the wind in the wake of a turbine. This energy is recovered over distance as the wake exchanges energy with the surrounding wind. Turbulence produced in the wake of turbines and turbulence produced by terrain features such as forests or hills can have a substantial impact on turbine life. The effects of turbulence can vary with the flow angle and inclination of the incoming wind. For example, one of the most damaging scenarios is when one half of a rotor experiences turbulence from an upwind turbine and the other half is exposed to undisturbed flow.



Atmospheric flow over the Bolund peninsula (Denmark) with wind direction from the escarpment on the west side. The red color indicates high wind speeds, and blue indicates low speeds.



Wind field behind an ENERCON E-66 wind turbine with hub height of 65 meters. The blue color indicates high wind speeds, and white indicates low speeds.

Meandering wake behind an ENERCON E-66 wind turbine. Blue represents high wind speed, and white represents low speed.

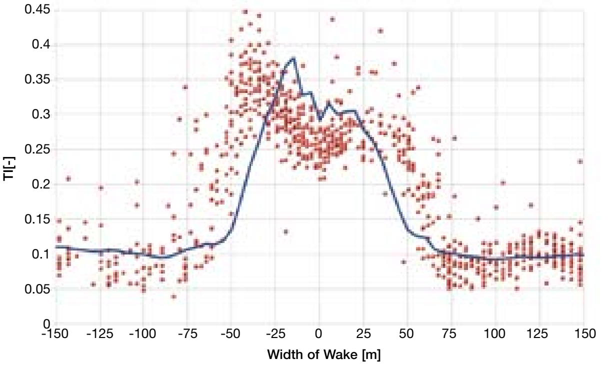
In Germany, loan organizations and regulatory authorities require an assessment of every new wind farm project to ensure that turbulence loads generated by terrain and wind turbine wakes are within acceptable design limits. Terrain and wind turbine wakes must be considered together because they have additive effects on turbulence. Typically, the approach to computing these loads is to use standard empirical formulas to estimate terrain-generated and wake-generated turbulence intensity and to use blade momentum theory to estimate the loading on the blades.

However, the accuracy of these empirical calculations can be limited because they do not incorporate the actual geometry of the rotors and terrain. The result is that the calculations need to be heavily calibrated to produce safe wind farm designs. These empirical models show that wind turbines typically must be spaced at a distance of about four rotor diameters from each other. Required spacing can be even greater if terrain-generated turbulence is a major factor.

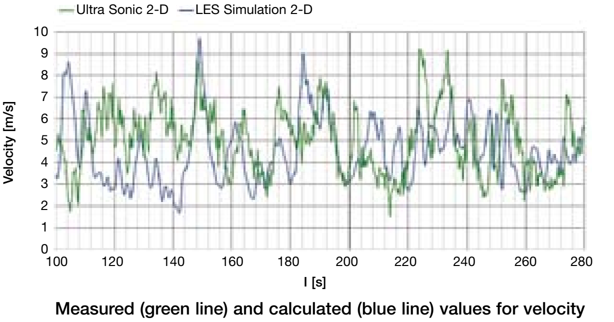
Developers of wind farms are demanding more accurate methods for calculating the turbulence intensity generated by wakes and terrains so that they can produce more energy from a given parcel of land. F2E has been using computational fluid dynamics (CFD) over the last decade to address this important issue. The company's engineers generally have had little difficulty in determining the wind velocity in the wake of a turbine, but calculating the turbulence intensity is much more challenging.

Conventional Reynolds-averaged Navier–Stokes (RANS) models reduce the computational time required to simulate turbulent flows by time-averaging the velocity field, pressure, density and temperature over time. This approach eliminates turbulence fluctuations and makes it possible to model turbulent flow in a reasonable time on desktop computers. RANS methods are effective at predicting the overall and steady-state behavior of a wind farm; however, their accuracy suffers in modeling unsteady turbulent flows that are typically found in wind farms.

Over the past three years, F2E has used the LES turbulence model with very positive results. It numerically resolves the larger turbulence scales and models the smaller scales to provide accurate transient solutions of the flowfield. Most recently, F2E engineers used ANSYS FLUENT software to model the full geometry of two ENERCON E-66 wind turbines with 66 meter rotors using the LES technique. The fluid dynamics results were validated with data collected in an actual wind farm using ultrasonic anemometers. Fluid dynamics accurately predicted the velocity of the incoming wind along with turbulence intensity at the downwind turbine over the width of the wake.



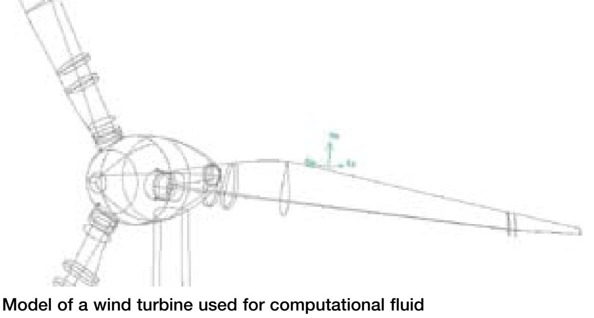
Simulation of local turbulence intensity (blue line) compared to measurements (red dots) across width of wake



Measured (green line) and calculated (blue line) values for velocity magnitude at hub height in center of wake over 180 seconds

Fluid dynamics did a good job of predicting the variation in the velocity over a period of 300 seconds, a figure that can be used to calculate turbulence intensity and the aerodynamic loads that turbulence creates. Quick horizontal shifts of the wake from one side to another are detectable on a 25 second time scale. Another critical factor in achieving this level of accuracy was the accurate modeling of the incoming wind field and, particularly, the variation in its wind direction and the wind shear.

Fluid dynamics and LES techniques from ANSYS can be used to calibrate and greatly improve the accuracy of empirical calculations that are currently used to calculate blade loads under wake conditions. Calculation of blade loads should make it possible to substantially increase the amount of energy that can be generated by new wind farms and to increase the safety and the lifespan of wind turbines. ■



Model of a wind turbine used for computational fluid dynamics simulation

Computer-generated image of the Terrafugia Transition in flight. The company unveiled a production prototype in 2010.

Fast Lane to Sky High

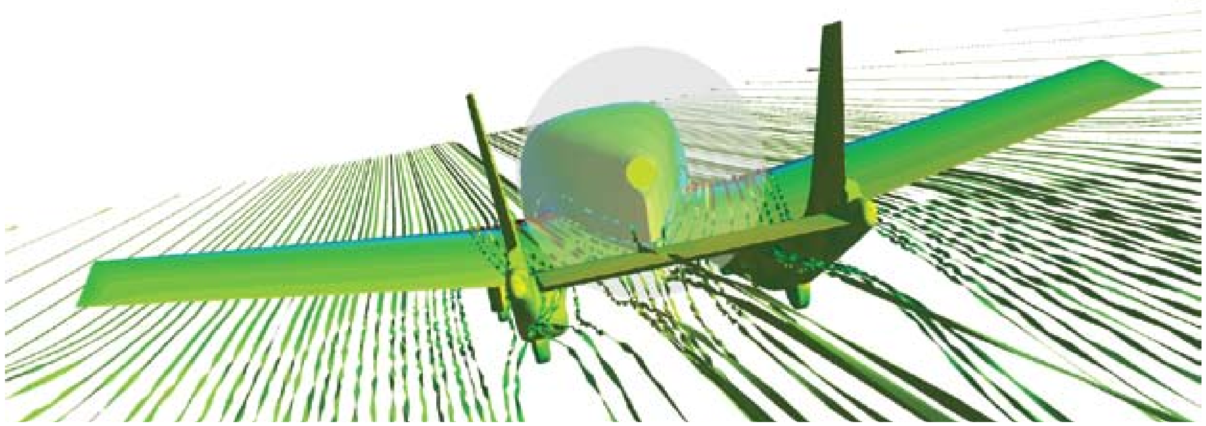
Fluid flow simulation software co-pilots design of production prototype roadable aircraft.

By Gregor Cadman, Engineer, Terrafugia, Woburn, U.S.A.

Since the earliest days of the aviation industry, inventors and entrepreneurs — from motorcycle racers to homebuilt-aircraft enthusiasts to the largest automakers — have sought to develop the iconic mashup of future transportation technology known as the "flying car." Some of their attempts did manage to test successfully and even reside in the Smithsonian. Sporting names such as Autoplane, Aerobile and Airphibian, these machines were impressive for their time, but they never lived up to their mythological pedigrees as conjured by science fiction authors and filmmakers. The concept is a proven one. But due to the engineering challenges involved in combining a lightweight, aerodynamic aircraft with the stability needed for long-distance driving, to date none of these vehicles, which are more accurately called "road-ready" airplanes, have been successfully brought to market.

Recently, however, the Boston-area startup Terrafugia revealed its production prototype for the first commercially available street-legal aircraft with hopes of beginning manufacturing in 2011. Terrafugia — Latin for "escape from land" — used simulation tools from ANSYS to arrive at a production prototype of its innovative Transition® Roadable Aircraft. After earning global attention following successful test flights in early 2009, the new prototype was revealed to the industry in 2010 at the annual Experimental Aircraft Association (EAA) AirVenture Oshkosh national airshow. It was a critical step in commercializing this one-of-a-kind vehicle.

With a flight range of up to 490 miles and cruising speed of 105 mph, the Transition can also drive up to 65 mph on the road. It is capable of transforming from plane to car in less than 30 seconds. The sophisticated design features foldable wings that span over 26 feet, a rear-wheel-drive system for the road, and a rear pusher propeller for flight. While the Transition is not designed to replace anyone's car, it is intended to drive at normal

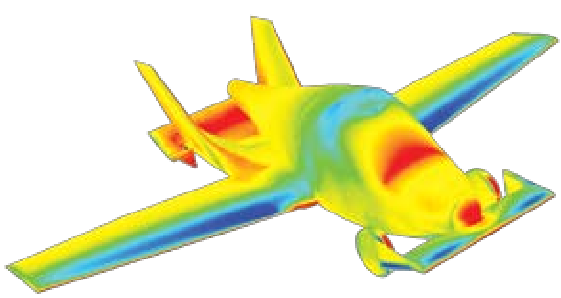


Rear view showing pathlines passing by Terrafugia's beta prototype design flying at 105 mph

The Transition's foldable wings allow the aircraft to be driven on the highway as well as parked in a typical residential garage.

highway speeds, so that owners can easily access their local airports. This required Terrafugia's engineering team to simultaneously consider the aerodynamics of flying and driving — activities that exert very different forces on the vehicle. While physical tests in a wind tunnel helped to validate the initial concept design, they were both time- and cost-intensive. For this reason, Terrafugia engineers turned to ANSYS FLUENT software to make and verify design modifications for the new production prototype, working in a virtual simulation environment that saved time and money while also enabling engineers to assess a complex range of design considerations.

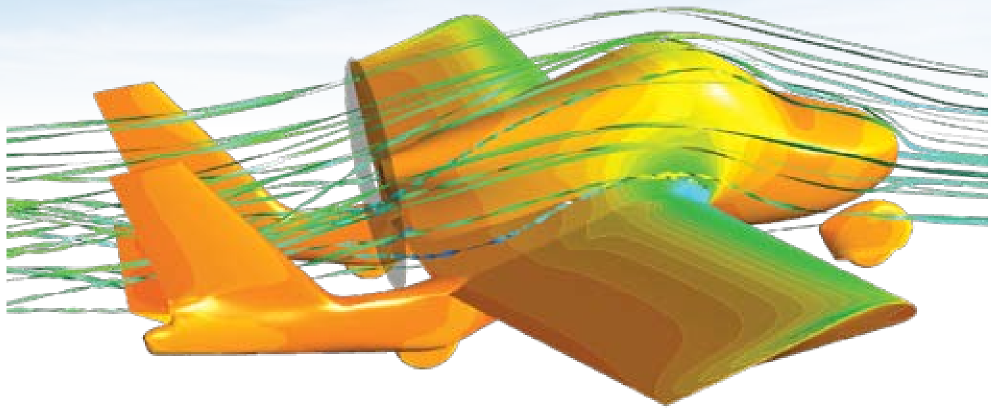
Unlike a typical car or plane, the Transition has a host of extra components to consider when analyzing air flows around the vehicle, as wheels, propellers, foldable wings and other shapes affect dynamic flows whether it is driving or flying. Using the modeling capabilities of the software, Terrafugia engineers conducted whole-vehicle airflow tests to study the effects of the slightest design change on overall performance. The team addressed issues such as maximizing wing lift in the air while simultaneously minimizing the effects of crosswinds along the road. Without an ability to work in a virtual environment, Terrafugia's team would have had to construct complicated physical models, modify or rebuild them, and conduct hours of real-world testing. Simulations powered by fluid dynamics software from ANSYS enabled rapid testing and verification of some modifications to the Transition design, based on the physical performance of the initial proof-of-concept vehicle.



Proof-of-concept design of the Transition showing pressure contours on the vehicle surfaces. Since the Transition was classified as a multi-purpose passenger vehicle, a bumper was not legally required, so engineers conducted simulations to see if they could eliminate this feature. ANSYS models confirmed that the canard was not needed, and this feature has been eliminated in the production prototype design.

Wind tunnel tests revealed an adverse interaction between the vehicle's front suspension and its canard, which serves as a wing while flying and a front bumper while driving. Rather than relying on additional physical testing, Terrafugia used ANSYS technology to further explore this interaction as well as potential solutions. As the work of the engineering team progressed, it became clear that, while the canard configuration had initially been integral to the design, it was undesirable from a number of standpoints. With the Transition receiving classification as a multipurpose passenger vehicle, the full-width bumper requirement for passenger cars — the original reason for the canard — was no longer applicable. Engineering simulation software verified that lighter weight, better flight characteristics and improved looks were all potential benefits of a canard-free design.

Another challenging aerodynamic design aspect of the Transition was attaining a wing stall speed — the speed at which an aircraft stops flying — of under 52 mph, which is a requirement for the light sport category of aircraft. Since slower in-air speeds generally create safer flying conditions, it was important for the Terrafugia engineers



Production prototype design showing airflow pathlines over the vehicle body. The VBM plug-in to ANSYS FLUENT enabled Terrafugia engineers to model the vehicle's propeller under near-stall conditions, which helped to ensure the safety of the aircraft while in flight.

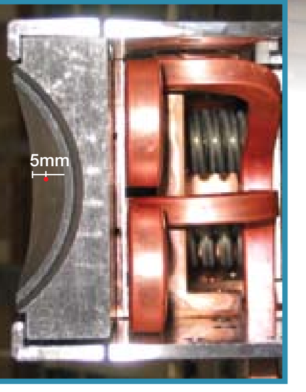
to design the vehicle to operate at a low speed for safety and stability without stalling. Stall prediction can pose a difficult problem, even for sophisticated CFD tools. However, with close support from ANSYS experts, Terrafugia developed a detailed engineering approach necessary to obtain accurate predictions, including the use of a virtual blade modeler (VBM) that plugged in to the fluid dynamics software and created additional capabilities to model the Transition's propeller. After deploying the specialty VBM tool, the team reshaped the wing and the remainder of the body as needed to match the weight and center-of-gravity requirements of the vehicle.

The new design improves both the in-air and on-road performance of the Transition as well as ensures that the vehicle lends itself to full-scale manufacturing. Terrafugia's team found that simulation software from ANSYS was critical in advancing them to the production prototype stage with a high degree of confidence in their design. The company credits ANSYS tools for helping to bring their product to market so quickly.

Glass Jaw

Simulation helps to solve collimator jaw design problem in the Large Hadron Collider.

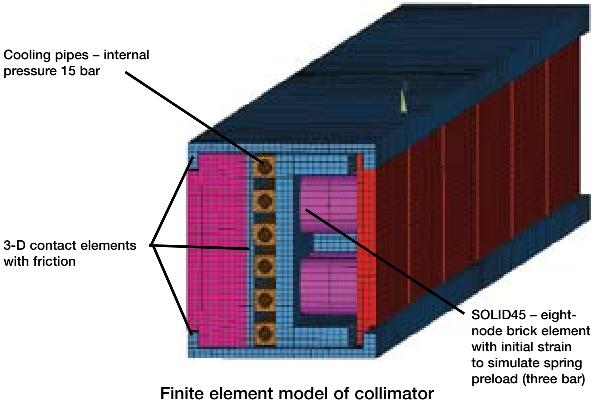
By Alessandro Bertarelli and Alessandro Dallocchio, Engineering Department, Mechanical and Material Engineering Group, European Organization for Nuclear Research, Geneva, Switzerland



Collimator cross section

The Large Hadron Collider (LHC) at the European Organization for Nuclear Research (Conseil Européen pour la Recherche Nucléaire, or CERN) is the world's largest and most powerful particle accelerator. Two beams of subatomic particles called hadrons — either protons or lead ions — travel in opposite directions inside the circular accelerator, gaining energy with every lap. The LHC's collimators scrape away particles that have gone slightly off track to prevent damage to the highly sensitive superconducting magnets. But the collimators need to be able to withstand an error that might pound them with a substantial fraction of the beam itself — considering that the beam could melt almost 1 ton of copper.

When researchers tested the LHC collimator prototype with several shots at different beam intensities, they discovered that the carbon–carbon collimator jaw nicely survived the impact, but its metal support suffered a permanent deflection strong enough to put the collimator out of action. CERN staff immediately set to work to understand what caused the problem and how to correct it.



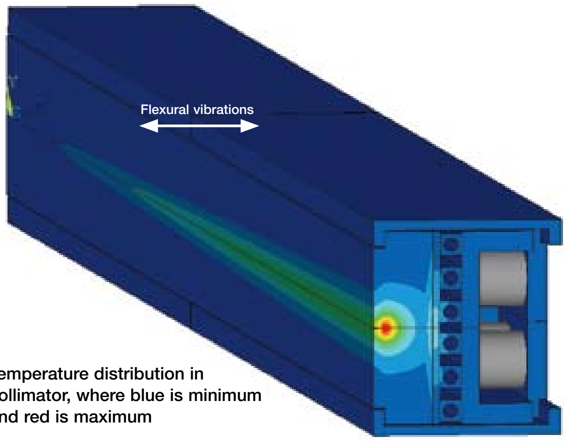
Finite element model of collimator

Sited near Geneva, Switzerland, the LHC is being exploited by physicists to study the smallest known particles that are building blocks of all things. The team uses the LHC to re-create conditions that existed just after the big bang by colliding two beams head-on at very high energy. The target of study is the Higgs boson, a theorized but not yet discovered particle that may help explain the masses of various subatomic particles. The LHC is being used to seek out dark matter that is believed to make up 96 percent of the mass of the universe. Scientists hope to discover the differences between matter and antimatter to help in understanding why so little antimatter is left in the universe even though the big bang is believed to have produced equal amounts of matter and antimatter.

The LHC uses a stored energy of 360 MJ per beam. This is two to three orders of magnitude above what other proton colliders can handle. CERN's collider is contained in an underground circular tunnel, 50 meters to 175 meters deep, with a circumference of 27 kilometers. The collider tunnel contains two adjacent parallel beam pipes, and each contains a proton beam that rotates in opposite directions around the ring. The parallel beams intersect at four points, forcing them to collide with each other. Some 1,232 dipole magnets keep the beams on a circular path, and an additional 392 quadrupole magnets focus the beams to increase the chances of collision at the intersection points.

The most critical elements of the collimators are the jaws, which are made of carbon–carbon composites, which encircle the beam and are designed to block any stray particles that separate from the beam. The jaws are supported by a brazed sandwich structure encompassing the main support bar, cooling pipes and interface plate.

The collimator is at risk when the beam is moved from one accelerator line to another. This needs to be done frequently because the beam is accelerated to maximum energy by moving it through a family of progressively more powerful accelerators. Kicker magnets are used to knock the beam out of its current line and inject it into the new line. If there is an error in the configuration of the kicker magnets, the beam might be steered too much in one direction. The collimator must be able to withstand accidents such as an injection error that may cause 3.2×10^{13} protons each with an energy of 450 gigaelectronvolt (GeV) to hit the collimator jaws. An electron volt is the kinetic energy gained by an electron when it accelerates through an electric potential difference of 1 volt.

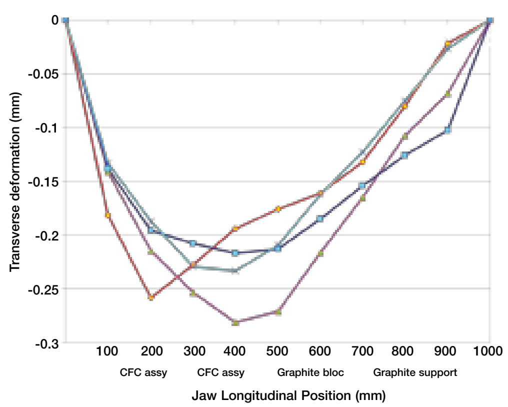


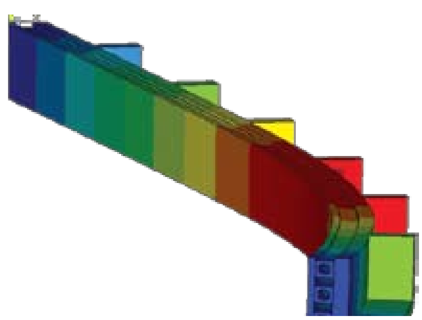
Temperature distribution in collimator, where blue is minimum and red is maximum

To validate the collimator design, engineers carried out tests using another CERN accelerator on a fully operational prototype of a collimator. The jaws were submitted to a series of impacts at 450 GeV over 7.2 μs. Measurements performed on jaw assemblies and metal supports revealed a permanent deformation of the metal support of over 300 μm. CERN researchers theorized that this deformation was caused by thermally induced vibrations due to very fast heating. Even when a structure is free to expand, when the heating process is faster than the typical stress relaxation time, material inertia prevents free thermal expansion, causing stress waves. Simulation was clearly required to better understand the problem.

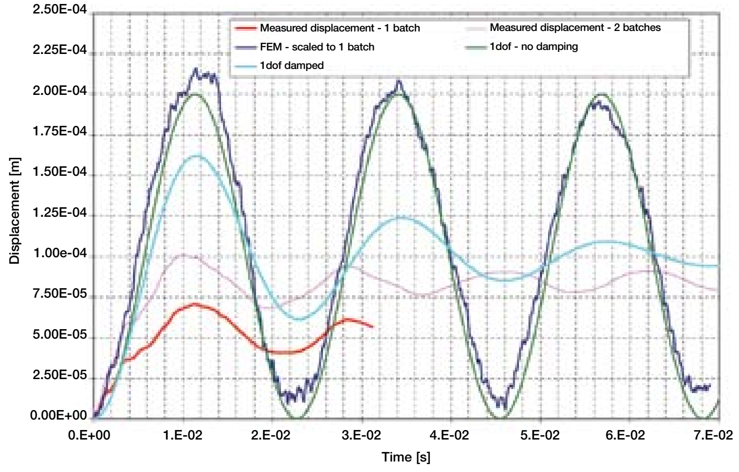
Applications with such great mechanical complexity are typically addressed with explicit dynamics codes. But for this case, explicit analysis would have been very awkward because of the need for multiphysics integration and the complexity of the mechanical structure, which includes different materials that are held together with clamps. Since the stresses are well below the elastic modulus, there are no shock waves that would require an explicit dynamics code. The problem could instead be simulated using an implicit finite element model following the rules of thermo-elasticity. ANSYS Mechanical implicit finite element analysis software provides multiphysics capabilities that integrate thermal and mechanical analysis. In this application, integration increases accuracy and reduces the amount of time required to prepare the model. ANSYS Mechanical also offers comprehensive element technology, an extensive library of material models, auto contact detection for assemblies and powerful solver capabilities.

The collimation block was modeled as a rectangular beam, simply supported at its edges. The energy distribution applied to the collimator was determined with FLUKA, a particle physics Monte Carlo simulation package. The beam energy was introduced to the finite element model in the form of a 3-D table using the HGEN command. Transient thermal simulation was used to calculate the temperature distribution as a function of time. Researchers performed structural dynamic analysis by applying the temperature distribution as nodal loading at different time steps. Elastoplastic analysis used the multilinear kinematic hardening model for metallic components. The team developed special algorithms to apply a temperature distribution changing in time and space at different substeps of the elastic–plastic analysis. The integration time step of 0.1 μs was based on the





Residual displacement based on physical measurements (left) and simulation (right) matched very closely.

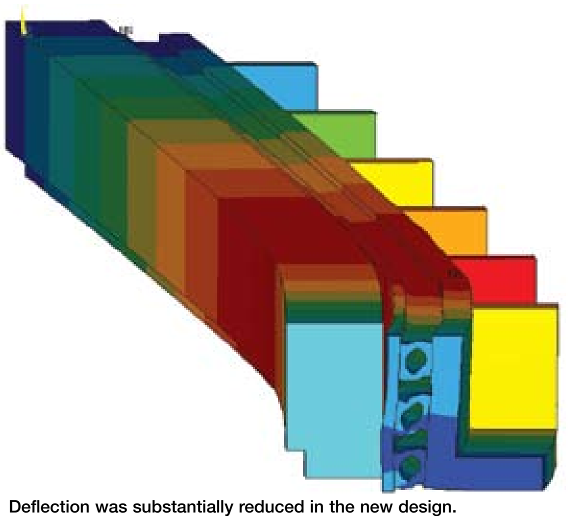


Simulation predictions correlate well with measured deflection.

preliminary analytical estimation to avoid numerical damping. Finally, static analysis evaluated residual plastic deformation.

The simulation results included the temperature rise throughout the structure, making it possible to use simple formulas to predict plasticization. The maximum stress is well above the proportional limit of copper, so the thermal shock generated plastic strains, as theorized by the researchers. The largest residual plastic strains of 0.12 percent were seen in the 3–mm thick copper plate. These strains are eccentric with respect to the neutral axis of the metal support, leading to permanent deflection away from the beam axis.

The simulation results matched the physical measurements remarkably closely. When comparing the physical measurements in the line graph to the simulation predictions in the contour plot, the transverse residual deflection of 350 µm predicted by the simulation closely matched the measured value of 300 µm. The dynamic response predicted by the simulation also correlated well with laser Doppler vibrometer measurements. The simulations are higher in magnitude than the physical measurements because the simulations do not take damping into account.


Deflection was substantially reduced in the new design.

The study team, made up of senior and junior engineers, fellows, and students, used the validated simulation model as the primary tool to solve the problem. CERN researchers evaluated a number of different geometries and materials to determine their impact on the simulation of the support structure. Using these results, they decided to modify the jaw assembly series design by changing the thin plate material from OFE-copper to the higher yield strength Glidcop® material. Glidcop is a family of copper-based metal matrix composite alloys mixed primarily with aluminum oxide ceramic particles. The addition of small amounts of aluminum oxide greatly increases the copper's resistance to thermal softening and enhances elevated temperature strength.

An updated model of the series jaw assembly including Cu-Ni pipes and Glidcop support beam and thin plates showed that deflection was reduced from 300 µm to 16 µm, which was within acceptable limits. This figure was confirmed by experimental tests carried out on a second prototype of the collimator. The use of simulation in this application made it possible to rapidly diagnose the problem and develop an acceptable solution while eliminating the need to build additional prototypes, excepting the final design. ANSYS Mechanical software played a key role by providing the full range of physics capabilities needed to accurately simulate a very complex problem. ■

Designing Batteries for Electric Vehicles

Numerical simulation can be used to accelerate battery development and address safety concerns.

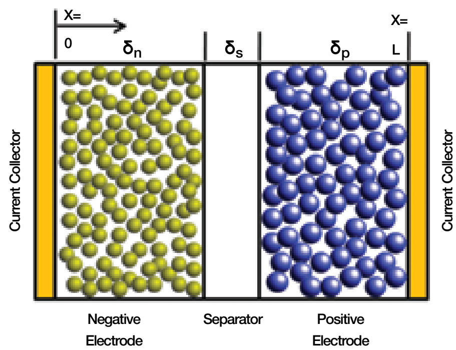
By Xiao Hu, Lead Engineer, ANSYS, Inc.

The lithium-ion battery is a preferred candidate as a source of power for the hybrid electric vehicle (HEV) and electric vehicle (EV) because of high energy density, high voltage, low self-discharge rate and good stability. HEV and EV applications require very large lithium-ion batteries, but, during high power extraction required to drive a vehicle, these large batteries may experience a significant temperature increase — which can lead to safety concerns. A properly designed thermal management system is crucial to prevent overheating and uneven heating across a large battery pack, which can lead to degradation, mismatch in cell capacity and thermal runaway. Design of the thermal management system requires knowledge of the cooling system as well as the amount of heat that will be generated by cells within the battery pack.

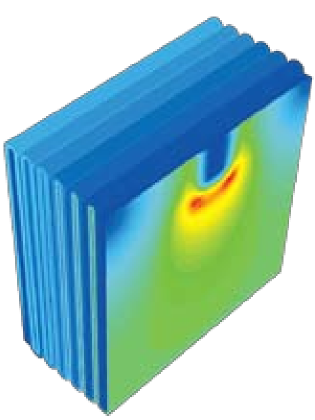
Simulation can assist in thermal design at both the cell level (a single battery cell) and the system level (a battery module or a complete battery pack).

On the battery cell level, the focus is on detailed heat generation and temperature distribution within a battery cell — information mainly used by manufacturers and battery researchers. Experimental data reveals that the rate of heat generation varies substantially with time over the course of charging and discharging. Heat can be generated from internal losses of Joule heating and local electrode overpotentials, the entropy of cell reaction, mixing and side reactions. When only the most important effects of Joule heating and local electrode overpotentials are considered, heat generation can be expressed by open-circuit potential and the potential difference between positive and negative electrodes. By using models to predict potential and current density distribution on the electrodes of a lithium-ion battery as a function of discharge time, the results can be used to calculate the temperature distributions of the lithium-ion battery. These temperature distributions can then be used within ANSYS CFD software to examine the effect of the configuration of the electrodes — the aspect ratio of the electrodes and the placing of current collecting tabs — as well as the discharge rates on the thermal behavior of the battery.

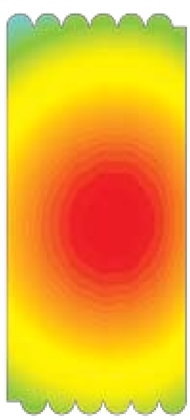
While this type of model provides detailed information about temperature and current density distribution, it requires experimental testing data as input. This model cannot predict the impact of design changes on battery thermal performance without conducting testing again. However, a physics-based electrochemistry model can be used to investigate the impact of battery design parameters on battery performance. The electrochemistry model includes geometry parameters, properties and



Schematic of a lithium-ion cell sandwich consisting of composite negative and positive electrodes and separator

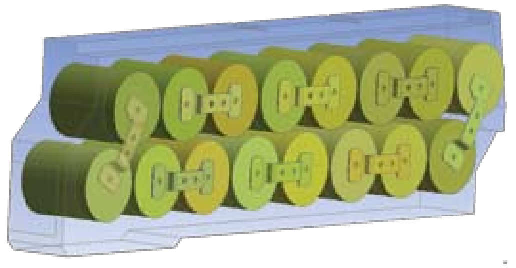


Current density distribution within a battery cell

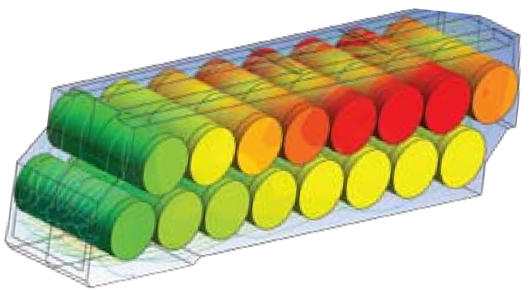


Temperature distribution within a battery cell

Temperature distributions from this fluid dynamics simulation were shown to be in good agreement with those from the experimental measurement.



Automotive battery module with 16 cells



Fluid flow solution for the 16-battery cell module

A complete battery thermal–fluid dynamics analysis including optimization can be done using ANSYS FLUENT software entirely within the ANSYS Workbench environment.

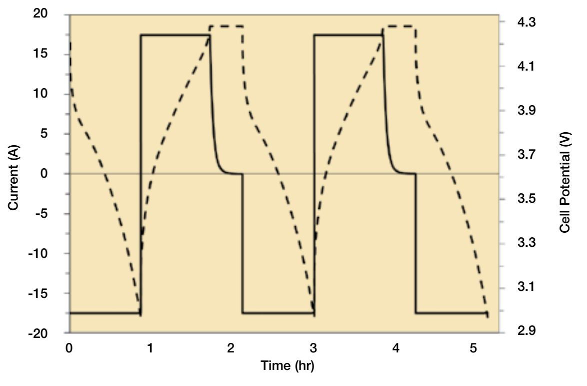
temperature, with the last being the most important. A physics-based model also can provide inputs that would otherwise need experimental testing to obtain. The most famous physics-based model originally was proposed by Professor John Newman from UC Berkeley. Such a model has been implemented in Simplorer software from ANSYS to allow determination of many important factors in battery design.

Charge and discharge cycle results can be plotted along with concentration profiles during discharge. From this information, optimization of the initial composition of electrolytes in the cell can be performed. When this study was performed, results showed that a higher initial concentration of electrolytes leads to as somewhat lower conductivity in the separator but a much higher conductivity in the composite cathode, where this is extremely important. By plotting concentration profiles under different temperatures, battery designers can determine when the limiting current occurs to specify the temperature range that the system needs to maintain to avoid reaching limiting current. In addition, battery runtime is a strong function of temperature, and battery runtime is longer with higher operating temperature. This can be confirmed from a physics-based electrochemistry model, but it presents another problem: Higher temperature brings safety concerns, shortens battery life, and becomes another optimization issue in battery design.

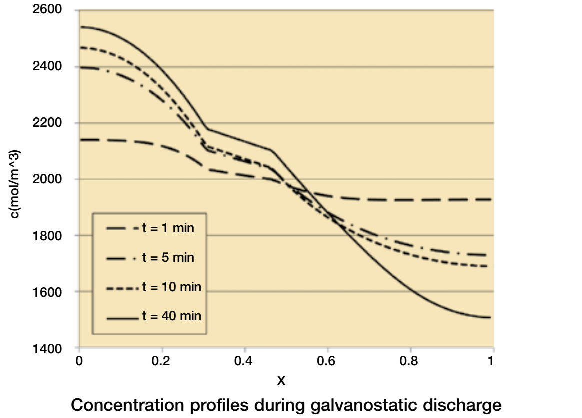
System-level design engineers working at the module or pack level have a different set of requirements. Typically, these engineers cannot afford to simulate as many details as engineers working at the cell level can; they also have a different set of simulation goals. Engineers using computational fluid dynamics (CFD) for battery thermal management are interested in maintaining the desired temperature range, reducing pressure drop and maintaining temperature uniformity. Fluid dynamics has been widely used to predict flow and heat transfer in many industries, and it applies well to battery thermal management. By using the ANSYS Workbench platform with ANSYS CFD software, a complete battery thermal–fluid dynamics analysis, including optimization, can be done entirely within the same environment.

While fluid dynamics simulation can give detailed thermal information about battery thermal management systems, it is time consuming to perform many transient simulations under different drive cycles. Model order reduction techniques exist for extracting a model from CFD results, and the extract model, called Foster network model, gives the same solution as that from the full CFD model — but it runs much faster compared with CFD. The model order reduction process is handled automatically by Simplorer software using CFD results as inputs. This process opens the door for simulations that would otherwise have been impractical, such as battery thermal control system analysis.

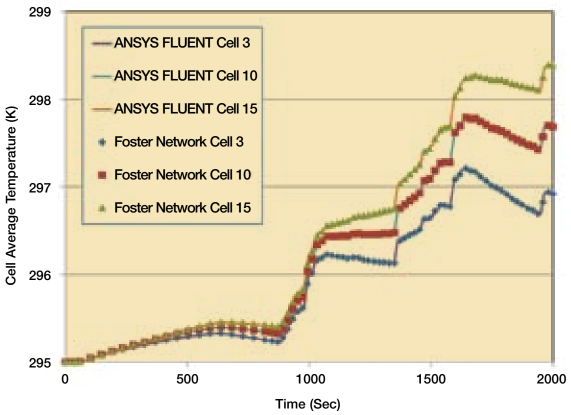
For electrical engineers, the primary concern is the electric performance of the battery rather than the thermal



Battery charge and discharge cycle results from John Newman's electrochemistry model



Concentration profiles during galvanostatic discharge



Comparison of CFD results with Foster network results. In this case, the Foster network model took 20 seconds to run, whereas traditional CFD required a few hours.

performance. An accurate and yet simple-to-use thermal model that couples with a battery electric circuit model is required. This can be accomplished through VHDL-AMS, an IEEE® standard hardware simulation language supported by Simplorer.

Designing batteries for HEV/EV involves many challenges, and designers focusing on different aspects have different requirements. The benefit with ANSYS software is that it offers models ranging from cell-level electrochemistry to system-level thermal management. ■

Reference

Hu, X.; Lin, S.; Stanton, S.; Lian, W. *A Novel Thermal Model for HEV/EV Battery Modeling Based on CFD Calculation*, Proceedings of IEEE Energy Conversion Congress and Expo, Atlanta, U.S.A., September 12–16, **2010**.

Hu, X.; Lin, S.; Stanton, S.; Lian, W. A State Space Thermal Model for HEV/EV Battery Modeling. *SAE*, **2011**, 01-1364.

Fuller, T.F.; Doyle, M.; Newman, J. Simulation and Optimization of the Dual Lithium Ion Insertion Cell. *Journal of Electrochem. Soc.*, **1994**, vol. 141, pp. 1–110.

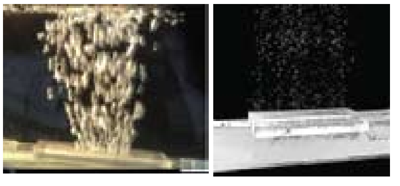
Microbubbles Keep Green Energy Blooming

Algae-derived biofuel production gets a strong pulse from flow simulation.

By William B. Zimmerman, Professor of Chemical and Biological Engineering, University of Sheffield, U.K.

Image © iStockphoto.com/kamil.

The world faces a shortage of fossil fuels; at the same time, efforts are being made to reduce carbon dioxide (CO_2) emissions. Production of biofuels from certain kinds of algae offers a potential solution to both problems. Like other photosynthetic organisms, algae capture CO_2 and sunlight and then convert the elements to oxygen and the energy required for their life cycles. The algae biomass itself includes proteins, carbohydrates and fatty acids. It is these fatty acids — or natural oils — that offer great potential as a renewable feedstock for refineries, where they can be transformed into fuel to power gasoline or diesel engines.

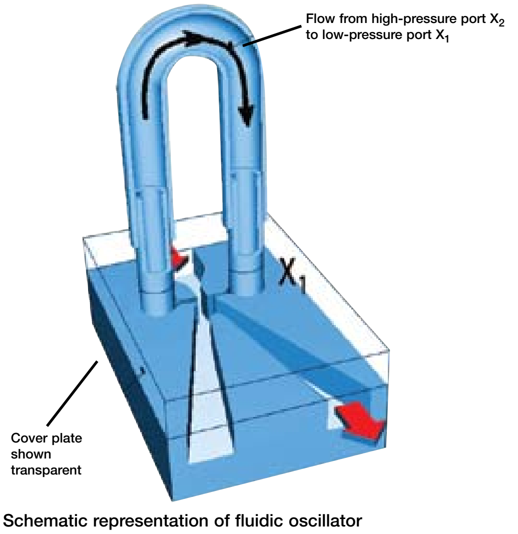


Left: A conventional gas diffuser produces relatively large bubbles. Right: A gas diffuser with the fluid oscillator shows tremendous reduction in bubble size.

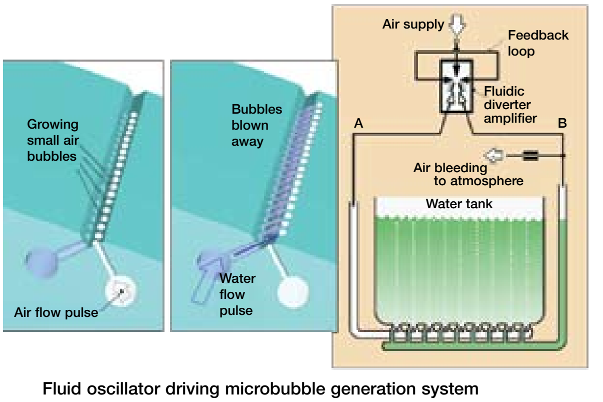
However, most processes to manufacture algae-derived biofuels are too expensive to be commercially viable — since the lowest production prices in the United States are currently in the $20 to $30 per gallon range. Two of the largest costs involved are the capital expenses required to build the bioreactors (producing algae on a large scale) and the energy required to operate them.

To reduce these costs, one of the developing applications being studied is to introduce CO_2-rich fossil fuel exhaust gases into the bioreactor as microbubbles. Among the groups tackling this problem is a research team from the chemical engineering department at the University of Sheffield in the U.K. The group has made a major step forward in reducing the cost of producing algal biofuels from exhaust gases by creating microbubbles that are about 20 microns in diameter. The major advantage of smaller bubbles is that they have a higher surface-area-to-volume ratio. Because these 20 micron microbubbles are 50 times smaller than conventional 1 millimeter fine bubbles, they have a correspondingly higher surface area per unit volume. Thus, they provide 50 times greater mass transfer rates, which could potentially translate into a tenfold higher yield in algal biofuel production [1].

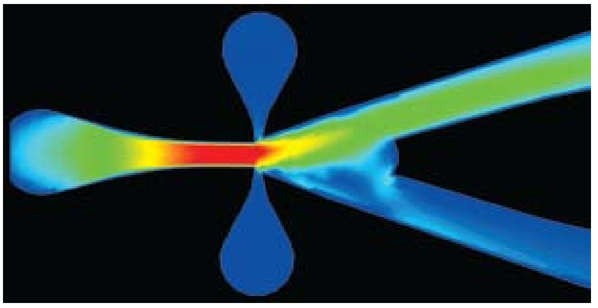
The key to producing bubbles this small is a unique oscillator design developed using ANSYS FLUENT fluid dynamics software (part of a fluid dynamics Academic Product bundle from ANSYS) to evaluate the initial design concept and perform parametric studies to select appropriate nozzle width and flow rates that provide the right operating conditions. The traditional method of producing bubbles for reactors has been to force the gas



Schematic representation of fluidic oscillator



Fluid oscillator driving microbubble generation system



Top view of velocity magnitude contours on midplane of oscillator showing supplied gas flow directed into the upper outlet channel and no flow through the two teardrop-shaped control nozzles

under pressure through a lattice of very small holes. When finally separated from the nozzle, however, the bubble has a diameter that is often many times larger than the hole, because its separation is controlled by the surface tension of the water. The key idea of the Sheffield team's microbubble generation method was to limit bubble growth time using an oscillator to supply gas to the bubble formation holes. Growth is terminated at the end of each pulse of gas. Bubbles are then removed from the nozzle using a pulse of water that alternates with the gas pulses so the growth of new bubbles can begin in the next oscillation period. Using this method, no bubble can reach the size typically formed from steady gas blowing.

In the oscillator design [2,3], steady gas flow is supplied to the oscillator inlet. The main flow stream then exits one of the two outlet channels depending on the action applied by flow through a bidirectional control loop. The flow exits through the upper outlet channel, causing a decrease in pressure at the upper control nozzle. The upper control nozzle then draws gas through the feedback loop from the lower control nozzle, where the pressure is higher. After the flow in the feedback loop tube gains momentum, the flow exiting the upper control nozzle causes the main jet — because of inducement (or amplification) effects — to divert to the lower outlet channel. After a delay needed for the feedback flow to gain momentum in the opposite direction, the main jet is diverted back to the upper outlet, resulting in a periodic switching process. The frequency of the oscillation is controlled primarily by the length of the feedback loop and the gas supply flow rate.

Sheffield's team performed simulation of an early oscillator prototype with ANSYS tools. Researchers meshed the 3-D geometry with 70,000 tetrahedral cells and used the renormalization group (RNG) k-ε model to handle turbulence at lower Reynolds numbers. The results matched the performance parameters of the early prototype, so the team proceeded to use fluid flow simulation to improve the device's controllability, including virtual testing of geometries with different control nozzle widths and different input flow boundary conditions. The researchers compared the results for the different design alternatives and then selected the final proof-of-concept design.

With the noted 50-fold increase in mass transfer rate afforded by this oscillator design, CO_2 dispersal is accelerated in the bioreactor and should enhance algae growth rate by a factor of 10. The microbubbles more efficiently strip the oxygen that algae emit during photosynthesis, thus permitting much higher algae densities in the water. These improvements should substantially reduce the capital and operating expense required for a given volume of biofuel production, as more algae can be grown more quickly in a smaller amount of space.

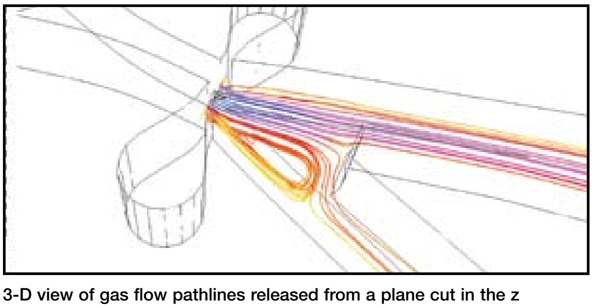
Beyond displaying a supercharged growth rate, the experiments showed an 18 percent reduction in the energy required for bubble production compared to conventional fine bubbles. Researchers attribute this, in part, to reduced friction losses due to the oscillatory flow. The team thus predicts that engineering simulation will play an even greater role in the coming design of a commercial-scale bioreactor, when it will become critical to optimize all components of the design to minimize capital expenses. ■

References

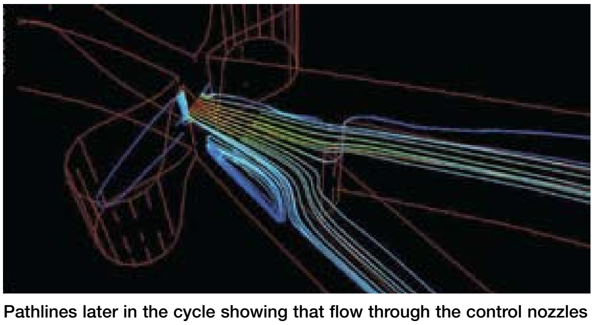
[1] Zimmerman, W.B.; Zandi, M.; Bandalusena, H.C.H.; Tesař, V.; Gilmour, D.J.; Ying, K. Pilot Scales Studies of Microbubble Mediated Airlift Loop Bioreactor Growth of Microalgae Dunaliella Salina. *Applied Energy*, submitted in 2010.

[2] Tesař, V.; Bandalusena, H.C.H. Bistable Diverter Valve in Microfluidics. *Experiments in Fluids*, **2010**; doi:10.1007/s00348-010-0983-0.

[3] Zimmerman, W.B.; Hewakandamby, B.N.; Tesař, V.; Bandulasena, H.C.H.; Omotowa, O.A. On the Design and Simulation of an Airlift Loop Bioreactor with Microbubble Generation by Fluidic Oscillation. *Food and Bioproducts Processing*, **2009**; Vol. 97, No. 3, pp. 215–227.



3-D view of gas flow pathlines released from a plane cut in the z direction and colored by velocity magnitude. The cusped splinter nose at the junction of the outlet channels generates an internal feedback loop that initially helps to direct the main flow to the upper outlet.



Pathlines later in the cycle showing that flow through the control nozzles has overcome internal feedback and the main jet is being switched to the lower outlet channel

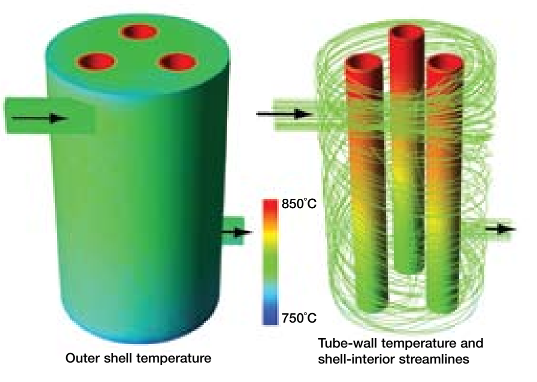
Reforming a Fuel Cell Modeling Process

Coupling flow simulation with complex chemistry tools brings a united front to analyzing leading-edge energy systems.

By Robert J. Kee, George R. Brown Distinguished Professor of Engineering, Colorado School of Mines, Golden, U.S.A.

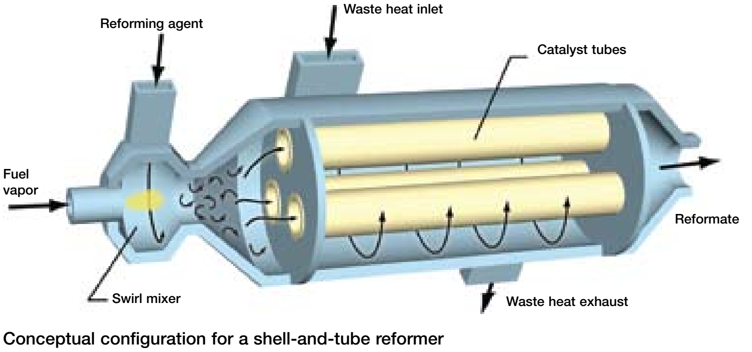
Since its basic principle was first demonstrated in the early nineteenth century, fuel cell technology has evolved into many different variations. The underlying mechanism common to all fuel cells is conversion of chemical energy into electricity by means of reforming the fuel into hydrogen along with the subsequent electrochemical oxidation of hydrogen into water. Depending on the type of fuel cell and its application, the fuel can be lighter hydrocarbons — such as natural gas, propane or methanol — or heavier liquids, like diesel or jet fuel. The key advantages of fuel cells over systems that burn fossil fuels include fewer moving parts and overall reduced pollutant emissions. Some of the challenges in developing fuel cells for more widespread use are the high cost of catalyst or other fabrication materials, the difficulty of hydrogen storage, and very complex chemistry.

Solid oxide fuel cells (SOFCs) in particular have been the subject of much research in recent decades: They have the ability to reform many different fuels, and their high operating temperatures offer the side benefit of using the exhausted heat energy. With support from the U.S. Office of Naval Research, a team from the Colorado School of Mines (CSM) has been working with ANSYS simulation tools to model the chemistry, electrochemistry and fluid mechanics of an SOFC stack system. Such a system — for example, an auxiliary power unit (APU) used by a Navy vessel — commonly comprises a shell-and-tube design that includes internal or external reformers, depending on the fuel. The endothermic steam-reforming operation is supported by circulating the exhausted heat energy from the exothermic electrochemical oxidation within the fuel cell. The complex catalytic chemistry is confined within the tubes, while the three-dimensional fluid mechanics of the air flow surrounding the tubes is complex but does not involve chemical complexity.

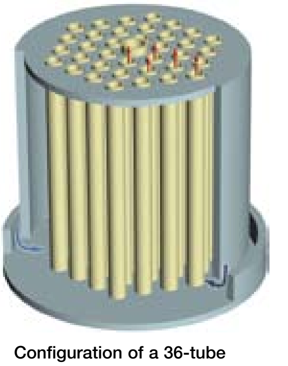


Results from the simulation of a shell-and-tube reformer simulation for the catalytic partial oxidation of propane in which an ANSYS FLUENT model of three-dimensional fluid flow and heat transfer is coupled with a CHEMKIN-based plug-flow model. Shown are the temperatures on the outside of containment shell (left) and catalyst tubes (right).

To evaluate the full anode-supported SOFC stack configuration — in which the anode side is the tube side and the cathode side is the shell side — the CSM team needed to couple the complex chemistry with the three-dimensional fluid mechanics. On the shell side, researchers considered the fluid flow



Conceptual configuration for a shell-and-tube reformer



Configuration of a 36-tube anode-supported SOFC stack

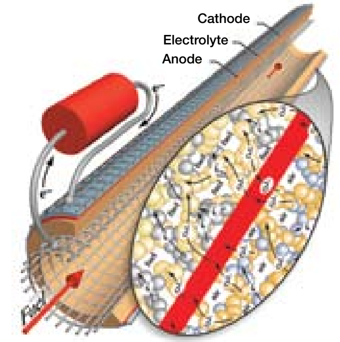
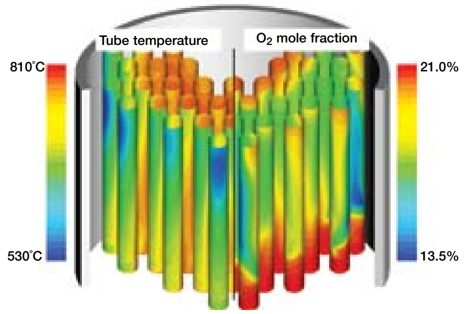


Illustration of a single anode-supported SOFC tube. Electric current is generated on the inside of the tubes (anode) and is discharged on the tube exterior (cathode). The balloon shows the essential microscale electrochemical phenomena in the porous composite electrode layers.



Results from modeling a 66-tube SOFC stack operating on a mixture of hydrogen, carbon monoxide and methane, which are produced by reforming hexadecane. The right-side tubes show gas-phase oxygen mole fraction on the tube surfaces. The left-side tubes show tube surface temperatures.

and heat transfer, including thermal radiation among all tubes and the containment shell. For this task, the team chose ANSYS FLUENT software (a component of the ANSYS Academic Research CFD product bundle) to model the complex, but nonreactive, fluid mechanics.

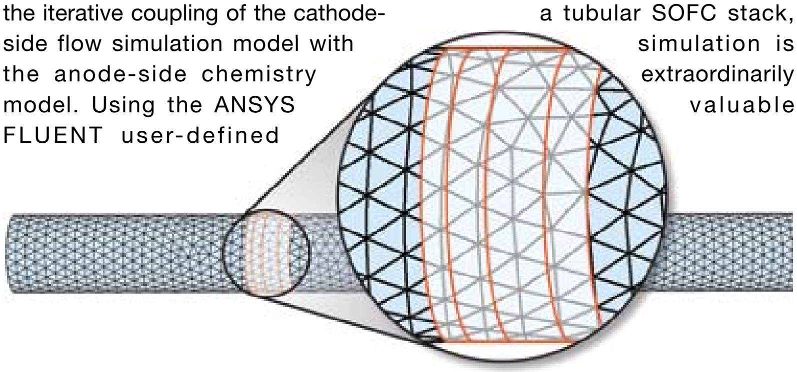
On the tube interior (anode) side, the simulation of chemical kinetics for reforming practical military logistics fuels — diesel in the case of an APU — demands hundreds of surface reactions and thousands of gas-phase reactions. One-dimensional chemistry tools such as CHEMKIN™ or CANTERA can handle the reaction kinetics and charge transfer, as long as the fluid mechanics can be modeled simply. In this case, the fuel reforming chemistry and charge-transfer electrochemistry inside the tubes are all complex, but the fuel flow is indeed simple enough and can be modeled in a one-dimensional tool as a plug flow.

The next aspect of the overall simulation process was coordinating the iterative coupling of the cathode-side flow simulation model with the anode-side chemistry model. Using the ANSYS FLUENT user-defined function (UDF) capability, the research team automated the process of averaging three-dimensional temperature and oxygen mole fraction data from the shell-side simulation and mapping it onto the tube-side band mesh. Additionally, the UDF directed the tube-side chemistry model to supply heat flux and oxygen mass flux boundary conditions from each of the tubes back to the shell-side fluid flow model.

Results of the simulations revealed that there can be significant temperature variations between different tubes. The tubes were generally cooler at the bottom, which was caused by a combination of internal fuel reforming and heat transfer to the shell-side air. Because the outer tubes acted as radiation shields, the inner tubes generally operated at higher temperatures. Shell-side air was introduced from below and exhausted at the top, and, therefore, the oxygen decreased from bottom to top.

When designing and optimizing a tubular SOFC stack, simulation is extraordinarily valuable because it is important for all tubes to deliver similar performance. The coupled model is useful for investigating the effects of design considerations, such as tube packing and air-flow alternatives, on the overall performance of the stack. However, since the tube interior geometries were not fully resolved in this research due to their complex microstructure, extensions to the work under consideration include performing detailed three-dimensional ANSYS FLUENT simulations of the charge transport through the porous anode material on representative tube volume sections to calculate the effective electrical conductivity. This microscale-effective conductivity could then be used as an input for the full one-dimensional tube-side simulation to further improve transport modeling inside the tubes, thus enabling the beginnings of a multiscale analysis.

The capability to couple complex flow and heat transfer using flow simulation with complex chemistry and electrochemistry is a powerful new tool for certain classes of reacting-flow problems. The CSM team developed the ANSYS FLUENT UDFs to be sufficiently general so that a range of one-dimensional chemistry tools could be incorporated. This approach to modeling tubular configurations can be useful beyond fuel cells, as it is directly applicable to geometrically related layouts, such as battery packs, nuclear fuel rods or cracking furnaces. ■



A three-dimensional ANSYS FLUENT face mesh on an SOFC tube with an overlying one-dimensional band mesh

Accelerating CFD Solutions

Several recent enhancements in ANSYS FLUENT solver capabilities accelerate convergence and reduce solution time.

By Mark Keating, Principal Engineer, ANSYS, Inc.

Many solver performance enhancements have been introduced to ANSYS FLUENT fluid dynamics software over the past few releases. These capabilities can dramatically improve the speed and reliability of simulation. Running the solver out of the box does not always guarantee optimum solver settings for any particular application. So by understanding and using solver technology appropriately, a user can obtain faster results and better convergence.

Pressure-Based Coupled Solver

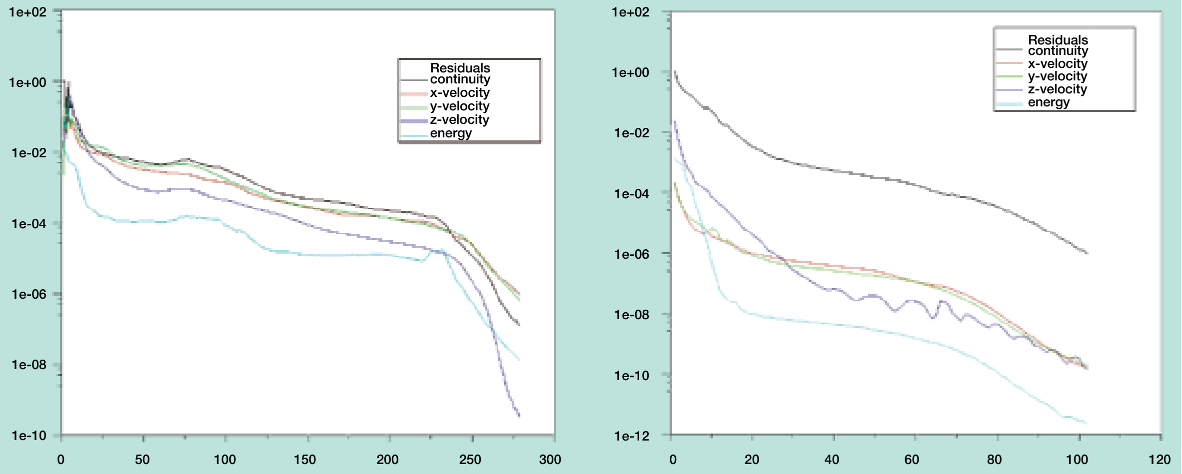
The pressure-based coupled solver (PBCS) was introduced in 2006, and its usage is growing. This solver reduces the time to overall convergence, by as much as five times, by solving momentum and pressure-based continuity equations in a coupled manner. Though there is a slight increase in associated memory requirements for using this solver, its benefits far outweigh the drawbacks. The PBCS is becoming the solver of choice for subsonic applications. When using it, re-ordering the grid is always advisable. The default explicit under-relaxation factors (URFs) for pressure and velocity of 0.75 are generally robust values, but they should be reduced for skewed meshes, when oscillatory convergence is experienced, or when higher-order discretization is employed to about 0.4 to 0.5. Taking the turbulence URF up to 0.95 to 0.99 can help to accelerate viscous cases. To access the PBCS, change the p–v coupling in the drop-down list from SIMPLE to Coupled.

Pseudo-Transient Method

The pseudo-transient solution method, introduced in version 13.0, is a form of implicit under-relaxation for steady-state cases. It allows users to obtain solutions faster and more robustly than previous versions of ANSYS FLUENT software, especially for highly anisotropic meshes, when using PBCS and density-based (DBNS) implicit solvers. This method uses a pseudo-transient time-stepping approach. In general, the time per iteration is slightly higher, but in some extreme cases the number of iterations required for convergence using this method has dropped by an order of magnitude or more. Usually, overall speedups of 30 percent to 50 percent can be expected. Cases with multiple reference frame (MRF) zones should benefit from using this method. The table shows the levels of improvement possible.

Cases	Courant number-based coupled (iterations)	Pseudo-transient coupled (iterations)
Backward facing step (turbulent: SST)	750	75
Film cooling benchmark (turbulent: SA)	2,300	1,350
Flat plate, SST transition model	1,200	100
Rotor/stator with mixing plane model	500	250
Centrifugal pump	220	50
Axial compressor stage	400	110

Solver speedups achieved using the pseudo-transient coupled solver in ANSYS FLUENT 13.0



Standard Initialization, 279 Iterations

Hybrid Initialization, 102 Iterations

Residual plots for a heat exchanger case show the effect on iteration count of hybrid initialization. This case does not converge with default under-relaxation and standard initialization, but it converges well with the default under-relaxation factors. It is more than two times faster when hybrid initialization is used.

Initialization Methods

Providing an initial data field that is close to the final solution for steady-state cases means the solver has to do less work to reach the converged result. Therefore, this reduces simulation time. Typically, many users employ standard initialization; some use patching for localized control, especially for moving domains or multiphase analyses. Interpolation files are used to initialize cases, and the interpolation workflows available in ANSYS FLUENT have been improved in recent releases. There are also other initialization techniques for further accelerating the simulation convergence.

Full multigrid initialization (FMG), introduced in 2006, provides the initial and approximate solution at a minimum cost to overall computational expense. The feature is accessed via the text user interface (TUI) and re-orders the grid as part of the process. The commands are:

```
Solve>initialize>set-fmg-initialization
Solve>initialize>fmg-i
```

The overall initialization time using this approach is much longer than that using standard initialization by zone, but it allows a much quicker solve. FMG solves Euler equations and is available for single-phase flows only. FMG initialization provides the best-guess initial solution. It is particularly suited to turbomachinery as well as external and compressible flow problems.

Hybrid solution initialization was introduced at version 13.0 and uses a collection of recipes and boundary interpolation methods to efficiently initialize the solution based purely on simulation setup — so the user does not need to provide additional inputs for initialization. The method can be applied to flows ranging from subsonic to supersonic. It is the recommended method when using PBCS and DBNS for steady-state cases in ANSYS FLUENT 13.0. This initialization may improve the convergence robustness for many cases. Unlike FMG, this initialization method can be used for multiphase flows.

Summary

A number of solver settings are available to aid solution acceleration and convergence within the ANSYS FLUENT solver. Individually, these techniques can be used to reduce solution times; combined, they offer even greater capability. For example, in some cases, moving from a segregated solver to PBCS with FMG and the pseudo-transient method has resulted in up to 100 times speedup. These features show the benefits of investigating and taking advantage of new solver technologies. ■

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

Send address corrections to AdvantageAddressChange@ansys.com.