ANSYS®

Excellence in Engineering Simulation

ADVANTAGE

ISSUE 3 | 2016

Breakthrough Energy Innovation

Turning a New Leaf
Tree-like wind power generation

Power Tool for Chips
Reducing chip power consumption

Big Wheel
Wheels improve truck fuel economy

BREAKTHROUGH ENERGY INNOVATION: **SEIZING THE OPPORTUNITY**

By viewing energy efficiency not as a requirement — but as a strategic opportunity — engineers have a chance to shine.

By **Rob Harwood**,
Director,
Industry Marketing,
ANSYS

Historically, many companies focused on waste reduction, "green" product development, recycling and other environmental initiatives because they were required to. Sustainability became a corporate priority based on external drivers such as ever-tightening regulations, shareholders' demand for lower operational costs, and customer concerns about the environmental footprint of the products they were buying.

But today sustainability has taken on a much larger, more strategic role in most businesses. According to McKinsey's Global CEO Survey, executives today value sustainability because they view it as closely aligned with their company's core goals, mission or values. They recognize that sustainability can actually drive revenue growth.

Energy is at the heart of corporate sustainability initiatives and, by its very nature, impacts all industry sectors. Much attention continues to be focused on how energy-efficiently products are manufactured — as well as how energy-efficiently they perform in the field. Traditional questions such as "How can we make our products consume less energy to save customers money, and how can we reduce the number of components and the amount of materials to improve margins?" continue to be important.

Today, however, the world's leading engineering teams are being challenged to take their energy focus a step further by asking "How can we completely reinvent our products to differentiate ourselves — and dramatically increase sales — via energy-related innovation?" As competition increases and consumers become more demanding, it's increasingly clear that "more of the same" is not enough to drive increased revenues. Incremental product improvements will fall short in meeting both external market needs and internal business goals. Only breakthrough energy innovation will deliver the technological disruption required to change the game and seize market leadership.

> "Energy is at the heart of corporate sustainability initiatives and, by its very nature, impacts all industry sectors."

Product development teams are responding to this challenge in two ways. First, they are adding significant complexity to their existing, already highly tuned products. Second, they are designing disruptive new products that replace long-standing incumbents by operating outside historical design paradigms and constraints. In both cases, engineering teams face the same problem. The design variables, interdependencies and unexplored trade space have suddenly become substantially larger, with little experience as a guide. "We just haven't done this before" is becoming a new engineering norm.

In this environment of exploration and discovery, engineering simulation is an essential tool. By working in a risk-free virtual design environment, product developers can quickly explore design trade-offs and arrive at optimal decisions that balance energy improvements with reduced costs and increased performance.

For example, when designing products that require energy to move, engineers can predict the impact of weight reduction on their products' structural strength and durability over time. They can arrive at revolutionary designs that deliver breakthrough energy innovation — without investing energy, materials, time and money in prototypes and physical testing.

This issue of *ANSYS Advantage* highlights many of the ways our customers are delivering breakthrough energy innovation by leveraging the power of engineering simulation in five core application areas: advanced electrification, machine and fuel efficiency, aerodynamic design, effective lightweighting and thermal optimization.

We hope the articles in this issue shine a light on the pivotal role engineers and engineering simulation are playing in producing breakthrough energy innovation. We also hope these stories will energize and inspire your own innovative efforts to seize the energy opportunity, whatever industry you are in.

Breakthrough Energy Innovation
ansys.com/breakthrough

Table of Contents

Breakthrough Energy Innovation

ABOUT THE COVER

Innovation is driving breakthroughs in many aspects of the energy industry. New Wind has developed a tree-like wind power generator that enhances the urban landscape.

Join the simulation conversation
ansys.com/Social@ANSYS

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

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

Executive & Managing Editor
Chris Reeves

Editorial Advisers
Amy Pietzak
Tom Smithyman

Editorial Contributor
ANSYS Customer Excellence
North America

Senior Editor
Tim Palucka

Editors
Erik Ferguson
Kara Gremillion
Thomas Matich
Mark Ravenstahl
Ravi Ravikumar
Walter Scott
Anatole Wilson

Art Directors
Ron Santillo
Dan Hart

Designer
Dan Hart

Cover Illustration
Ron Santillo

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
U.S.A.

Subscribe at ansys.com/magazine

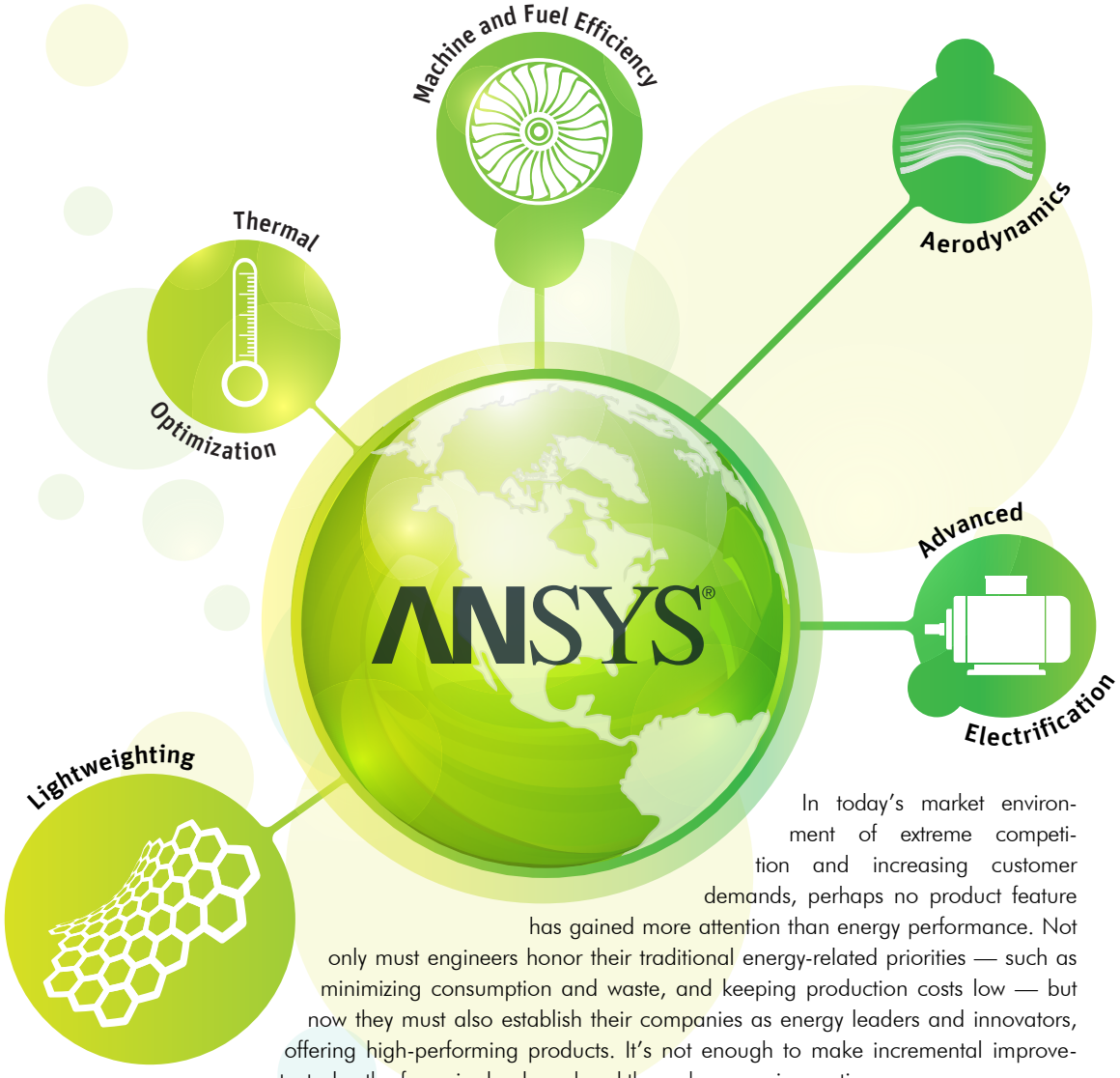
Realize Your Product Promise®

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge, or put on wearable technology, chances are you've used a product where ANSYS software played a critical role in its creation.

ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

Making a Breakthrough

Engineering simulation helps product development teams around the world create energy-specific innovation in five key areas.



In today's market environment of extreme competition and increasing customer demands, perhaps no product feature has gained more attention than energy performance. Not only must engineers honor their traditional energy-related priorities — such as minimizing consumption and waste, and keeping production costs low — but now they must also establish their companies as energy leaders and innovators, offering high-performing products. It's not enough to make incremental improvements; today the focus is clearly on breakthrough energy innovation.

Because simulation enables product development teams to make engineering decisions quickly and confidently, discarding some ideas and embracing others, it has assumed greater importance when pushing product designs beyond incremental improvements. While simulation is being widely applied to study a diverse range of energy-related performance issues, ANSYS has identified five applications that are especially critical as companies worldwide pursue breakthrough energy innovation: advanced electrification, machine and fuel efficiency, aerodynamic design, effective lightweighting and thermal optimization.

By **Todd McDevitt,** Marketing Director, ANSYS

In each of these areas, ANSYS customers are leveraging the power of engineering simulation to break new ground in energy performance. Their success should inspire other companies to ask and answer challenging product development questions via simulation — leading to performance enhancements that help redefine existing product categories and create entirely new ones.

ADVANCED ELECTRIFICATION

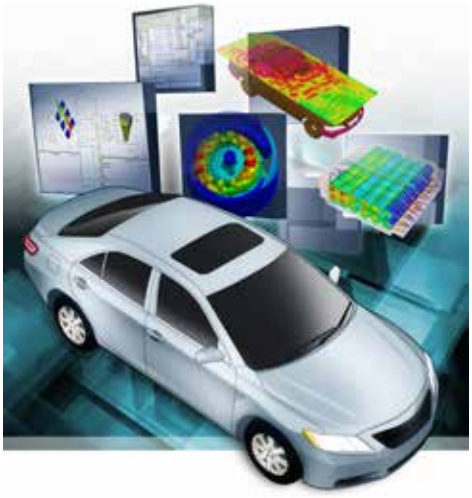
Hybrid and electric vehicles have been making headlines for years, but the automotive industry is not the only source of electrification innovation. In many industries, electrically driven products are replacing mechanical power systems because they offer a number of performance advantages, including lighter weight, smaller footprints and lower maintenance.

How big is the opportunity for aggregate energy savings? Consider this: In the United States alone, more than 40 million industrial machines convert electricity into work for manufacturing operations. Over a motor's lifetime, more than 97 percent of its total cost is accounted for by its power consumption; the remaining 3 percent of its cost represents the capital investment required to develop and manufacture the motor.

While it may be initially time- and cost-intensive for companies to examine their core mechanical systems and replace them with innovative electric systems, the long-term payoff in better conversion rates and energy savings should justify this investment. This is the reason so many engineering teams are currently focused on advanced electrification initiatives.

Because electrification programs require trade-offs on many design variables, and the simulations can be large and complex, ANSYS has worked to accelerate solution run times and streamline parametric analysis. For example, many customers benefit from high-performance computing (HPC) solutions for the ANSYS electromagnetic product suite. WEG Industries — one of the world's largest manufacturers of electric motors — applied ANSYS Maxwell in an HPC environment to study and improve energy efficiency. By leveraging HPC-enabled electromagnetic simulations, WEG's engineering team has decreased its computation times by a factor of 70 over nonsimulation design methods.

But HPC compatibility is not the only way ANSYS is increasing its support for electrification initiatives. On page 30, learn how AMD is using ANSYS PowerArtist to identify and eliminate areas of power waste within its chip designs.



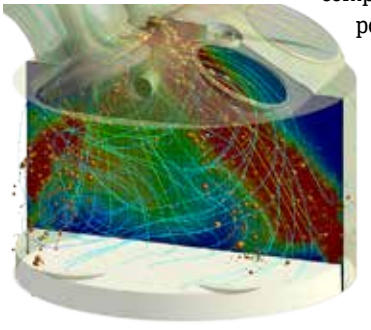
ANSYS simulation solutions enable automotive engineers to evaluate sensors, actuators, motors and other components interacting with electronic circuits and control systems.

MACHINE AND FUEL EFFICIENCY

Planes, cars, power plants, production facilities — all depend on engines or generators that must work optimally to maximize energy efficiency. This is a complex task for engineers, because it's never enough to focus on one component. Instead, designers must look at how each component operates in conjunction with every other component in the machine system, under a diverse set of operating parameters. For instance, pumps, motors and loads must be exactly matched under startup, operating and peak conditions.

This system-level engineering approach means making complicated trade-offs. Higher firing temperatures might increase fuel efficiency, but also result in higher emissions, faster materials degradation or other negative performance aspects. As product development teams work to balance machinery energy efficiency with concerns about durability, safety, reliability and cost, engineering simulation offers an environment in which intelligent solutions can be achieved quickly. Product developers at Magneti Marelli Powertrain applied simulation to understand the trade-offs involved in adding a turbocharger to improve fuel economy and reduce emissions. Because turbochargers add heat, the team designed an intercooler system that reduced outlet temperatures by 8 percent and improved overall fuel economy by 5 percent. Simulation accelerated the development process for the new intercooler by reducing prototype iterations and allowing any design issues to be addressed at the earliest possible stage.

This issue of *ANSYS Advantage* highlights how BorgWarner, Tecumseh and Aditya Birla Science & Technology have also used ANSYS software to solve energy challenges related to machine and fuel efficiency.



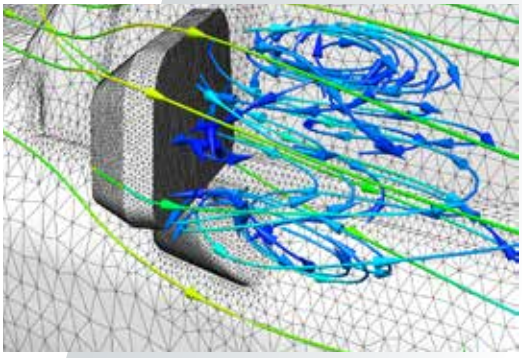
ANSYS Forte robustly and accurately simulates IC engine combustion performance with nearly any fuel, helping engineers rapidly design cleaner-burning, high-efficiency, fuel-flexible engines.

AERODYNAMIC DESIGN

Reducing aerodynamic drag has proven critical to increasing the energy efficiency of planes and cars. In fact, drag is responsible for 22 percent of the fuel consumed by a typical highway truck. Not only can aerodynamic improvements significantly decrease fuel costs and protect razor-thin profit margins, they can also help commercial airlines and trucking companies meet new fuel economy standards and environmental regulations. However, physical aerodynamics testing, which typically involves large wind tunnels and complex instrumentation, is expensive and time-consuming.

When transportation companies require an accurate prediction of vehicle aerodynamics, they are increasingly relying on engineering simulation. The American Institute of Aeronautics and Astronautics depends on computational fluid dynamics (CFD) simulations to produce industry benchmarks and help aerospace companies meet industry performance requirements. At Piaggio Aero Industries, engineers have cut the time involved in evaluating new wing designs by more than 90 percent using ANSYS CFD and ANSYS DesignXplorer.

New Wind is maximizing the efficiency of its wind generators via simulation. (See page 8.) On page 22, an article describes how Emirates Team New Zealand is leveraging CFD simulation to design a new, more aerodynamic sail design that's set to revolutionize America's Cup yacht racing.



Automotive mirror aerodynamics

EFFECTIVE LIGHTWEIGHTING

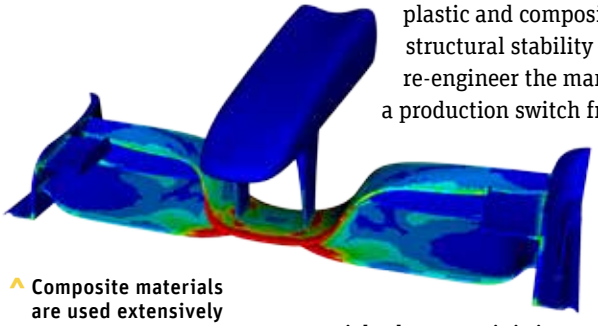
In the transportation sector, few engineering issues receive as much attention as lightweighting, which aims to increase energy efficiency by reducing vehicle weight. This can be accomplished in two ways, each of which presents significant design trade-offs.

First, product development teams can replace traditional materials, such as steel and aluminum, with lighter-weight alternatives such as plastic and composites. In doing so, engineers need to ensure that the long-term structural stability and durability of components remain high. They also need to re-engineer the manufacturing process for these components, since there will be a production switch from, for example, forging to injection molding. In the case of composites, engineers must also design the composite layup in an optimal manner, while guarding against special material concerns, including curing, springback and residual stress.

A second method to achieve lightweighting is by reshaping parts to minimize material. By changing the width, thickness and size of parts, engineers can significantly reduce overall weight; however, it is imperative that product functionality is not adversely affected.

Simulation offers a rapid, cost-effective way for engineers to analyze the effects of new materials and new part geometries before moving on to costly physical prototypes. In the virtual world, design teams can consider the weight benefits of a new design and determine if that lower weight is offset by any performance issues — arriving at an optimal, lightweight solution. As one example, KTM Technologies, a consulting firm specializing in composite engineering, used simulation to design a sports car shell that is 20 percent lighter, without sacrificing strength or stability.

On page 39, an article showcases the lightweighting initiative at Accuride, which has resulted in a 23 percent weight reduction in the company's wide-base wheels for commercial trucks.



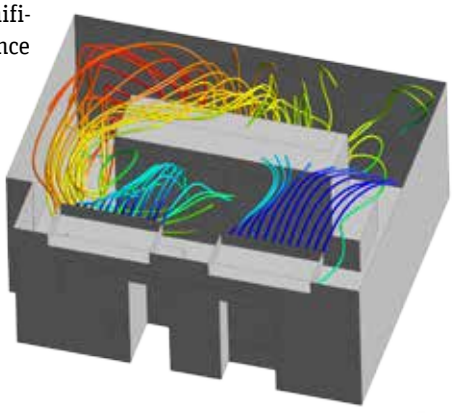
Composite materials are used extensively in motorsport because of their lightness and strength.

THERMAL OPTIMIZATION

Thermal optimization helps many products perform at their best and safest. The issue of overheating inside consumer electronics products has received a lot of media attention because of numerous adverse incidents. In the manufacturing sector, 36 percent of total energy consumption is spent on heating applications. Optimizing industrial heating operations can have a significant impact on a company's bottom line.

While these examples are diverse, thermal optimization of all these products can be achieved via engineering simulation, which predicts not only internal temperatures but also reveals the physical phenomena behind the results. Simulation provides an early-stage method to identify and address any thermal issues before individual components are brought together into a cohesive system. Product development teams can save significant time and costs, while also ensuring the safe performance of their products under real-world operating conditions.

Thermal optimization is a special concern for companies that manufacture heating and cooling products, making simulation a tool with high strategic value. At Whirlpool Brazil, which designs gas burners for free-standing ranges, built-in ovens and cooktops, engineers apply ANSYS software to predict the heating properties of their product designs before the prototype phase. By minimizing rework and accelerating design iterations, the team has reduced overall development time by 30 to 40 percent.



Simulating air flow in any room, from a data center to an auditorium, helps engineers reduce heating and cooling energy and expense.

WHAT SETS THE LEADERS APART?

Across these five application areas, we've observed some key characteristics of those engineering teams that are leading in energy innovation. These findings have been verified by third-party research, including a study conducted by Aberdeen earlier this year.

First, innovation leaders invest in advanced technology. Not only do they leverage the comprehensive ANSYS simulation technology platform, but they also use additive manufacturing, big data analytics and other leading-edge solutions to advance their efforts. They recognize that technology leadership leads to market leadership.

Second, they take a systems view. In today's world of smarter and more sophisticated products, with complex energy demands, it is not enough to look at each component in isolation. Instead, product development teams must look at the implications for the entire system, making informed trade-offs that balance energy improvements with other performance characteristics.

Finally, energy leaders take a bold view of the future. They're not focusing on "me too" product design tweaks, but on radical innovations that have the potential to change the category. They view the current focus on energy as a chance to differentiate their business with a dramatic reimagination of how energy is used, generated or stored by their product solutions.

At ANSYS, we're honored to be working collaboratively with so many of the world's energy leaders to arrive at next-generation solutions. The innovators profiled in this issue of *ANSYS Advantage* — and dozens of other ANSYS customers — are changing the way the world thinks about and uses energy. Their creativity and engineering diligence will deliver benefits for years to come.

Turning a New Leaf

By designing a tree-like wind power generator with steel branches and plastic leaves, New Wind has created an aesthetically pleasing alternative energy source for urban environments, bringing energy generation closer to people.

By **Julia Revuz**,
Former Engineer in Aerodynamics Studies,
New Wind, Paris, France

"Engineers were able to perform *fluid–structure* interaction *multiphysics simulations* to develop a robust, efficient *plastic leaf* for the wind tree."

While walking through the Jardin du Luxembourg, a beautiful garden square in Paris, Jérôme Michaud-Larivière noticed that the leaves on the trees were trembling though the air was calm. Michaud-Larivière knew that movement means energy, and if harnessed, it could be used to generate electricity. What if, he thought, you could build a wind-based energy generator on a human scale, closer to people in urban environments, something that would blend in aesthetically with the garden scenery, something like — a tree?

As a result, he started a company called New Wind that has designed l'Arbre à Vent® (the wind tree) — a 10-meter-high, 7.5-meter-wide tree-shaped wind turbine with a steel trunk and branches and 63 plastic "leaves" (called Aeroleafs®). These leaves capture the wind and transfer its energy through a generator and microcontroller at the base of each leaf, with each tree capable of generating 3 kW of instantaneous power, or about 1900 kWh per year.

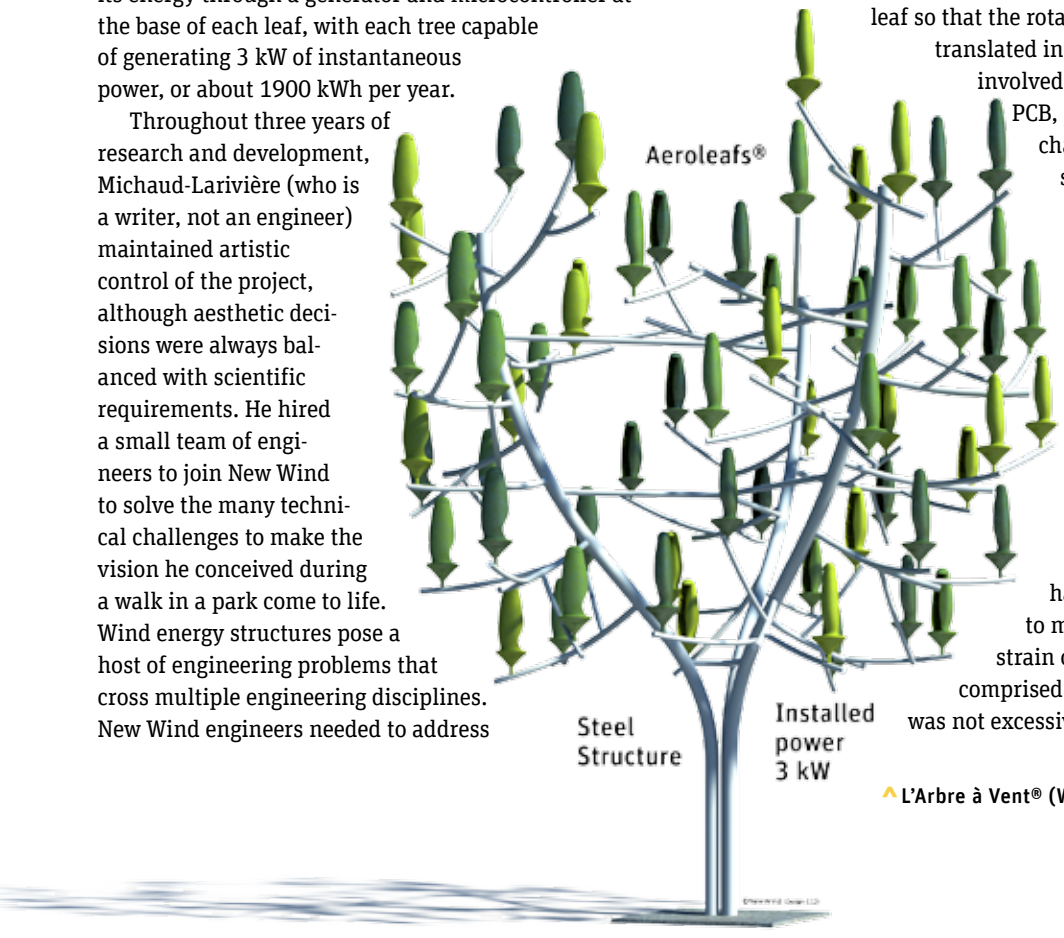
Throughout three years of research and development, Michaud-Larivière (who is a writer, not an engineer) maintained artistic control of the project, although aesthetic decisions were always balanced with scientific requirements. He hired a small team of engineers to join New Wind to solve the many technical challenges to make the vision he conceived during a walk in a park come to life. Wind energy structures pose a host of engineering problems that cross multiple engineering disciplines. New Wind engineers needed to address issues such as wind loading, durability, electronic generators and controls. ANSYS electronic, fluid and structural design software solutions played a large role in overcoming these multiphysics challenges.

DESIGNING A WIND TREE

There were many engineering challenges. Each artificial leaf had to capture the available wind with maximum efficiency, so their physical shape and size were critical. Determining the optimal number of leaves on a tree of the desired size was also important, along with their placement in the 3-D volume of the tree. How do you locate them so that one leaf doesn't interfere with the ability of its nearest neighbor to capture its share of wind, which could be blowing from any direction?

Designing a small power plant into the base of each leaf so that the rotation of the leaf could be translated into electricity on the spot involved a set of magnets and a PCB, presenting electromagnetic challenges. Also, the optimum speed of rotation of the leaf had to be controlled to maximize power generation efficiency.

Structurally, the forces on each plastic leaf in high winds had to be calculated to ensure that the resulting strain would not cause the leaf to fail. The combined stresses of the wind on the leaves had to be taken into account to make sure that the overall strain on the structural steel that comprised the trunk and branches was not excessive.

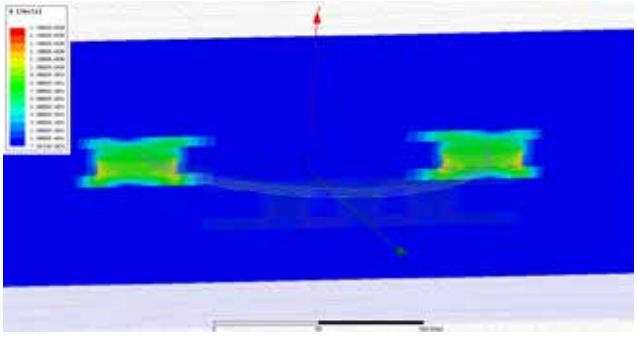


^ L'Arbre à Vent® (Wind Tree) model

> **"New Wind engineers were able to *design and predict* the real-world operation of the *aerodynamic*, structural and electromagnetic parameters of the *wind tree*."**

ENGINEERING A FUNCTIONAL LEAF

The engineering team realized that the best approximation to a leaf would involve a commonly used vertical-axis wind turbine on a rotating shaft called a Savonius turbine. These turbines are typically cylindrical, with two opposing quadrants cut out to form curved, S-shaped scoops that catch the wind and spin the turbine on its axis, creating torque. Because a cylindrical leaf was not acceptable for aesthetic purposes, an ellipsoid shape was chosen instead.



Dynamic 3-D magnetic field simulation with ANSYS Maxwell allowed engineers to determine plate thickness, type of magnets, material used for plates, thickness of the air gap and size of the magnets for each leaf generator.

Engineers began the design work using an open-source software simulation package, but it soon became apparent that time-consuming software modifications would be necessary to perform the required simulations. Based on previous academic and professional experience with ANSYS solutions, New Wind engineers chose ANSYS Fluent, ANSYS Mechanical and ANSYS Maxwell to tackle the aerodynamic, structural and electromagnetic challenges they faced. They were soon performing fluid–structure interaction multiphysics simulations to develop a robust, efficient plastic leaf for the wind tree.

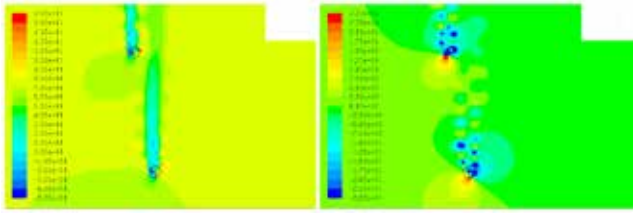
Efficiency, along with the predetermined size and shape of the tree, placed limits on the possible range of leaf dimensions. The leaves had to be large enough to efficiently capture the wind and small enough to be aesthetically proportional to the rest of the tree. Engineers ran parametric simulations using ANSYS Mechanical and ANSYS Fluent to determine the optimal leaf size and material. The leaf size ultimately was determined to be 376 mm at its widest point. It was constructed of acrylonitrile butadiene styrene (ABS), an inexpensive and readily available plastic. This design optimized wind-capturing efficiency and acceptable strain on the leaf during high-wind conditions. Based on wind-tunnel tests of two leaves, they determined that a maximum of 63 leaves could be distributed on the tree structure.

CHECKING FOR AERODYNAMIC INTERFERENCE

Sixty-three leaves was the theoretical maximum based on two assumptions:

- The individual influences of any number of leaves on a randomly chosen reference leaf can be added to give the overall influence on the reference leaf.
- The influence of two leaves on each other does not depend on their locations.

These assumptions had to be confirmed using CFD simulations before the engineers could proceed with the design based on 63 leaves. If the first leaf to encounter wind blowing from a certain direction prevented sufficient wind from reaching a nearby leaf, energy efficiency would be impaired. Interactions between leaves had to be minimized.

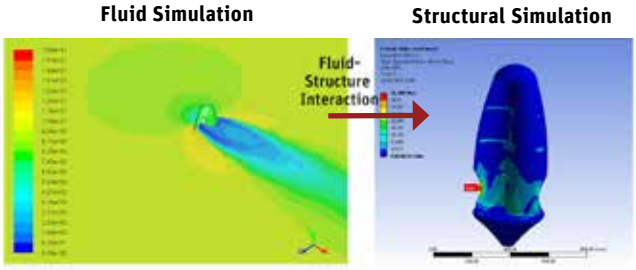


Using fluid dynamics simulation, engineers were able to determine placement of the leaves on the tree so that the wake of one did not affect the next leaf and more power could be generated. Simulations show little effect from wind speed (left) but significant interaction from static pressure (right).

The aerodynamics team used ANSYS Fluent CFD simulations to study the wind flow around a leaf under various velocity and directional conditions to determine the nature of flow in the wake behind a given leaf. Essentially, these simulations were performed to validate the experimental results of two leaves in a wind

tunnel, which the team had done earlier. Fluent CFD results confirmed the experimental results of the wind-tunnel tests to a good approximation. CFD showed that the wake of one leaf did not affect the efficiency of a nearby leaf. Also, simulation of three leaves confirmed the assumption that the influence of a number of leaves on a reference leaf can be summed. So 63 leaves was a valid number.

Finally, engineers performed a 3-D Fluent simulation on one leaf to measure the power coefficient, which is the ratio of the actual power that you can extract from a wind turbine of a particular design over the maximum available power. For the ellipsoidal leaf design, the power coefficient is 20 percent. Calculation of this value depends on the ratio of the speed of the rotating leaf and the wind speed. These two speeds will be important in describing the electromagnetic design of the wind tree.



The structure of the ABS wind turbine was examined using fluid–structure interaction.

PERFORMING ELECTROMAGNETIC SIMULATIONS

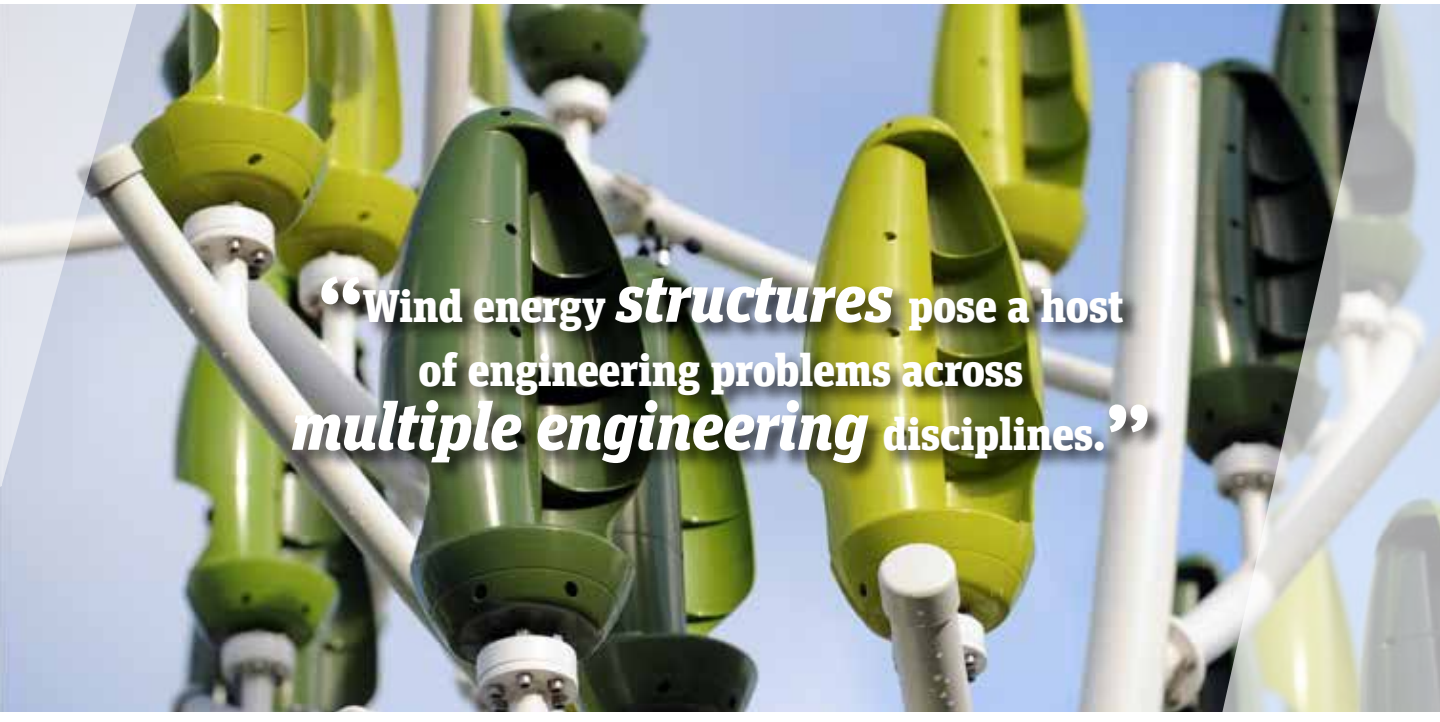
Each leaf of the wind tree has an electrical generator in its base comprising 16 coupled magnets on the rotor. The magnets move with the rotating leaf and coils on a PCB that together produce a three-phase voltage proportional to the rotation speed of the turbine and the magnetic field in the air gap.

New Wind engineers used ANSYS Maxwell simulations to design the generator, including specification of magnetic plate thickness, type and size of magnets, material used for the plates, and thickness of the air gap. A 3-D static magnetic field simulation was performed to determine the mean magnetic field in the air gap, while a 3-D dynamic magnetic field simulation was run to predict power generation parameters, such as induced current and voltage.

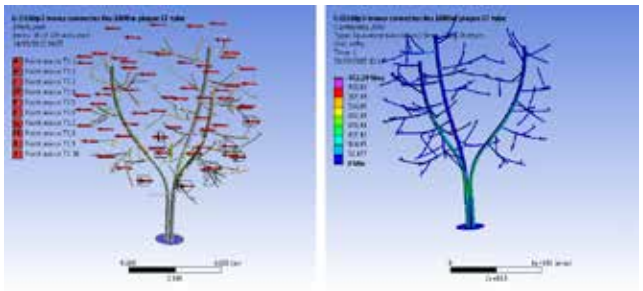
Each leaf generates an alternating current that is transformed into a continuous current (DC). A microcontroller in the bottom of the leaf controls the current of each mini wind turbine. For each leaf there is an optimum ratio between its rotation speed and the wind speed. To extract the maximum energy possible — 20 percent — from a New Wind–type leaf, it is important that the leaf rotates at a speed that produces this optimum ratio for the specific incoming wind speed. The microcontroller regulates the rotation speed of the leaf to achieve this ratio every time.

SIMULATING TRUNKS AND BRANCHES

In addition to fluid–structure interaction simulations performed with Fluent and Mechanical on the plastic leaf to ensure that stresses would not exceed the elastic limit of the ABS material, New Wind engineers also ran structural simulations on the steel framework of the wind tree. They



performed static structural calculations with Mechanical to determine the static loading on the 63 turbines, modeling each as a point mass with an assigned mass value where it was attached to the framework. They also extracted the shell model from a CAD model. By allocating masses and wind forces acting on each turbine, these simulations confirmed that stresses were within acceptable limits everywhere on the wind tree.



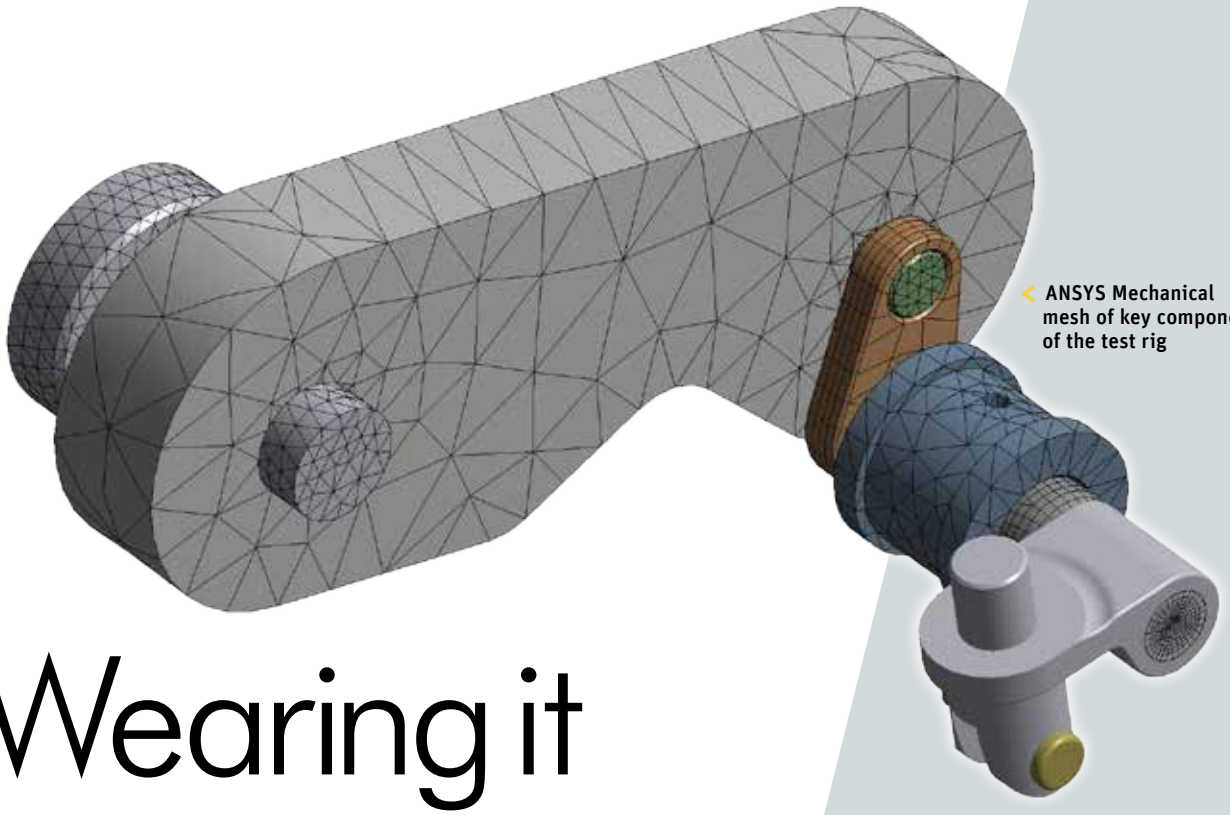
^ Wind loading was examined using structural simulation.

MAKING WIND ENERGY CHIC

With the help of ANSYS Fluent, Mechanical and Maxwell engineering simulation solutions, New Wind engineers were able to design and predict the real-world operation of the aerodynamic, structural and electromagnetic parameters of the wind tree that their company's founder had envisioned only a few years before.

The first prototype wind tree was "planted" in Bourget in 2015, and a wind tree was put on display at Roland Garros, the site of the annual French Open tennis tournament, in May 2016. Reviews from people were overwhelmingly positive. Some large companies and municipalities have expressed interest in purchasing and installing wind trees around the EU. Engineering work, including simulation, continues in an effort to further improve all aspects of the wind tree's operation, and to reduce its cost. While the wind tree is not going to replace large-scale wind farms because it can't produce as much power, New Wind hopes to change the image of wind power in general to a more positive one by demonstrating it in a pleasing form on a human scale. ^

Fluid-Structure Interaction
ansys.com/leaf



ANSYS Mechanical mesh of key components of the test rig

Wearing it WELL

Instead of predicting wear on nonuniformly loaded sliding surfaces using long and expensive physical tests, **BorgWarner** tested a new capability in ANSYS Mechanical to predict wear analytically. The company is on its way to reducing the time to predict wear on a turbocharger wastegate shaft from several days to one day. This capability will expedite design iterations between designers and analysts, and help to create more accurate wear-test rigs.

By **Aliihsan Karamavruc**, Senior Computer-Aided Engineering Analyst, BorgWarner Turbo Systems, Asheville, USA

In turbocharged engines, exhaust gases drive a turbine that spins an air compressor to move additional air into the cylinders, so that they can burn more fuel with each combustion cycle. Turbochargers were originally developed to improve internal combustion engine performance, but today they are primarily used to reduce fuel consumption and emissions. The turbocharger uses wasted exhaust energy to contribute to engine efficiency and thus increase fuel economy. A turbocharged engine of equal power is also smaller, which decreases frictional and thermal losses to further improve the fuel efficiency of the vehicle. An engine with a turbocharger is lighter than a conventional engine with the same power, providing even more fuel savings.

The majority of turbocharged gasoline applications require a wastegate (WG), a boost-controlling device that allows a portion of the exhaust flow to bypass the turbine wheel as necessary. This in turn reduces the power driving the turbine wheel to match

Typical BorgWarner turbocharger

> "When designing turbocharger wastegate actuators, BorgWarner must ensure the life of the actuators over *several millions of cycles* of operation."

the power required for a given boost level. A WG prevents the boost pressure from climbing indefinitely and consequently blowing the engine. An internal WG is built into the turbine housing and an external WG is constructed outside the housing.

BorgWarner is a global product leader in powertrain solutions, with a focus on developing leading powertrain technologies to improve fuel economy, emissions and performance. When designing WG actuators, BorgWarner must ensure the life of the actuators over millions of cycles of operation despite nonuniform loading that contributes to the difficulty of predicting wear patterns. A new ANSYS Mechanical feature that analyzes wear between sliding parts provides accurate predictions in one day — a big-time savings compared to the several days that were required in the past to perform physical testing.

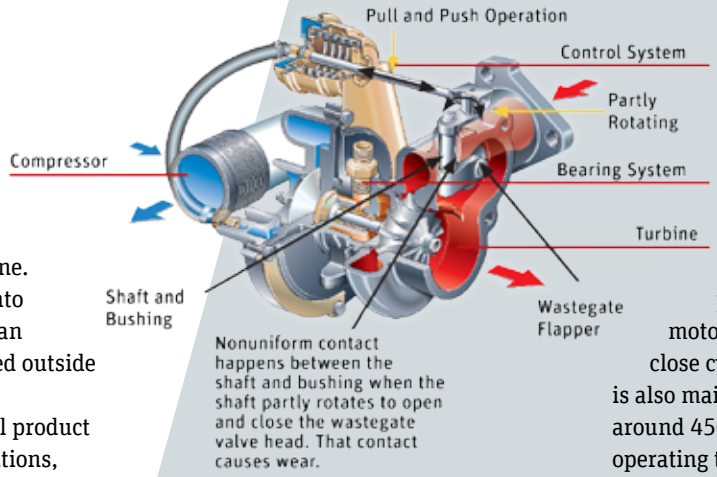


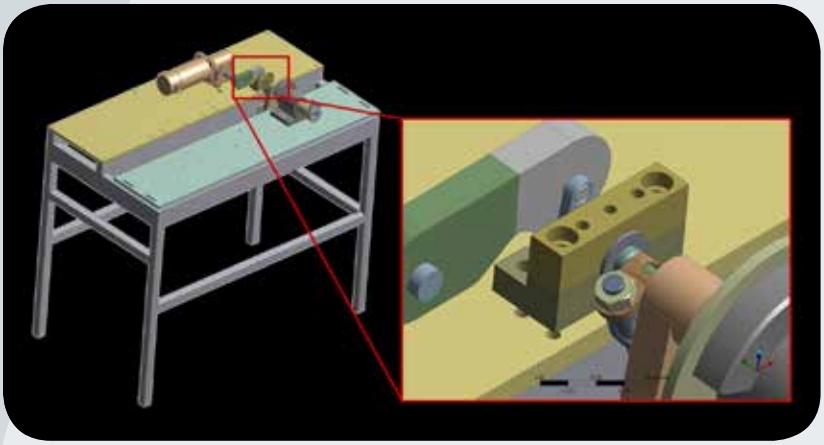
Diagram shows how a wastegate actuator works.

LIMITATIONS OF PHYSICAL TESTING

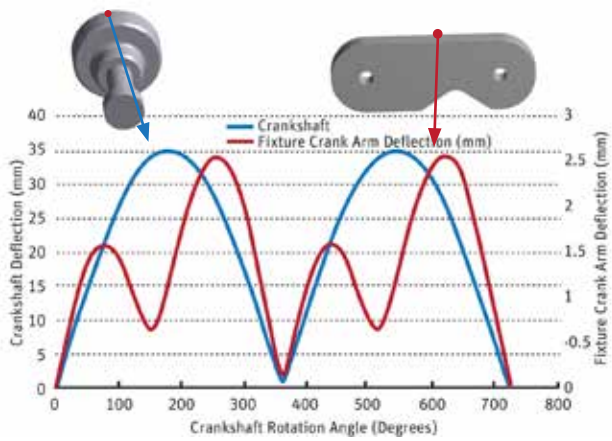
In a popular style of BorgWarner turbocharger, the WG actuator shaft is mounted vertically inside the turbocharger. One end of the shaft is connected to a flapper in the exhaust gas flow stream that seals the WG. Exhaust gas exerts a continual force on the flapper, while a spring at the opposite end of the shaft resists the force to keep the WG closed. The internal spring of an actuator is calibrated to a predetermined boost level. When this boost level is reached, the flapper opens and allows exhaust gas to bypass the turbine. From a wear standpoint, the primary concern is wear on the shaft and bushing, which is difficult to predict because of the nonuniform contact due to force exerted by the flow stream and rotational motion of the shaft.

In the test rig, a motor drives an eccentric crank shaft connected to a crank arm that moves the shaft back and forth over the range that would be driven by the flapper. A 12.8 kg mass hangs on the end of the shaft to represent the force exerted by the flow stream on the flapper. Each revolution of the motor represents one open-and-close cycle of the device. The shaft is also maintained at a temperature of around 450 C to replicate the real-life operating temperature. The test rig accurately predicts the wear experienced by the WG actuator during turbocharger operation, but it requires building an expensive prototype. In addition, running the test rig through enough cycles to predict the wear on the actuator takes about a week.

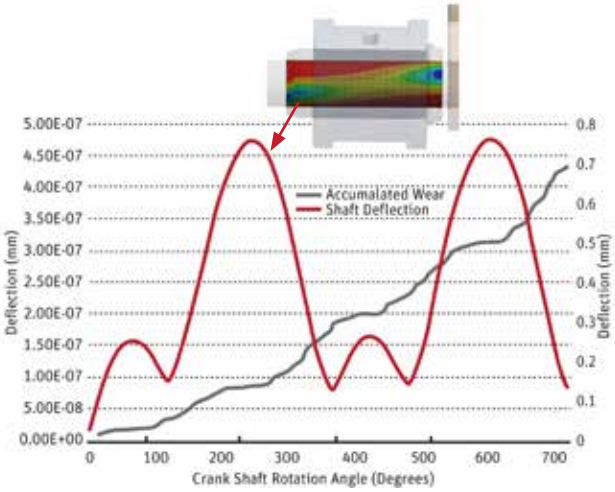
Without analytically predicting the wear, BorgWarner engineers often found that their first design did not meet wear-life specifications, so the entire design, build and test process needed to be repeated, often several times. The ability to determine wear



CAD model of test rig



Deflections of key components vs. crank shaft rotation angle



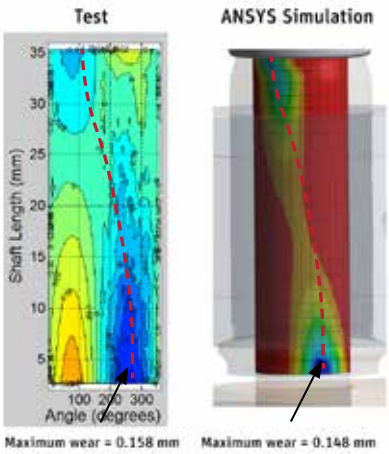
Wear as predicted by simulation. These results showed a good correlation with test results.

on the shaft and bushing prior to building a prototype would save time and avoid multiple prototypes. Until recently, the only method available to analytically determine wear had been using the Archard wear equation, which describes sliding wear based on the load, sliding distance, hardness of the contacting surfaces and a dimensionless constant K. While this equation is useful in predicting wear on evenly loaded surfaces, it does not address nonuniform loading, so it cannot be used in this case.

ANSYS MECHANICAL NOW CALCULATES WEAR

Recent releases of ANSYS Mechanical have given engineers the ability, for the first time, to calculate wear based on nonuniform loading. In this case, BorgWarner engineers began with a computer-aided design (CAD) model of the test rig, including the crankshaft, crank arm, bushing, shaft and pendulum assembly (which holds the weight representing the flow stream pressure). The boundary conditions for the model included a fixed support holding the bushing in place, a mass connected to the end of the shaft, and rotational joints in the bushing and crank arm. Material properties were defined as a function of temperature. Material hardness was defined as a function of the yield stress of the underlying elements, but temperature was not included in this simulation. The generalized Archard wear model was used to predict wear based on the loads calculated at each point in the contact zone by the ANSYS Mechanical simulation. The value of K was determined based on an engineering handbook. A frictional contact was used between the shaft and the bushing.

Engineers ran the simulation over 720 degrees of motor rotation, which amounts to two open-and-close cycles of the actuator. The contact nodes were moved as per the wear increment at each time step. Additional equilibrium iterations for the corrected deformation were then performed. The software performed rezoning whenever the mesh became distorted due to wear.



Wear results showed a good correlation with test results. Test duration was 4 days with 691,200 cycles at a temperature of 450 C and a rate of 2 Hz. ANSYS simulation represents only two cycles.

ACCURATE PREDICTION OF WEAR

The simulation results included deflection of components as a function of crank shaft rotation angle, which plays an important role in the resulting wear pattern. The simulation also calculated contact pressure as a function of crank rotation angle, an important predictor of wear. The contact pressure over the surface of the shaft as predicted by ANSYS software matched the wear patterns on a shaft that had undergone physical testing. The software also predicted the wear generated on each node of the shaft during two cycles of rotation. Engineers plotted accumulated wear over two cycles and extrapolated this information for the full one-week test period.

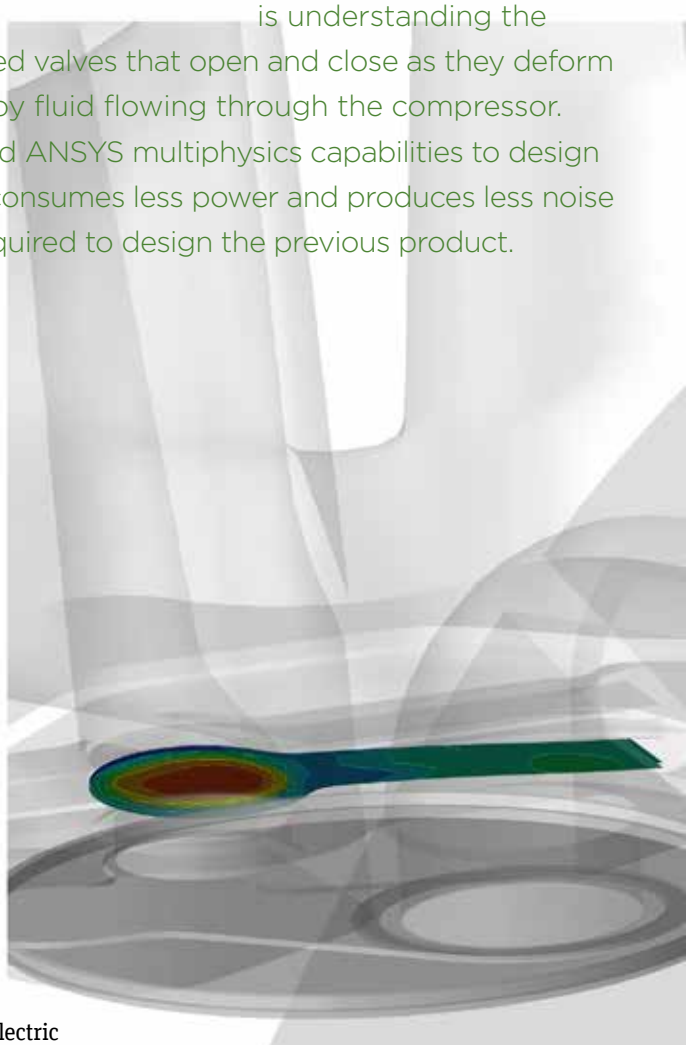
The new simulation capability in ANSYS Mechanical can predict wear with a high level of accuracy. This process reduces the time needed to investigate a design from several days to just one day. This capability will save BorgWarner time and money by making it possible to evaluate different design alternatives based on their wear performance prior to the prototype stage, so that just one prototype can be built with a high degree of confidence.

Structural Durability
ansys.com/durability

Green Machine

One of the biggest challenges in designing reciprocating compressors is understanding the complex operation of reed valves that open and close as they deform under loads generated by fluid flowing through the compressor. Tecumseh engineers used ANSYS multiphysics capabilities to design a new compressor that consumes less power and produces less noise in under half the time required to design the previous product.

By **Luis Lopez**, Research and Development Manager, and **Jônatas Lacerda**, Research Engineer, Tecumseh Products, São Carlos, Brazil; and **Celso Takemori** and **Edmar Baars**, Engineers, Vibroacustica, Joinville, Brazil



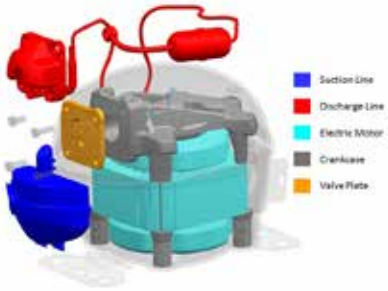
The plate valve deforms according to the surrounding transient pressure field.

Reciprocating compressors use a piston and cylinder, similar to an internal combustion engine, to move heat from a refrigerator's interior to the exterior. While an internal combustion engine burns fuel in the cylinder to move the piston and turn a crankshaft, a reciprocating compressor uses an electric motor to turn a crankshaft that moves the piston to compress refrigerant in the cylinder. In an internal combustion engine, valves that open to deliver fuel to the cylinder and allow exhaust gases to escape are mechanically driven by a camshaft. Reciprocating compressors, on the other hand, typically use reed valves whose open/close movements are driven by fluid flowing through the compressor.

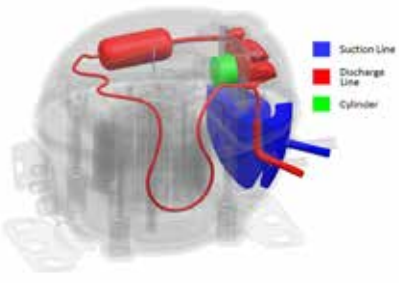
Much of a typical compressor's losses occur around these valves, so the compressor's energy efficiency and noise levels largely depend on their design and operation, as well as on the suction muffler (suction side) and shock–loop system (discharge side), which are used to attenuate noise. As a general rule, restricting flow reduces a compressor's efficiency while making it quieter. In the past, designing a new reciprocating compressor was largely by trial and

> **“Tecumseh engineers took only 18 months using *ANSYS multiphysics simulation*, a reduction of over 60 percent in time to market.”**

error. It required building and testing dozens of prototypes, and it took four to five years to complete. In designing a new TA[2] family of compressors, Tecumseh engineers took only 18 months using ANSYS multiphysics simulation, a reduction of over 60 percent in time to market. They achieved a 1.5 percent increase in energy efficiency — a gain that if applied across all of the refrigerators in Brazil would save enough electricity to power the homes of a city of almost 1 million inhabitants. Furthermore, noise was reduced by 3 dB, cutting the noise level in half. It is the same result as if two compressors were running and one was shut off.



^ Diagram of compressor parts



^ Fluid volume of compressor

DIFFICULT DESIGN CHALLENGE

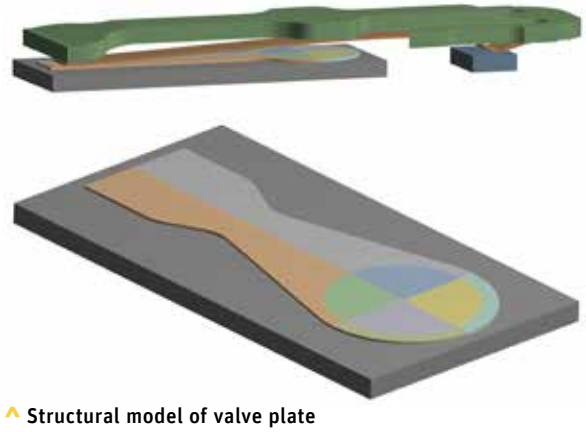
In a gasoline-fired engine, the valves are controlled mechanically, so their position at any point in the machine cycle is always known. However, for reciprocating compressors, the valves are controlled by pressure differences, and a fluid–structure interaction (FSI) simulation is needed during design to determine their position. The cycle begins when the electric motor moves the piston to enlarge the compression chamber, thus reducing its pressure. The pressure difference between the compression chamber and ambient pressure deforms the suction valve, which causes refrigerant to be sucked into the cylinder. When the piston reverses direction and increases the pressure in the chamber, the discharge valve opens to dispense pressurized refrigerant.

Tecumseh management tasked the engineering team with developing a replacement for the company’s TA family of compressors that delivered higher energy-efficiency and lower noise at a lower cost. Reducing the cost of the compressor required using a less expensive and less efficient electric motor, so it was even more critical to increase compressor efficiency. Meeting these conflicting objectives within a tight time frame required the engineering team to develop a complete software prototype of the compressor, incorporating both mechanical components and fluid flow.

SETTING UP A MULTIPHYSICS SIMULATION

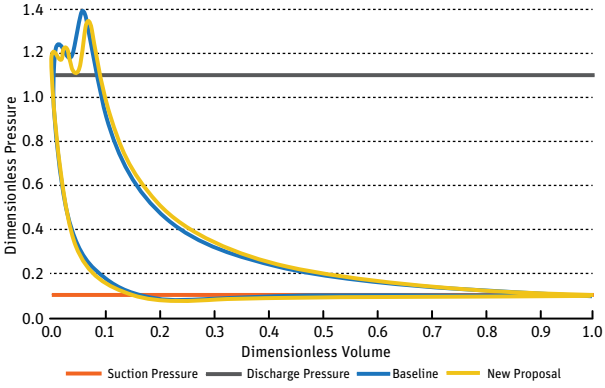
Tecumseh engineers selected ANSYS Mechanical finite element analysis software and ANSYS CFX computational fluid dynamics (CFD) software because these tools can easily be connected within the ANSYS Workbench environment to simulate FSI. They created both solid and fluid geometries in the ANSYS DesignModeler pre-processor, and used Workbench to automatically mesh components and define physical models and boundary conditions. Moving meshes represented the movement of the cylinder and the valves. Using the Workbench project schematic, engineers easily connected the output of each tool to the boundary conditions of the other to simulate FSI.

Pressure–volume diagrams generated by the simulation, which track cylinder pressure against cylinder volume over the operating cycle of the compressor, were extremely useful. The area near the top of the chart, where cylinder pressure exceeds discharge pressure, quantifies discharge losses. The area under the suction pressure line, where cylinder pressure is less than suction pressure, indicates suction losses. Suction losses are caused primarily by pressure drop across the suction port when the suction valve opens and closes. Discharge losses are caused by pressure drop across the discharge port.

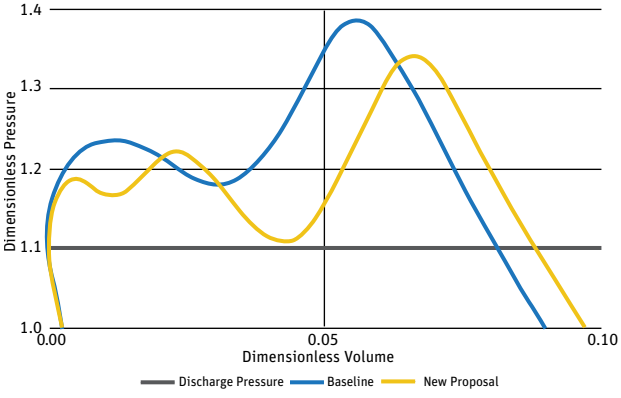


^ Structural model of valve plate

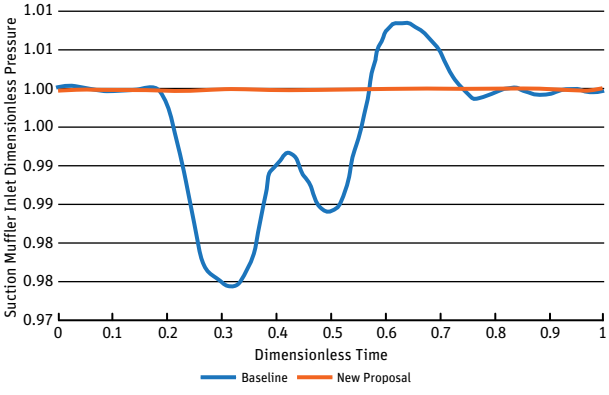
"*Multiphysics simulation* makes it possible for Tecumseh to resolve conflicting requirements while bringing *products to market* faster than ever before."



Pressure-volume chart quantifies losses on suction and discharge sides of compressor.



Discharge losses were reduced in the new design (blue) versus the previous generation (yellow).



Pressure–time history shows cyclical pressure variations that cause noise.

The simulation provided time histories of pressure at suction and discharge ports. Because noise is caused mainly by pressure fluctuations around the suction and discharge valves that exit the refrigerator cabinet, these time histories provided valuable information. Guided by the simulation results, Tecumseh engineers manually iterated the design of plate valves and other components, substantially increasing efficiency and reducing the noise the compressor generated. Since the suction side generates the majority of noise, engineers designed a more elaborate suction muffler. This new muffler necessarily had higher losses, but these were more than offset by improvements in the suction valve's efficiency.

A reciprocating compressor's efficiency is measured by a coefficient of performance (COP), which is the amount of work it performs in moving heat out of the refrigerator's interior, in watts, divided by its power consumption, also in watts. By simulating many different designs, Tecumseh engineers were able to make trade-offs that achieved the optimal combination of increased efficiency and reduced compressor noise. The resulting COP improvement of 1.5 percent is a major enhancement for a mature product, providing Tecumseh with a substantial competitive advantage.

The new compressor's noise levels were also substantially reduced. After the design was completed, Tecumseh engineers used ANSYS ACT Acoustics (available from the ANSYS App Store) to simulate the noise levels on the suction side of the old and new designs. The results showed an improvement on the suction muffler performance around 20 dB at most frequencies. Tecumseh engineers also performed a modal analysis of the compressor housing using ANSYS Mechanical, and, based on these results, they modified its design to reduce its response at the compressor's operating frequency, providing further noise reductions.

The appliance industry continues to demand reciprocating compressors with higher energy-efficiency and lower noise at a competitive price. Multiphysics simulation makes it possible for Tecumseh to resolve these conflicting requirements while bringing products to market faster than ever before.

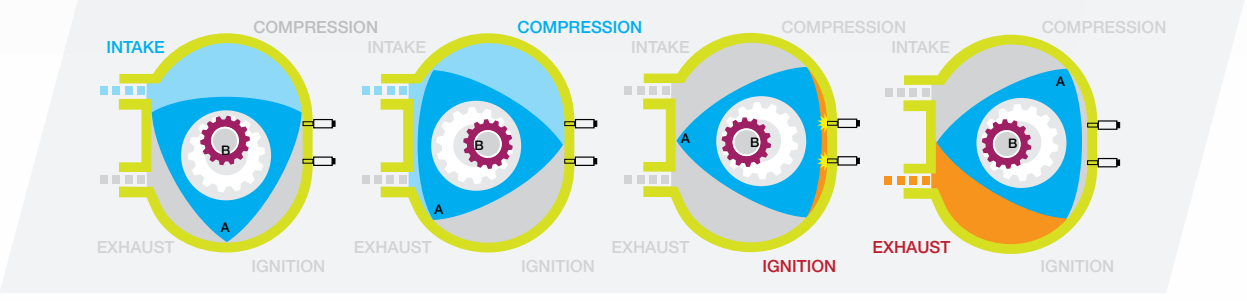
Tecumseh Products is supported by ANSYS Elite Channel Partner ESSS.

Pouring COLD WATER on It

To design one of the highest power-density rotary engines ever developed, engineers at Orbital Power needed to cool the housing sufficiently to preserve the life of the rotor tips. They accomplished this goal using ANSYS software to optimize the design of the water cooling jacket. This required less than one-third of the time that would have been required using build-and-test methods.

By **Shawn Okun**, President,
and **Wenwei Zeng**,
Combustion Engineer,
Orbital Power Ltd.,
Lakeland, USA

Most of us are familiar with internal combustion engines that have pistons that move back and forth, reversing direction. However, in a rotary engine (or Wankel engine, named after its inventor), the parts rotate and move only in one direction. A four-stroke cycle within a combustion chamber located in a peanut-shaped housing drives a three-lobed rotor. Intake, compression, ignition and exhaust occur within the four chambers defined by the spinning rotor inside the housing. When compared to piston engines, rotary engines are generally simpler, smoother and more compact. Their higher revolutions per minute, and high power-to-weight ratio, make them perfect for applications where high power and light weight are needed, such as for portability purposes.



^ Diagram of the four-stroke cycle of a generic rotary engine. *Courtesy: Y_tambe's file. Permission =GFDL*

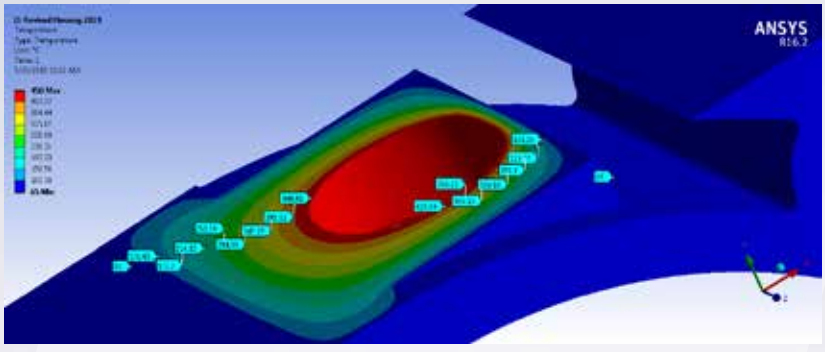
Orbital Power produces rotary engines ranging from 2.5 to 40 horsepower for applications such as generator sets and unmanned aerial vehicles. The company's new ORB-20A rotary engine enables a person to carry a 500-watt generator in a backpack. Its high power-density stems partly from the fact that the new engine burns heavy fuels, such as kerosene, that have higher caloric value than lighter fuels, such as gasoline. Heavy fuels are also less expensive, safer to handle, easier to store and transport, and have better lubricating qualities. Another Orbital Power engine/generator combination delivers 10 KVA with a 40 horsepower engine that weighs 17 pounds, a fifth of the weight of competitors' products.

> "Guided by ANSYS simulation tools, Orbital Power achieved a 93 percent decrease in cost and a 70 percent decrease in design time."

On the other hand, heavy fuels present a major design challenge because they generate more heat, and orbital engines already have a natural tendency to run hot. Combustion occurs near the tips of the rotors where they seal off the housing, and temperatures exceed the limits of all usable rotor tip materials. Orbital Power overcame this obstacle by using ANSYS CFX computational fluid dynamics (CFD) software to develop a unique water cooling system that keeps the temperature of the tips at safe levels.

THERMAL DESIGN CHALLENGE

The original concept design of the ORB-200L used air cooling, but Orbital Power engineers quickly determined that water cooling was needed to maintain safe operating temperatures. They developed an initial design in which water was pumped through internal passages in a water jacket to remove heat generated during combustion. Most of the heat generated in a rotary engine is near the ignition and exhaust chambers, so another goal of the cooling system is to distribute heat over the entire housing as much as possible.



Temperature distribution across water jackets on early development design shows hot spot in red.

Orbital Power engineers defined the geometry of the engine in computer-aided design (CAD) software and imported the model into ANSYS Workbench. The geometry contained many imperfections, such as tiny gaps and overlaps, which are not relevant to the flow analysis. Removal of such imperfections often improves the subsequent mesh and reduces the time it takes to generate the solution. Engineers used ANSYS SpaceClaim to remove these unnecessary features, correct the imperfections with automated tools, and directly edit and manipulate faces — in real time without rebuild errors. They used ANSYS meshing to create a mesh with more than 30 million elements and simulated the engine's performance with CFD.

The resulting temperature distribution showed numerous hot spots ranging up to 212 F, which is unacceptable for the materials used in the engine. The simulation showed that the hottest areas corresponded to recirculation zones that prevent cool water from entering. It also showed that the outlet temperature was above 190 F, and that the left side of the engine was much hotter than the right side.

Orbital Power ORB-200L rotary engine

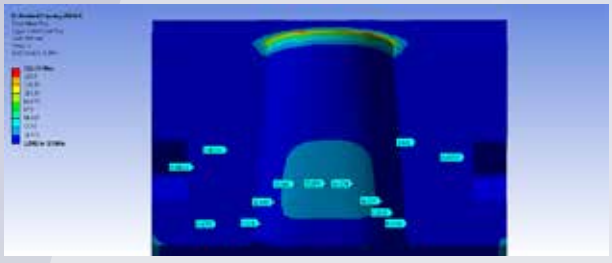
ITERATING TO AN OPTIMIZED DESIGN

Guided by the simulation results, Orbital Power engineers then created designs using fins and curved walls to alter the direction of flow through the water jacket in an effort to eliminate the hot spots and even out the temperature throughout the jacket. Engineers ran about 40 design iterations while manually adjusting the direction and angle of deflectors, the size and angle of the inlet and exhaust flow, and other variables based on the results of previous

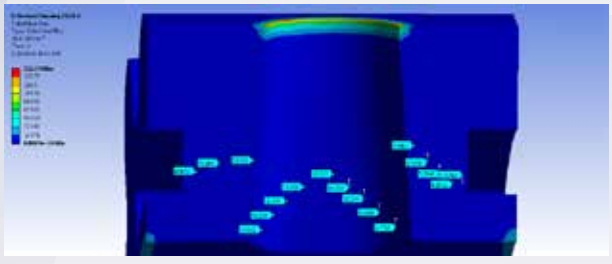
design iterations. They also changed the position of oil pipes that pass through the water jacket for oil cooling, which indirectly affect the water jacket by creating obstructions and releasing heat.

The team achieved a final design that yielded a much more even flow distribution across the water jackets. With a low pressure pump, the hottest point in the water jacket on the left side has a maximum temperature of about 190 F, which is an acceptable value. There are no strong temperature gradients in the water jacket, and the outlet temperature is less than 180 F. The spark ignition area is also cooled very well.

With a high-pressure water pump (as is used in heavy duty applications), the new water jacket design runs at even cooler temperatures. The temperature at all points in the water jacket is less than 160 F, with the hottest area being near the left core side. There is also a relatively even temperature distribution across the water jacket, and the spark ignition area is also well-cooled. The heat absorbed by the oil pipes is minimized by the passing of liquid coolant, reducing the oil temperature significantly. After physical testing and review, the optimized structure was validated in a controlled engine dynamometer testing facility with a dynamic pressure water pump. Results showed the highest effective temperature of the liquid cooling system was less than 185 F, and there was relatively even temperature distribution across the entire water jacket.



Another view of temperature distribution on early development design shows temperature on left side is higher than right side. The left core is between the intake port and the first spark plug, roughly 20 degrees from the port toward the spark plug.



Another view of temperature distribution on final design shows lower temperatures and more even temperature distribution.

SIMULATING TEMPERATURE GRADIENTS IN HOUSING

Orbital Power engineers then used ANSYS CFX and ANSYS Mechanical to address a temperature gradient issue near the exhaust port. The initial concept design showed a gradient from 450 F to 131 F in the cross-section of the housing near the right side of the exhaust port. In this area, the minimum wall thickness between the exhaust port and cooling channel is only 2 mm, resulting in a heat flux exceeding 25 W/mm^2, compared to less than 15 W/mm^2 in the surrounding area. This creates the potential for the coolant to boil and generate bubbles that can hurt the performance of the entire cooling system. Orbital Power engineers used ANSYS Mechanical to evaluate the effect of varying the minimum wall thickness. They ended up with a minimum wall thickness of 5 mm, which reduced the heat flux to 11 W/mm^2 (42 percent of the original value).

Orbital Power estimates that designing the water jacket through traditional build-and-test methods would have cost about $600,000 and taken about 40 weeks. Guided by ANSYS simulation tools, Orbital Power engineers iterated to an optimized design in about 25 man-weeks of engineering effort over a 12-week period; the company pegs the total cost of the simulation effort at about $40,000 including labor, software and computing expenses. This is a 93 per cent decrease in cost and a 70 percent decrease in design time. The optimized liquid cooling system provides temperatures that are 35 percent lower than air cooling and 20 percent lower than the original liquid cooling design, ensuring a long and reliable engine life.

> "The optimized liquid cooling system provides temperatures that are 35 percent lower than air cooling and 20 percent lower than the original liquid cooling design."

Thermal Management with ANSYS Multiphysics
ansys.com/thermal101

Faster than the WIND

America's Cup yachts have changed drastically over the last several years. The boats now fly over the water four times faster with the help of a wing that replaces the traditional sail. Emirates Team New Zealand is working to pull ahead of its competition by using ANSYS multiphysics simulation to evaluate thousands of alternative cases and develop the best possible design in its quest for the next cup.

By **Steve Collie**,
Aerodynamics Engineer,
Emirates Team New Zealand,
Auckland, New Zealand

The America's Cup, the premier event in the elite sport of yacht racing, is eagerly anticipated by fans. Held every few years since 1851, boats and sailors from around the world compete to win the highly prized trophy. The next event is slated for 2017 in Bermuda. The exciting race is a physical and mental challenge for the crew, but behind the scenes and before the race, America's Cup boats require extensive engineering that must occur on a tight schedule. Because physical testing of all possible improvements to a vessel of this complexity is inconceivable (not to mention financially untenable), Emirates

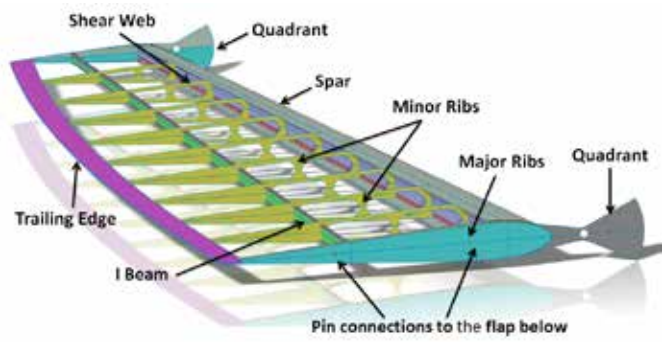
"The integrated suite of *ANSYS multiphysics* tools — fluids, structural and composites simulation — *helps ETNZ* smooth the way."

Team New Zealand (ETNZ) has turned to ANSYS multiphysics engineering simulation software to develop a reliable, robust and competitive craft.

Aerospace technology — including advanced aerodynamics, lightweight materials, wind tunnel testing and advanced simulation technology — has revolutionized the sport of America's Cup racing. Today's cup-class yachts use a wing that is more akin to an airplane's wing or airfoil than to a traditional sail. The wing enables the catamarans to transfer wind into forward momentum (instead of into lift, as in aircraft).

WING DESIGN

The wings are two-element airfoils comprising a main element along with a flap unit that is hinged from the back of the main element. This main element is a rigid structure that makes up the leading edge of the wing, which is the primary structural spar. The flap incorporates three segments that can be cambered to increase lift by changing the angle between the main element and the flap. Individual flap segments can be twisted to depower individual segments and change the center of aerodynamic pressure. Hydraulic actuators are used to camber and twist flap segments into the most efficient aerodynamic shape based on prevailing sailing conditions.



^ Simple model of single segment of flap used in stage 1 simulation

Race rules fix many aspects of the design of these yachts, such as hull shape, wing shape and deck layout. So, beyond the contribution that a skilled crew offers, the America's Cup is largely won or lost based on the ability of the underlying systems, including the wing structure, to deliver optimal performance under a wide range of sailing conditions. Emirates Team New Zealand engineers, who bounced back from a narrow loss in the last race in 2013, were under enormous time pressure to design these systems for a new test yacht within a five-month time frame to stay on track with their schedule leading up to the 2017 competition. Using ANSYS fluid–structure interaction (FSI) multiphysics and composites simulation, ETNZ is able to virtually test hundreds of variations early in the process to find the optimal design before performing any physical testing. Testing using the actual boat is time-consuming and expensive, and it would not be possible to physically test all design trade-offs without simulation.

WING STRUCTURE

Both the main element and the flap are built using lightweight ribs and a spar covered with thermoplastic film. The flap element itself comprises three sections — flap 1, flap 2 and flap 3 — running from bottom to top. Hydraulic actuators that pull and ease control lines are connected to control stations at the top and bottom of each flap section.

The ability to quickly and efficiently control flap twist is critical for fast, accurate, well-controlled sailing. Flap deformation must be optimized to provide the desired span-wise lift distribution while achieving weight targets. Extreme twist leads to large strains in the flaps. For example, flap 3 twists as much as 20 degrees across its length. Because of these large deformations — and a thermal boundary condition used to pre-tension the thermoplastic membranes on the surface of the wing — optimization requires nonlinear structural analysis.

PREDICTING AERODYNAMIC PRESSURES

To determine aerodynamic pressure using ANSYS CFD simulations, ETNZ engineers started with computer-aided design (CAD) models of the main element and flap. They twisted the CAD model to the full range of allowable flap shapes and created inverse-domain models for flow analysis. They simulated

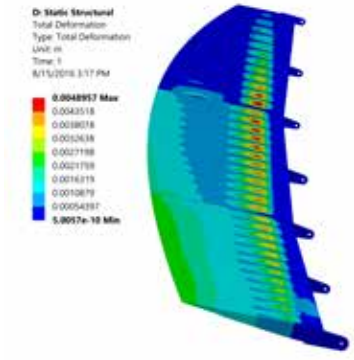
^ CAD model of wing

> **"Today's America's Cup–class yachts *use a wing* that is more akin to an airplane's wing or airfoil than to a traditional sail."**

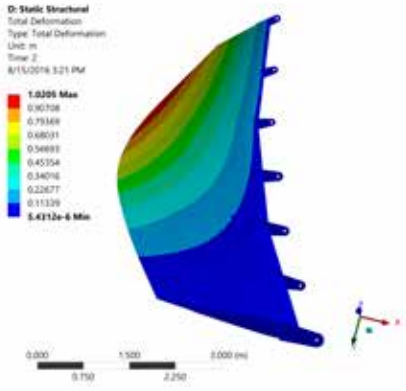
the performance of each shape under a wide range of wind speeds and angles, as well as boat states. The results were formulated into a matrix that was used in a performance simulator to predict boat performance based on wind conditions, trim of the wing, direction into the wind and more. ANSYS DesignXplorer minimized the number of simulations required to accurately represent the parameter space. This analysis required assessing the impact of an extremely large number of variables. The open ANSYS framework allowed the complete simulation process to be fully automated using in-house scripts.

based on the deformed wing shape, pressure data must be transformed to the undeformed wing shape, using scripts written by ETNZ engineers, so that it can be applied to the structural model. Pressures can then be mapped to the structure using ANSYS Mechanical. This process and model were developed with the assistance of technical experts from ANSYS and channel partner LEAP Australia, which enabled ETNZ to meet the tight deadlines.

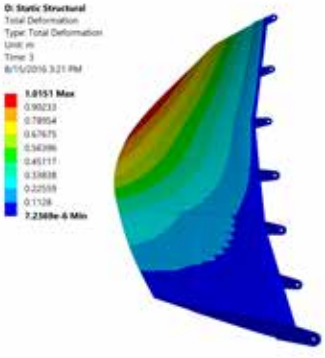
ETNZ engineers progressively increased the complexity of their structural models. Stage 1 was a simple model of a single flap used for structural design. Stage 2 was a combined model of all three flaps that was used primarily



Stage 3 simulation loadstep 1: A thermal boundary condition is used to tension the cover membrane.



Stage 3 simulation loadstep 2: Each control station is rotated to apply twist to the wing.



Stage 3 simulation loadstep 3: Aerodynamic pressures are imported from ANSYS CFX and applied to the wing.

STRUCTURAL SIMULATION

The flap design process began with an initial geometry proposal defined in CAD. Using ANSYS Workbench, engineers built the structural finite element model. The flaps are made of lightweight composites, so the complete layup was designed and optimized using ANSYS Composite PrepPost. To set up a one-way FSI simulation, pressures from the many CFD analyses were imported and mapped onto the structure. Because CFD simulations are carried out

to design the connection geometry. In stage 3, pressures determined by CFD simulations were used to load the model developed in Stage 2. This analysis determined the loads required to articulate the flaps, which were, in turn, used to design the hydraulic actuators and control system.

The team ran hundreds of iterations to optimize the geometry and laminate structure to achieve target shapes and minimize input moments under specified loading conditions. At the same time, effort was made to minimize the weight of the structure while ensuring that it could withstand the loads expected during a race.

Emirates Team NZ Sails toward America's Cup with ANSYS
ansys.com/ETNZ

> "The team ran hundreds of iterations to optimize the geometry and laminate structure to achieve target shapes."

THE BOAT LAUNCHES AND DESIGN CONTINUES

The designs described were used on a 45-foot test boat launched in summer 2016. Meanwhile, ETNZ engineers are in the process of creating a more comprehensive model that includes the main element and its connection to the yacht; it will later be expanded to the yacht structure itself. Engineers will use this model for detailed design of the main element along with bidirectional FSI to analyze the effects of the structure on the aerodynamics of the wing. Bidirectional FSI analysis will make it possible, for the first time, to investigate the wing's dynamic response, when it is struck by a gust of wind, when the sail is being trimmed, or when the boat undergoes maneuvers like gybes and tacks.

The current design will achieve speeds approximately four times faster than boats racing in the America's Cup 10 years ago. The main reason for this increase is that Cup teams now race catamarans, rather than the monohulls used in the past. Flying the boat on its foils (foiling) also has increased top speeds dramatically. Moving from sails to wings provides a further substantial speed improvement.

Converting wind to speed while wasting as little energy as possible is no easy task. ETNZ uses a 100-percent simulation-driven development process to test thousands of alternatives to meet its design goals for the wing, including accurate and fast control while keeping within weight targets. The integrated suite of ANSYS multiphysics tools that includes fluids, structural and composites simulation helps ETNZ smooth the way. The end goal is to repeat its successes of 1995 and 2000 and bring the cup back home to New Zealand.

Emirates Team New Zealand is supported by ANSYS channel partner LEAP Australia and composites design specialists from ANSYS.

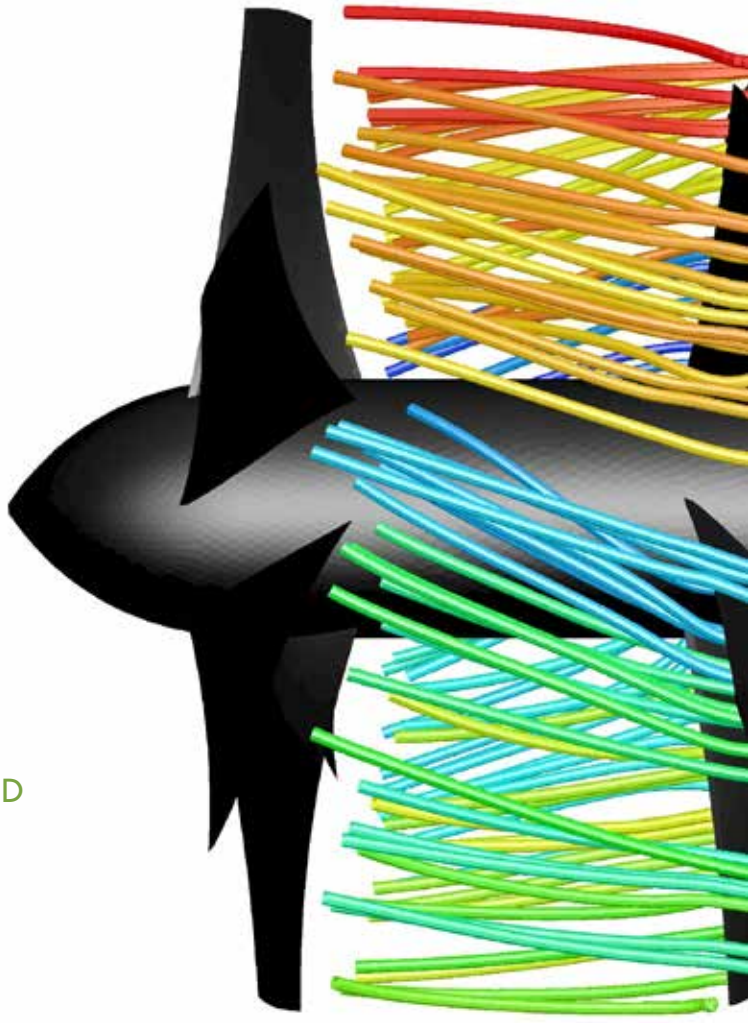
POWERING ENGINEERING

Teams in elite sports like Formula One racing and yachting must push their equipment to the highest level of performance to remain competitive. They adopt the best technology from every realm and push the envelope to gain an edge. Engineering simulation has been embraced in these sports because it allows team engineers to virtually test multiple design variations to gain the best possible results long before the equipment is built.

In the case of an Americas Cup yacht, converting wind to forward momentum while wasting as little energy as possible is not much different from driving a **wind turbine** to generate power. They both require capturing the wind to create energy; if more is captured (and controlled), a better result is achieved. The same is true of any **turbine** (whether driven by gas, wind, steam or water) to generate electricity or to propel an aircraft. And, just like a yacht, an F1 car or an aircraft, aerodynamics and weight contribute to how much energy is consumed.

Engineering simulation is critical to any engineer designing on the leading edge of breakthrough energy innovation — whether in industry, sport or academic research.

COLD CASH

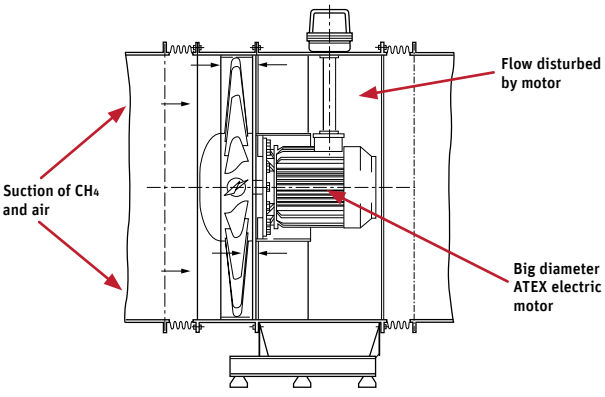


A FAN DESIGNED WITH MULTIPHYSICS SIMULATION OFFERS A POTENTIAL OF 1 BILLION EUROS IN LIFETIME SAVINGS FOR ALL OF THE LNG PLANTS OPERATED BY A LARGE GLOBAL PRODUCER.

By **David Ohayon**,
Chief Technology Officer,
Nova Simulations,
Marseilles, France

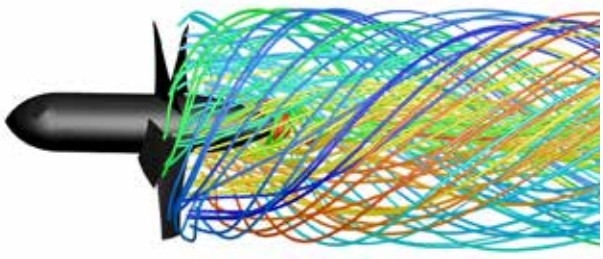
The flammable nature of liquefied natural gas (LNG) creates the need for strict safety procedures while it is condensed by refrigeration from its gaseous state. LNG plants operating in extremely cold climates, such as in Russia and Norway, must use heated enclosures to provide comfortable working conditions for plant operators. Safety regulations require that these enclosures be equipped with ventilation capable of completely exchanging the air inside the enclosure with outside air (air change) 12 times per hour in normal operation and 18 times per hour in emergency conditions, when elevated levels of methane, the primary flammable component of natural gas, are detected.

Existing gas plants typically address this ventilation challenge by using 20 off-the-shelf fans to ventilate a process room 50 meters long by 30 meters wide by 20 meters high. The large motors that are directly connected to each fan create an obstruction that reduces aerodynamic efficiency. Placing the motors inside ventilation ducts also puts them in contact with the airflow and creates an ignition risk. The fans consume about 311 kilowatts (kW) during normal operation, resulting in high electrical costs in the remote, off-the-power-grid areas where the plants are typically located. Furthermore, because each fan has its own motor, it is often necessary to shut down the entire condensation process when maintenance is required on just one of the fan motors because the remaining fans are not capable of meeting ventilation requirements.

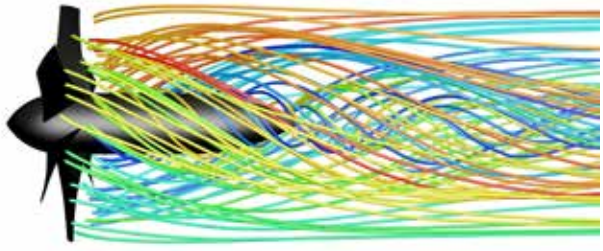


^ Current fan design

Flow streamlines generated by both rotors show how tangential flow is canceled out.



Flow streamlines generated by downstream rotor in isolation show high tangential flow.



Flow streamlines generated by upstream rotor in isolation show high tangential flow.

DEALING WITH TANGENTIAL FLOW

Nova Simulations proposed the design of a custom fan to one of the world's major oil and gas companies. The custom fan would drastically reduce operating costs by optimizing aerodynamic efficiency for this application, and by placing the motor outside the duct. This external motor placement would require the addition of a gearbox, which would create some power losses. But Nova Simulations engineers knew that a custom fan design optimized specifically for this application could more than make up for these losses. The key would be to maximize the amount of air flowing parallel to the axis of the fan (axial flow).

In a regular fan, the spinning motion of the blades generates two types of flow: the preferred axial flow and the less efficient tangential flow — tangential to the circumference of the fan. The energy that goes into producing tangential flow is not available for generating axial flow. Static guide vanes

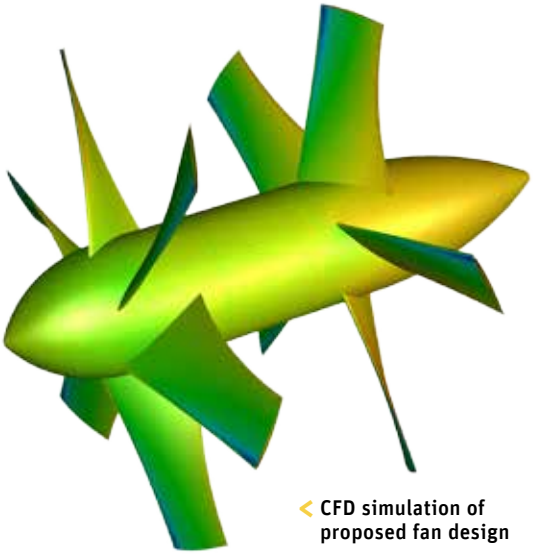
Modeling Fan and Blower Systems
ansys.com/fan-and-blower

"The savings across all of *the company's LNG plants* are estimated at *1 billion euros* once all of the plants are equipped with the *new fan*."

are often used to convert tangential flow into axial flow, but this conversion process also generates considerable losses.

Rather than simply designing the fan to minimize tangential flow, Nova Simulations engineers had the idea of using tangential flow to their advantage by positioning two rotors operating in parallel but with opposite directions of rotation within each fan. They knew that each rotor by itself would generate substantial tangential flow, but the downstream rotor could be designed so its tangential flow would cancel out that produced by the upstream rotor, resulting in axial flow conditions. Working in tandem, the two rotors would maximize the desired axial flow through the ventilation duct — something that a regular fan with one set of rotors blades could not accomplish.

Next, they faced the challenge of validating this idea and optimizing the performance of the fan to generate the highest possible airflow for the lowest possible power consumption. In the past, engineers used airfoil design methods to estimate the performance of various design concepts. These methods required many simplifying assumptions, so building and testing a considerable number of prototypes was needed to produce a working design. But in this case the cost of building even a single prototype was so high that funding could be obtained to build only one prototype — and obtaining this funding required near certainty of success.

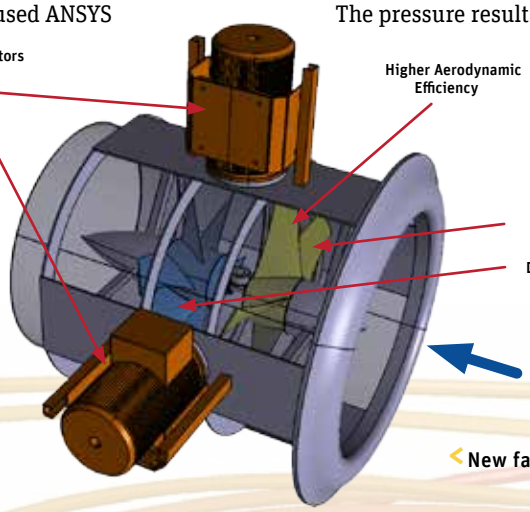


< CFD simulation of proposed fan design

ITERATING TO AN OPTIMIZED DESIGN

To optimize their chances of getting the design right the first time, Nova Simulations engineers used ANSYS Fluent computational fluid dynamics (CFD) software to produce a complete virtual prototype of the fan. CFD takes the full 3-D geometry of the blade and duct into account, eliminating the need for simplifying assumptions and accurately predicting the performance of any proposed fan design. Engineers used ANSYS design exploration to perform the many design iterations required for the CFD study. By varying the number of blades, the blade profile, the pitch of each rotor, the distance between rotors, and the duct diameter, they were able to determine the most efficient design. First they optimized the performance of each individual rotor, and later, the combined performance of the two rotors. The result was that they optimized the aerodynamics of the two rotors synergistically so that each operates separately as efficiently as possible, while the two rotors combine to convert the tangential flow produced by both fans into straight axial streamlines. Simulation allowed them to virtually test many fan designs by automatically exploring many parameters and avoided the need for multiple prototypes.

Carbon fiber fan blades were used to deliver the high strength-to-weight ratio required for this application. The pressure results from the CFD simulation were used to determine the strength requirements for the rotor blades. Nova Simulations



< New fan design

> **“Testing showed that the *prototype performed* nearly exactly as predicted by *simulation*.”**

engineers used ANSYS Composite PrepPost to model the complex composite structure, including the number of layers and the shape, thickness and orientation of each layer. ANSYS Composite PrepPost predicted the ultimate strength and progressive damage over time due to delamination, cracking, pull-out and other destructive mechanisms. Engineers optimized the design of the blade to meet the requirements of the application at a minimum weight.

TWO LAYERS OF REDUNDANCY

Engineers worked with a power transmission company to design a gearbox that provides 95 percent efficiency. The use of a gearbox makes it possible to move the motor out of the duct, which helps to improve the efficiency of the fans; it also increases safety by removing the motor from the airflow, which contains combustible gases. The two rotors operate at the same speed to maximize the efficiency of the gearbox. Each fan is coupled to two motors to provide intra-fan redundancy, enabling the plant to continue operating even when a motor is down for maintenance. In addition, the four fans needed for each process room are coupled to provide an additional layer of redundancy: Even if both motors coupled to a particular fan are down, the fan continues to be driven by other motors.

Prototype of new fan

The simulation predicted that the new fan design operating at 900 rpm would provide a flow rate of 24 cubic meters per second while consuming 12 kW. Increasing the flow rate to 40 cubic meters per second at 1,500 rpm while consuming 50 kW yields a maximum efficiency of 72 percent, nearly double the efficiency of the existing fans. Based on the simulation results, the oil and gas company funded the construction of a prototype and contracted with an independent ISO 5801 laboratory to test its performance. Testing showed that the prototype performed nearly exactly as predicted by simulation. It delivered a flow rate of 24 cubic meters per second at 900 rpm while consuming 10 kW, and 40 cubic meters per second at 1,500 rpm while consuming 46 kW, for a maximum efficiency of 74 percent. The fans will normally operate at 900 rpm to provide 12 air changes per hour, but in the event of an emergency they can be quickly increased to 1,500 rpm to deliver 20 air changes per hour — two air changes per hour higher than the current design. The increased number of air changes means that the process room can be more quickly cleared of methane in the event of a leak.

Nova Simulations engineers are now working on preparations to manufacture the new fan to equip the oil and gas company’s LNG plants, and also for other possible applications where ventilation is critical, such as tunnels, underground parking garages and underground mines. Based on prototype testing, four of the new fans should be able to safely ventilate an LNG process room. The 20 fans used in current process rooms draw 311 kW, while four of the new fans draw only 184 kW, including the draw from the new gearbox. In a typical LNG plant, this reduction in power consumption alone will generate annual savings of 1.5 million euros per year. This approach also makes it possible for the process plant to continue operating under normal conditions as long as four of the eight total motors are available. Thus the plant should never need to be shut down again for fan maintenance, as frequently occurs using the current ventilation system. The reduction in operating costs and downtime provided by the new fan is expected to reduce the cost of operating a single LNG plant by 100 million euros over its lifetime. The lifetime savings across all of the company’s LNG plants are estimated at 1 billion euros once all of the plants are equipped with the new fans.

POWER RETOOLING FOR CHIPS

In a conventional approach, chip power-consumption analysis begins late in the design flow and occurs typically when the physical design is complete. At this stage, possible changes that can impact power are limited by schedule and cost considerations. AMD used ANSYS PowerArtist to apply a design-for-power approach at an early stage in the design flow, making it possible to reduce power consumption to unprecedented levels.

By **Mark Silla**,
Principle Member of Technical Staff,
and **Dave Larson**,
Member of Technical Staff,
AMD, Austin, USA

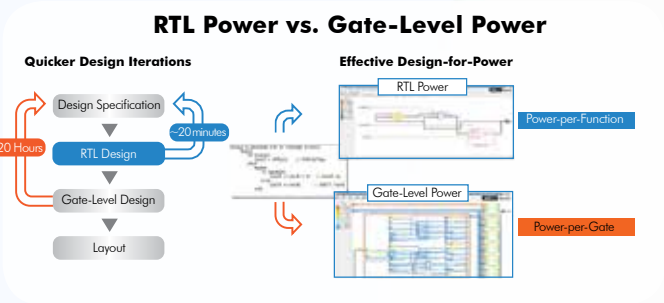
Power has been at the forefront of chip design for mobile applications and is now a key design concern for the Internet of Things (IoT), automotive, networking and other applications. Since the 1960s, the electronics industry has reliably followed Moore's Law. Gordon Moore predicted that computing power would double nearly every 18 months. Achieving this hasn't come easily, as engineers need to continually balance power, performance, reliability and cost. Modern server class processors, for example, contain billions of transistors that switch on and off at gigahertz frequencies, consume several hundred watts of power and generate significant heat. Device temperature is among the many factors that affect device performance — hotter chips run slower, become unreliable and can fail prematurely. The inability of the chip–package to dissipate heat is now becoming a performance bottleneck, limiting the ability to run chips at higher frequencies and also restricting the number of transistors per device. Therefore, reducing chip power consumption often makes it possible to increase performance while reducing the cost of powering and cooling the servers.

In the past, AMD engineers addressed power consumption using power analysis tools that operate at the gate and transistor levels. However, this approach is limited for several reasons. Any design changes at this late stage require re-synthesis of the design followed by an extensive verification process using a workflow with multiple tools. This substantially increases time to design closure. In addition, changes are limited because these tools optimize within the predetermined high-level architecture of the design. When the design is represented as a multitude of gates and transistors, it is also difficult to identify power hotspots at architectural or functional levels.

More recently, AMD engineers used ANSYS PowerArtist power analysis software on a processor to evaluate power consumption earlier in the design flow, and achieved an extraordinarily high level of power efficiency. By establishing a methodical approach of tracking register transfer level (RTL) power over various activity scenarios, they identified areas of significant wasted power consumption, and then addressed them through specific RTL changes.

MOVING POWER ANALYSIS UPSTREAM

Power consumption studies run on early-stage designs are limited in accuracy because key physical design elements are not completely defined. At later stages of design implementation, when power can be much more accurately estimated, changes are expensive and run the risk of delaying product introduction. AMD engineers overcame this challenge by adopting RTL as the abstraction to address power in a relative sense. RTL also provides a functional view of the design — for example, at the multiplexer and adder levels — that enables efficient power debug, in contrast to individual logic gates such as AND/OR. PowerArtist RTL power analysis runs multimillion-instance designs in minutes, which enables the designer to quickly evaluate multiple what-if scenarios. It also models physical effects such as clock-tree and wire capacitance that enable predictable accuracy for early design decisions. A typical server-class processor combines a central processing unit (CPU) with a data fabric that communicates with random-access memory (RAM). In the past, it took six to eight weeks to generate power consumption numbers based on the physical implementation for such designs, at which point the design had typically progressed to a stage where the analysis results were irrelevant. Not only did ANSYS PowerArtist trim analysis time to a single day (a reduction of 98 percent), but early findings enabled key decisions that reduced power beyond even the designer's expectations.

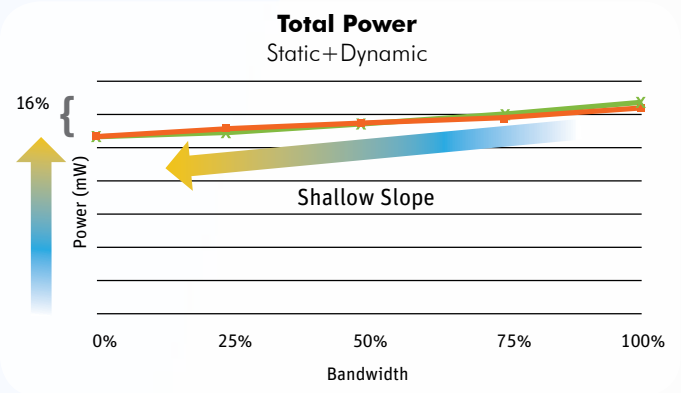


RTL Workflow

BANDWIDTH VERSUS POWER

AMD engineers established a robust methodology to address power that starts with generating the correct activity scenarios. Engineers ran RTL simulations to exercise the design from idle to the highest bandwidth, and then examined power hotspots in various sections of the design. This unique approach allowed the team to carefully examine the relative difference in power between the low and high bandwidth scenarios.

They made two keys observations. First, PowerArtist computed the power consumption in the idle mode as only 16 percent lower than the 100 percent bandwidth mode of operation, and more than 50 percent of this power was consumed by the clock distribution network alone. Second, idle power was high, primarily because many inactive blocks were subject to clock toggling.



Initial ANSYS PowerArtist simulations showed that idle power was only 16 percent less than maximum power.

ONLY CIRCUITS DOING WORK SHOULD CONSUME POWER

AMD engineers established the goal that only circuits that were doing work should consume power. They used PowerArtist to identify design elements that were continuously supplied with a clock signal, even when they were not active, and therefore provided opportunities for improvement. The RTL owners used these clearly defined opportunities to significantly reduce power in their blocks.

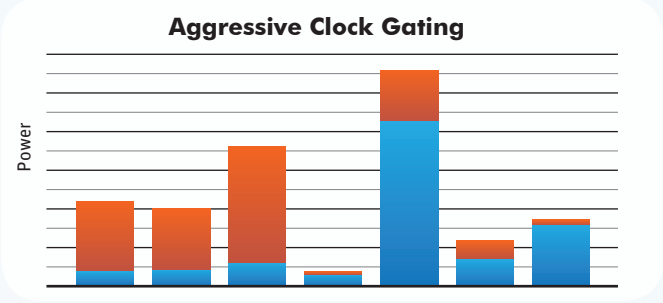
For example, PowerArtist identified numerous cases in which a multiplexer was fed by several cones of logic — only one of which was active at a time — yet all of the cones remained continuously activated. The RTL design owners added clock gates to turn off power

ANSYS PowerArtist
ansys.com/powerartist

to the inputs and corresponding logic cones that were not active. Wakeup signals were used as another effective mechanism to alert disabled logic to capture data or to respond on the next cycle, generating a one-clock cycle latency cost-per-wakeup sequence but saving significant power.

CUTTING THE CLOCK POWER

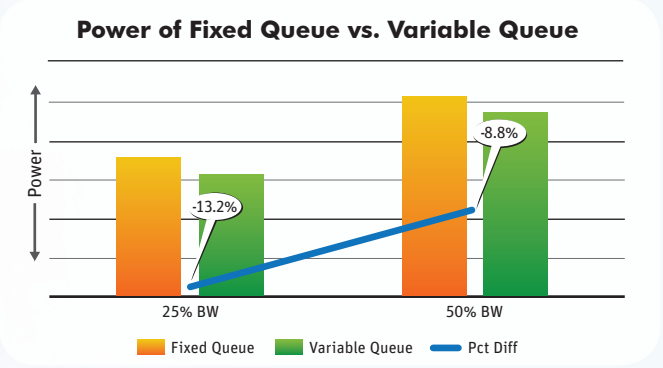
AMD engineers explored different clock-gating architectures to determine the impact on power consumption. When the inactive logic is moved farther downstream from the clock, power is increasingly wasted in the clock distribution network. Engineers created rules to group specified classes of logic so that they could shut down sections of the clock tree as close as possible to the root to maximize power savings. In some cases, wakeup time has too great an impact on performance so the logic continues running despite the resulting power inefficiency. The team labeled these as intentional inefficiencies.



Aggressive clock gating reduced idle power by 25 percent. Red shows additional power wasted when not using aggressive clock gating and blue indicates idle power with aggressive clock gating enabled.

OPTIMIZING QUEUE DEPTH

AMD engineers ran experiments to determine the impact of queue depth on power consumption and performance. They concluded that if the queue was busy, increasing its size often reduced power consumption. However, if the queue was relatively inactive, then reducing its size could be beneficial. They also added logic to adjust the queue size on the fly based on utilization. The ability to adjust queue size in this way made low-bandwidth cases more efficient and demonstrated typical advantages of approximately 10 percent power reduction.

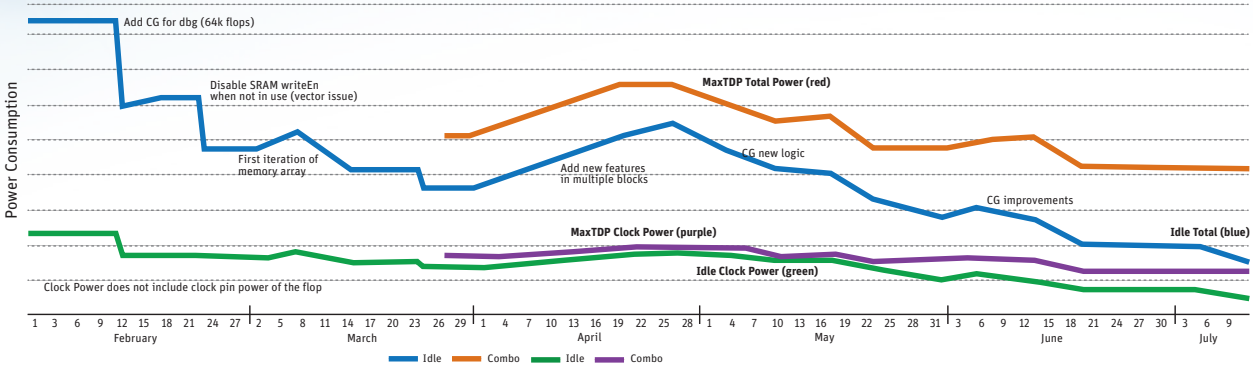


Variable queues provided power consumption improvements across the bandwidth spectrum.

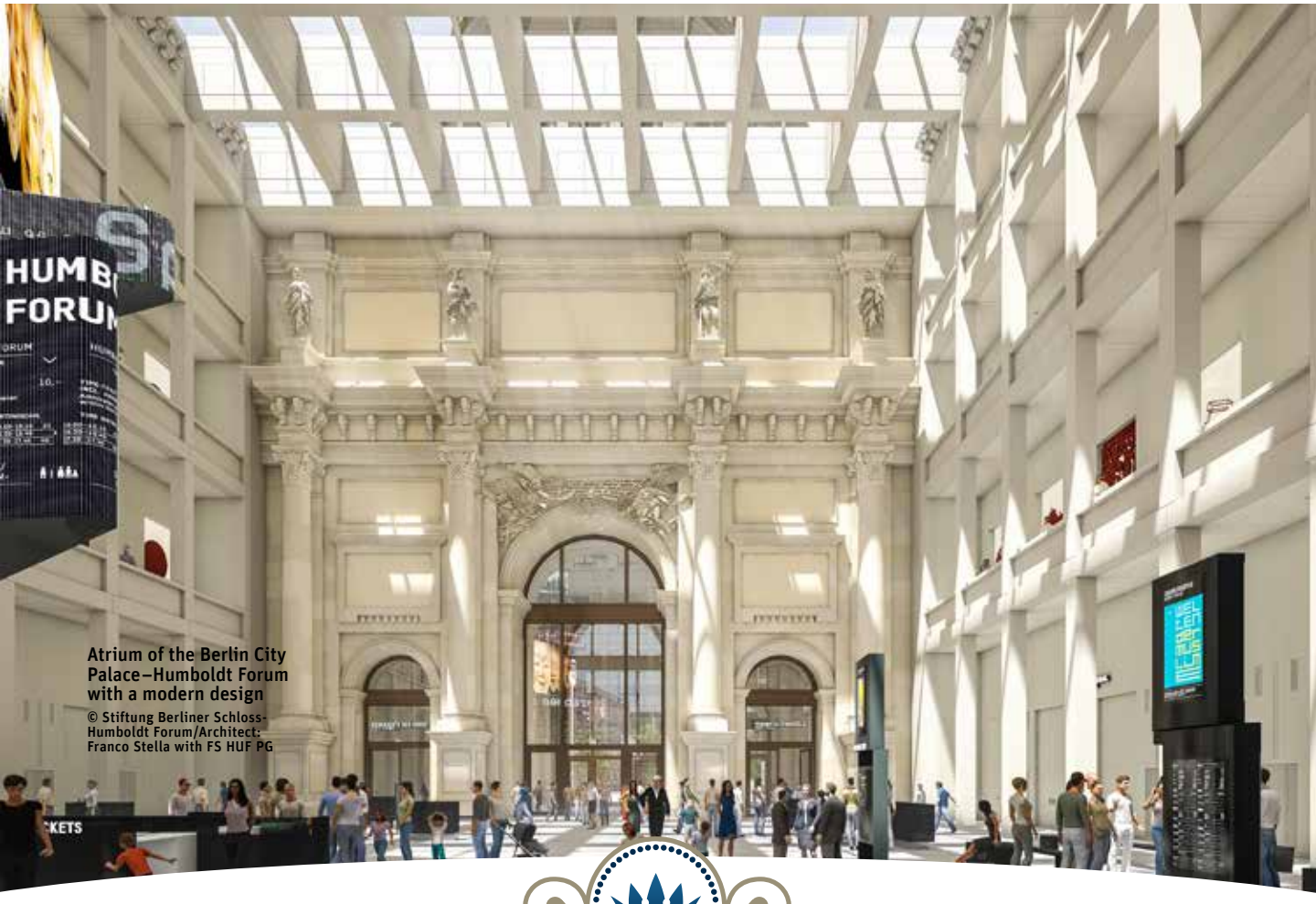
EARLY POWER ANALYSIS

Engineers reviewed the PowerArtist results, implemented the suggestions, and reran the simulations and power analysis to quickly verify the effectiveness of the suggestions. They ran weekly regressions to track power, allowing for rapid analysis and verification of modifications. Over the course of the project, the idle power was reduced by more than 70 percent. The improvements in idle power not only benefited the idle case but also created a 22 percent improvement in the maximum TDP case. The slope of the power versus bandwidth curve improved by 400 percent.

All in all, performing power analysis simulations earlier in the design flow made it possible to produce substantial reductions in power consumption, which in turn enabled performance improvements. AMD plans to integrate ANSYS PowerArtist RTL power analysis technology into its standard methods.



Power consumption was steadily reduced over the course of the project.



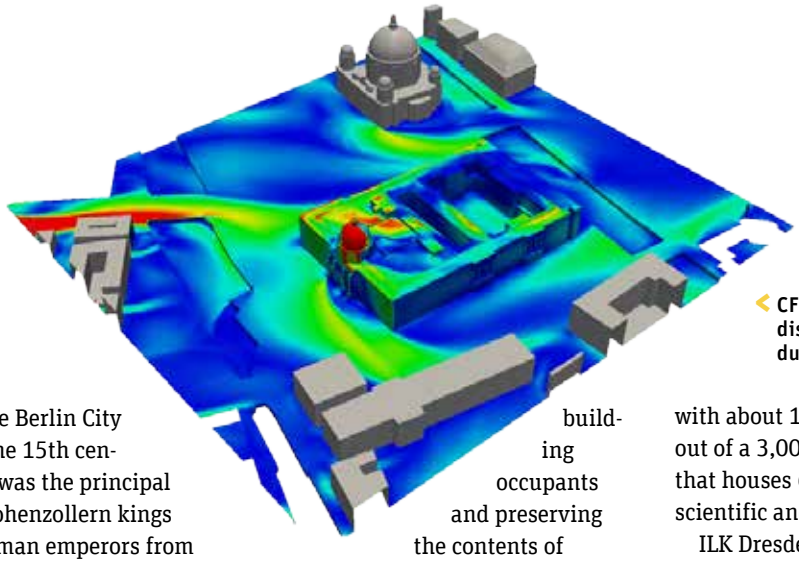
Atrium of the Berlin City Palace–Humboldt Forum with a modern design

© Stiftung Berliner Schloss-Humboldt Forum/Architect: Franco Stella with FS HUF PG

FIT FOR A KING

By **Donald Stubbe**, Project Engineer, ILK Dresden, Dresden, Germany

Designing the climate control system for the reconstruction of the historic Berlin City Palace required meeting many, often conflicting, requirements for energy conservation, human comfort, artistic preservation and cost. ILK Dresden engineers used ANSYS Fluent simulation to guide development of a design that matches all of these objectives.



CFD simulation results display pressure distribution due to wind load.

Construction on the Berlin City Palace started in the 15th century. The building was the principal residence of the Hohenzollern kings of Prussia and German emperors from 1701 to 1918. Heavily damaged in World War II and later destroyed by the East German government, today this important historic site is being rebuilt with three façades in the original design, and the fourth façade and interior in a new design by Franco Stella. The building, to be called the Berlin City Palace–Humboldt Forum, will house a museum of non-European art, a live theater, a movie theater, an auditorium and two restaurants. It is scheduled for completion in 2019.

Sustainability and energy efficiency were top priorities in the design of the building's heating, ventilation and air-conditioning (HVAC) systems. European regulations mandate that buildings constructed after 2014 consume 25 percent less energy than the previous state of the art. ILK (Institute of Air Handling and Refrigeration) Dresden was given the task of validating the HVAC system's ability to meet energy-efficiency regulations, while at the same time ensuring the comfort of building occupants and preserving the contents of the museum galleries. ILK Dresden engineers used ANSYS Fluent computational fluid dynamics (CFD) software to simulate the initial design and investigate alternatives. They proposed changes in the design that will provide 10 to 20 percent improvements in comfort and energy efficiency while maintaining the energy efficiency and cost of the original design.

INTERDISCIPLINARY EXPERTISE

Triggered by the special challenges involved in this application, the building's owner asked ILK Dresden to develop an optimized equipment configuration to meet the requirements. ILK Dresden was selected for this project because of its interdisciplinary expertise in many fields of research and development, including heat and mass transfer, thermodynamics and fluid mechanics, refrigeration and refrigerating equipment, heat-pump technology and solar energy. The market-driven nonprofit organization with about 150 employees operates out of a 3,000-square-meter facility that houses 60 experimental and 25 scientific and technical labs.

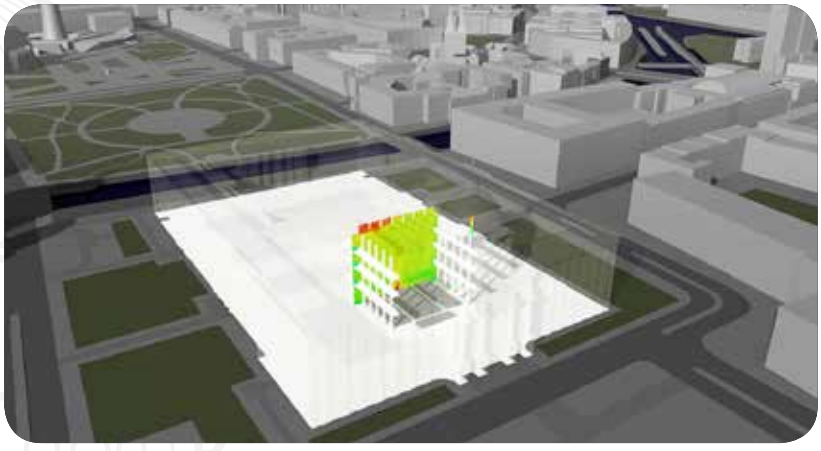
ILK Dresden chose ANSYS Fluent for this project because the software has been proven and validated beyond other alternative solutions to the point that ILK engineers and its clients have complete confidence in the accuracy of its results. The software package also provides the full range of physical models needed to handle virtually any building simulation challenge.

The simulation incorporated room air flow coupled with heat transfer and a solar radiation model. The engineers used climate data provided by the Deutscher Wetterdienst as boundary conditions for the simulation.

ILK Dresden engineers collaborated with the Dresden University of Technology (TUD) to make wind-tunnel measurements on a small scale model to validate the simulation of pressure

Northwestern side of the Berlin City Palace–Humboldt Forum matches the original historical building.

> “Sustainability and energy efficiency were top priorities in the design of the building’s HVAC systems.”



Pressure distribution in isoplanes

distribution due to wind load around the building. ILK Dresden engineers modeled the air distribution system, including the supply air volume, velocity and temperature, as well as the physical locations of the supply diffusers and air-return registers. The manufacturers of each of the fans delivered detailed performance specifications, including fan characteristics that provided airflow as a function of pressure.

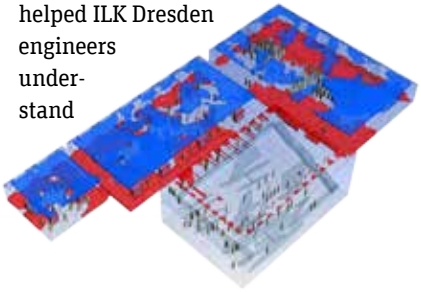
Engineers performed numerical simulations of the airspace in various exhibition, performance and meeting rooms for different usage scenarios and ambient conditions. For example, ILK Dresden engineers simulated one exhibition hall with different numbers of people as heat sources. They paid special attention to worst-case scenarios, such as the hottest and coldest days that could be expected in the Berlin climate.

The CFD simulation generated complete airflow patterns, including velocities and distributions of variables such as pressure and temperature, at all areas in the building. For spaces where the most critical concern is human comfort, engineers carried out simulation of room temperature, humidity and air velocity with particular attention to the draft risk as outlined in ISO 7730. ANSYS CFD provided graphics showing temperature mapped as color contour charts across horizontal sections at various levels above the floor. These results not only made it possible to determine the cooling performance of the design but also provided information that helped in understanding the reasons behind the design’s performance.

ITERATING TO AN OPTIMIZED DESIGN

While the initial design met energy efficiency requirements, it did not perform as well as desired in the areas of human comfort and artwork conservation. ILK Dresden engineers performed design exploration using ANSYS Workbench to increase performance in areas in which the original design was weak. First, for each parametric study, engineers determined input and output variables. For example, where air velocity was too high in a particular space, they defined the location, dimensions and mass flow rate of the air supply as inputs, and air velocity in the critical space as the output. Then they assigned design points with different values for input variables.

When engineers clicked “Update All Design Points,” ANSYS Workbench automatically triggered Fluent to solve each of the design points. The values of the variables were displayed for each design point in the table. These tables helped ILK Dresden engineers understand the sensitivity of the output to each of the input variables. This information was used in creating additional design iterations.



Constant values of relative humidity in selected areas of the building

CFD simulation played a key role in helping ILK Dresden engineers optimize the many, and often conflicting, requirements of the climate-control system. Engineers are confident that the palace will meet all requirements, including energy conservation, comfort, artistic preservation and costs.

Ventilation and Comfort Modeling
ansys.com/comfort

All Mixed Up

Aditya Birla Science & Technology Co. Pvt. Ltd. studied and developed an improved impeller for the mixing tank used in fiber manufacturing. The proposed design provides five times better mixing for solid suspensions using 12 percent less power. Simulation helped the engineering team make design decisions that balance mixing performance and power draw.

< Improved impeller design

By **Manoj Kandakure**, Lead Scientist, Process Engineering and Simulation, Aditya Birla Science & Technology Co. Pvt. Ltd., Maharashtra, India

Viscose staple fiber (VSF) is a man-made biodegradable fiber with characteristics similar to cotton that is used to produce fabric (often called rayon) for clothing and other purposes. VSF is produced by dissolving a solid-phase wood pulp slurry in caustic soda and forcing the solution through tiny holes in a metal cap. It emerges as filaments that unite to form a continuous strand that is solidified by passage through a liquid or heated air. Mixing the pulp slurry and caustic soda solution consumes time and electrical power, so manufacturers of this fiber can save costs and production time with more efficient mixing. Aditya Birla Group is the world's largest producer of VSF. The engineering team at Aditya Birla Science & Technology Co. Pvt. Ltd. used computational fluid dynamics (CFD) to study and develop an impeller that would increase mixing efficiency and reduce power consumption.

SIMULATION OF VSF PRODUCTION PROCESS

With revenues of about $41 billion in 2014, Aditya Birla Group is also the world's largest producer of carbon black (in addition to VSF) and a leader in aluminum and copper production, branded clothing, mobile communication, life insurance and grocery stores. The group's research arm, Aditya Birla Science & Technology

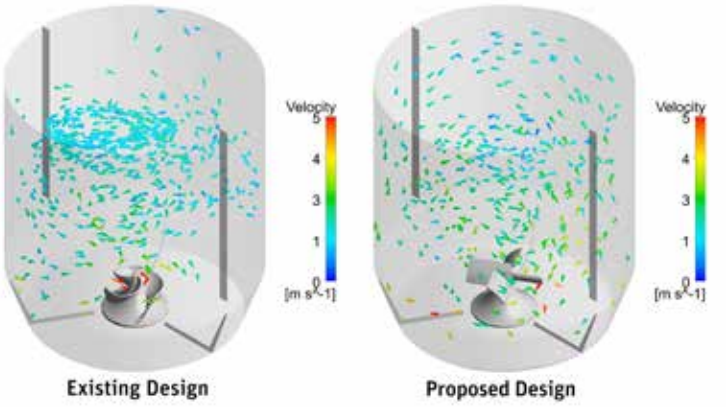
> "Aditya Birla Group used CFD to study and develop an impeller that increased mixing efficiency by a factor of more than five and reduced power consumption by 7 kW."

Co. Pvt. Ltd., works with the various business units to improve process performance.

The first step in VSF production is mixing of caustic soda and wood pulp, generally in a three-to-one ratio, in a mixing tank. The mixing tank is cylindrical with a height-to-diameter ratio of approximately 1. A pyramid-shaped impeller is located at the bottom of the tank.

The team began by performing steady-state multiphase simulation of the existing mixer configuration with ANSYS CFX CFD software. The tank was divided into about 1.7 million unstructured tetrahedral elements. A fine mesh was generated near the impeller and a coarse mesh was used in the rest of the tank. The team employed the frozen rotor mixing model to accommodate impeller motion, and utilized the Euler–Euler inhomogeneous multiphase model to simulate the multiphase liquid–solid mixing nature of the system. The standard k-ε turbulence model accounted for the turbulent nature of the mixing process. The reactions taking place inside the tank were not modeled.

For the simulation, engineers assumed that the tank was initially filled with a solid slurry and liquid solution with a 0.42 density ratio. Because the liquid solution is the heavier component, it occupies the bottom of the tank at the beginning of the simulation. The mixing value was defined as the percentage of tank volume having a liquid volume fraction of 0.65 to 0.85. Engineers integrated pressure on impeller blades as determined by CFD calculation to obtain the torque, which in turn was used to calculate power consumption.



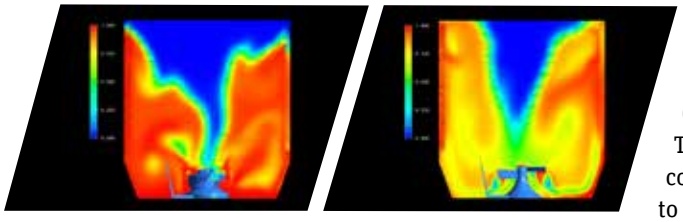
^ Mixing was improved with a reduction in power consumption.

SIMULATION OF EXISTING MIXER

Case I used the existing pyramid-shaped impeller rotating at 500 rpm with a diameter-to-tank-diameter (D/T) ratio of 0.258 and a centerline-to-tank-height (C/T) ratio of 0.17. C/T is significant because the liquid level in the tank must be kept above the impeller level to avoid splashing of solid slurry against the tank walls. Any increase in impeller height reduces the amount of slurry that can be drained from each batch. The normalized velocity vector plot shows that this impeller creates a good deal of circulation in the bottom of the tank, but there is very little suction in the top half of the tank. The simulation generated a liquid volume fraction profile showing a mixing value of only 11.26 percent, and predicted 60 kilowatts (kW) of power consumption.

ITERATING TO AN IMPROVED DESIGN

The team further evaluated other possible impeller designs in an effort to improve the mixing operation performance. To iterate to an optimal mixer design, they needed to trade mixing performance for power consumption. The final design, Case VI, used a curved-blade impeller placed near the bottom of the tank while reducing the rotational speed from 500 rpm (Case 1) to 350 rpm. The mixing value for this configuration was increased to 63 percent, which is 5.6 times better than Case I, while the power consumption decreased to 52.6 kW, even better than Case I. The impeller modeled in Case VI was found to



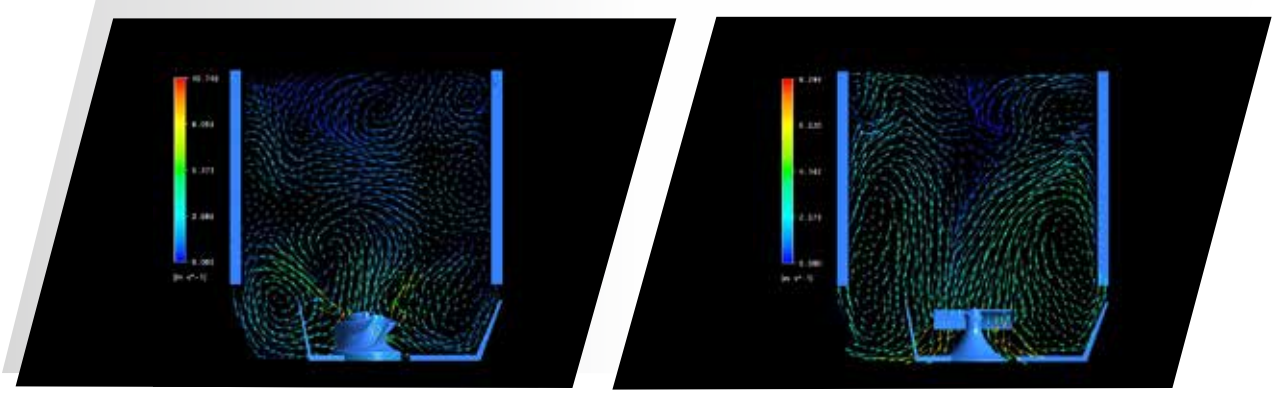
^ Liquid volume fraction profile throughout tank for initial (left) and final design (right). In the initial design, the top zone is rich in particles; there is poor mixing of the particles with the liquid and insufficient suction from the impeller to pull particles to the bottom. The improvement for the entire domain is greater than is shown in this single 2-D plane.

	Case I (initial configuration)	Case II	Case III	Case IV	Case V	Case VI (final)
Changes Made	Pyramid-shaped impeller	Pyramid-shaped impeller. Pitched-blade down-flow turbine.	Curved-blade down-flow turbine. Higher blade angle.	Curved-blade impeller along with cone.	Curved-blade impeller placed closer to bottom of tank.	Curved-blade impeller with smooth cone profile for better mixing. Reduced dead zones.
Effect of Change Over Previous Case		More suction in the top half of the tank. Better circulation near the tank bottom.	Stronger suction and even larger recirculation loops. High power consumption.	High power consumption.	Amount of slurry that can be drained from every batch equal to Case I. Improved flow profile and reduced dead zones. High power consumption.	Better mixing than Case 1. Reduced power consumption over Case 1.
Mixing Value (%)	11.26	30.71	77.6	68.28	67.1	63.0
Power Consumption (kW)	60	74.3	118.5	150	157	52.6

be the most efficient design that provides maximum benefits in terms of mixing efficiency and power consumption.

The changed design is expected to reduce batch time, improve throughput and reduce process costs. When compared to previous impeller design, the proposed design has better suction, good mixing value, a reduced amount of unmixed solids and lower power consumption. This is just one of studies conducted in the past decade in which simulation has helped Aditya Birla Group to identify and improve productivity and reduce costs across its manufacturing businesses.

Mixing
ansys.com/mixing



Velocity vectors for initial (left) and final design showing improved mixing performance. In the improved design, the curved cone and the impeller rotate as a single body, achieving better mixing. The proposed impeller generates stronger downflow than the existing design.

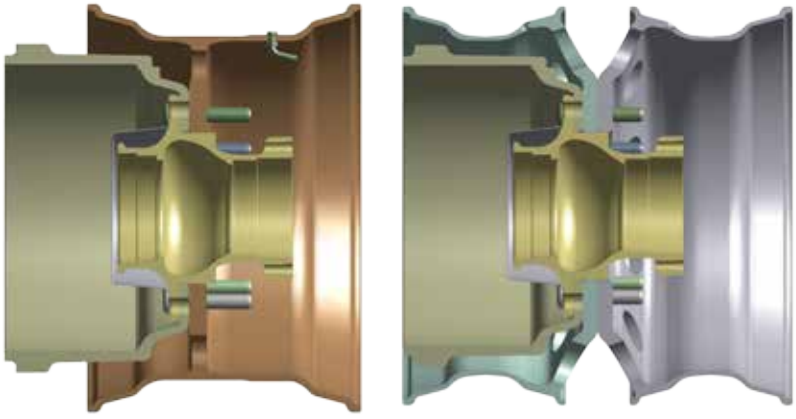
BIG WHEEL

The North American commercial trucking industry is challenged by many factors, including the continual need to be more efficient by carrying more load at lower cost and higher fuel economy. One way some trucking fleets address this need is to utilize wide-base wheels that offer weight savings and lower rolling resistance compared with traditional dual-wheel configurations. Accuride has enhanced the savings that truckers can achieve by using ANSYS simulation tools to reduce the weight of its Duplex® wide-base wheels by more than 23 percent in recent years.

By **Mike McLeod**,
Senior Project Engineer,
Accuride Corporation,
Henderson, USA

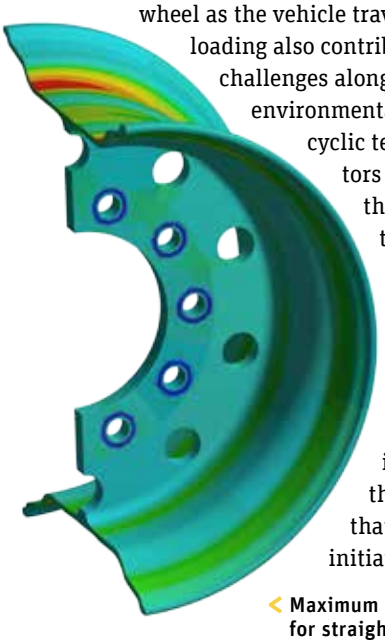
Most commercial trucks ride on sets of two 22.5-by-8.25-inch wheels with two tires on both drive and trailer axles. Commercial truck owners needing weight and fuel economy savings are considering the benefits of a single 22.5-by-14.0-inch wide-base wheel instead of the traditional dual-wheel configuration. Wide-base wheels reduce rolling resistance by about 30 percent, resulting in up to 10 percent fuel savings. Wide-base wheels are also substantially lighter than the two dual wheels that they replace. Further weight savings come from using one wide-base tire compared with two of the standard size. Accuride has provided truckers with further savings by using ANSYS simulation tools to reduce the weight of their forged aluminum wide-base wheels from 71 pounds to 55 pounds in current designs — an additional savings of 128 pounds on a typical tractor trailer.

"Commercial truck owners needing ***weight and fuel economy savings*** are considering the benefits of a single wide-base wheel."



^ New wide-base wheels (left) and a conventional set of two wheels (right)

Commercial truck wheels are typically constructed from steel or forged aluminum. The materials and processes are relatively basic, but the minor details of the material and processes make all the difference in achieving a long-lasting and durable product. To ensure that these wheels are reliable, engineers are faced with structural challenges like metal fatigue and fracture, including standard cyclic fatigue, fretting fatigue, weld fatigue and crack growth. Primary loads on wheels come through the tires, which exert very high axial and radial loads just from tire inflation of up to 131 psi. Additionally, each 14-inch-wide wheel must carry 12,800 pounds of load generated from the vehicle weight, and cycle around the wheel as the vehicle travels. Side and impact loading also contribute to metal fatigue challenges along with eventual environmental factors. Accelerated cyclic testing and safety factors can more than double the load for which the wheel must be designed. Predicting fatigue life is very challenging because the relationship between fatigue life and material stresses is nonlinear, and there are many factors that influence crack initiation. Often small changes in design or processing can have a significant influence on actual wheel performance.



< Maximum principal strain contour for straight rolling using analytical tire load model

ENSURING RELIABILITY

Accuride does not release wheel designs for use until high test standards are met. Commercial wheel designs are generally tested against SAE standards. These test standards include straight rolling of a wheel and tire assembly, and a rotating bending test of just the wheel at elevated loads to a specified number of cycles. Due to the potential variability in testing, Accuride significantly exceeds industry standards to ensure reliability. Physical testing is very expensive and time-consuming. For every prototype design that is tested, custom-built tooling and manufacturing processes must be developed to generate sufficient samples for design validation. Prototype builds and testing can take weeks and even months. If all samples do not pass the qualification testing, then a new design iteration is started and the clock is reset for samples and testing, thus prolonging product release. Therefore, minimizing design iterations is critical to managing development costs and time-to-market goals.

As part of the design process, it is critical to validate that the manufacturing process can repeatedly create the same product with minimal variation. The potential variations that must be controlled include both geometry and all aspects of the manufacturing process. The manufacturing process has a significant influence on actual wheel performance. These processing effects must be considered during the early design phase. Failure to attain manufacturing control will mean process changes or, potentially, product redesign.

> **"By taking advantage of the *ANSYS Workbench environment*, workflows can be set up so that only the input *geometry needs* to be changed for each analysis iteration."**

PREDICTING FATIGUE PERFORMANCE

Continually improving a safety-critical component subject to fatigue loading requires advanced knowledge of the product loading and the effects of manufacturing processes. An Accuride engineer starts by constructing standardized finite element models (FEM) using ANSYS Mechanical within ANSYS Workbench to provide an initial evaluation of stresses and strains for different load cases. Integration with a parameterized CAD model makes it easy to evaluate multiple designs at this preliminary stage. As the design progresses, the analyses begin to include predictions of crack initiation using ANSYS nCode DesignLife. The ANSYS stress and strain predictions flow directly into a custom fatigue analysis template setup in nCode DesignLife. The nCode DesignLife component is critical because fatigue problems are governed by the stress history and other factors besides the maximum stress values.

After preliminary design work is complete using fatigue life predictions, Accuride engineers may begin more advanced optimization methods to further reduce weight while trying to maintain minimum levels of predicted fatigue performance. The optimization process is similar to the preliminary design work except that ANSYS Workbench systems are parameterized, including input and output variables. Tables of multiple parameter combinations are set up and solved in a batch mode to evaluate the sensitivity of each parameter. Once key parameters are identified, ANSYS DesignXplorer tools can be used to further refine the design using design of experiment (DOE) methods.

^ Accuride 22.5 x 14.0 Duplex® wide-base wheel

The Accuride wheel analysis process has multiple phases. Evaluating and optimizing different designs potentially could take a significant amount of time. By taking advantage of the ANSYS Workbench environment workflows can be set up so that only the input geometry needs to be changed for each analysis iteration. Additionally, ANSYS ACT enables Accuride to further enhance the analysis process so that more time is spent reviewing results than building models.

The final design proposal is validated with simulation focusing on model quality and design details. Advanced meshing controls and quality checks ensure mesh quality in critical areas. Submodeling techniques enable even higher levels of detail at locations such as contact edges.

RESULTS

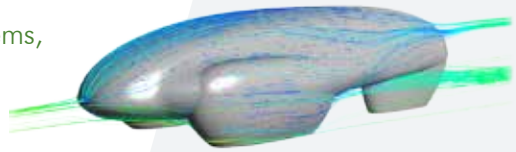
Using ANSYS and the analysis methods developed in-house, Accuride engineers have reduced the weight of their standard 22.5-by-8.25-inch forged aluminum wheel by 25 percent and the 22.5-by-14.0-inch wide-base Duplex® design by 23 percent in recent years.

Besides meeting customer lightweighting goals, thereby contributing to increased fuel economy and larger loads, weight reductions help Accuride maintain competitiveness by offsetting increasing production costs with raw material savings.

As the complex structural and design challenges of commercial truck wheels continue to increase, Accuride engineers are developing even more advanced analysis methods using the capabilities of the ANSYS Workbench environment and integration with specialty codes such as ANSYS nCode DesignLife. With these new tools, Accuride will continue to improve its design methodology to develop innovative and reliable truck wheel technology at an even faster pace.

THE FUTURE OF ENERGY INNOVATION

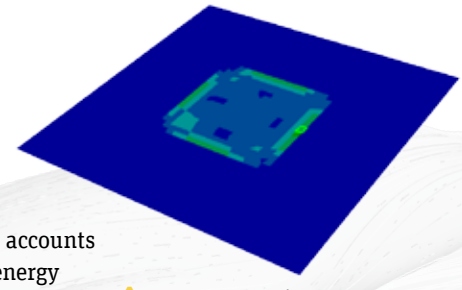
Commercial companies, research institutes and academic organizations are actively pursuing goals related to reducing power consumption, decreasing emissions and creating new methods to utilize existing energy resources. At the same time, engineering students around the world are honing their skills and developing new vehicles, often as part of team competitions, that use the latest technology to transform our method of travel. These students create solutions to real-world problems, and will become the next generation of workplace leaders to engineer energy innovation.



By *ANSYS Advantage* Staff

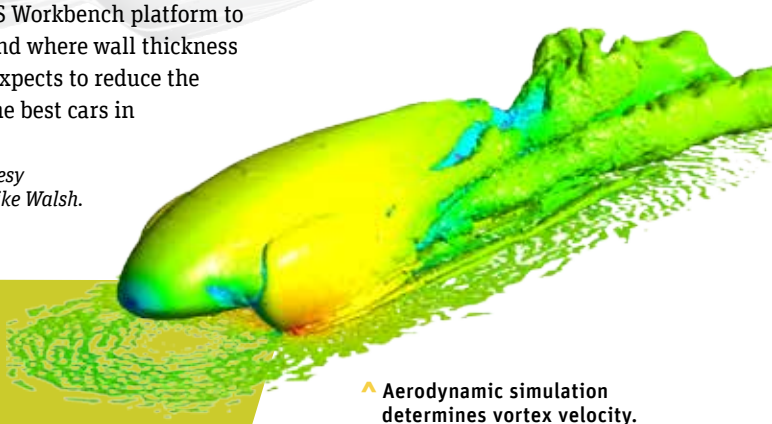
1 THE FUEL-EFFICIENT GEEC

The Galway energy-efficient car (better known as the Geec) is a battery-electric eco-car designed and built by engineering students from **National University of Ireland Galway** to complete in the Shell Eco-marathon Europe. The team uses ANSYS CFD to minimize aerodynamic drag, which accounts for about one-third of all energy consumption in a highly optimized car. Students leverage ANSYS Mechanical to support lightweighting the design of various components. To develop their next-generation car with a monocoque design, students count on ANSYS Mechanical and ANSYS PrepPost within the ANSYS Workbench platform to determine where additional material would be needed, and where wall thickness can be minimized to save weight. The NUI Galway team expects to reduce the car's weight by about 10–15 kg as it attempts to match the best cars in the competition.



^ ANSYS Composite PrepPost simulation to determine failure for rear mounting point

The Geec is supported by CADFEM UK & Ireland. Information courtesy Nathan Quinlan, Maeve Duffy, Rory Monaghan, Sean Scally and Mike Walsh.



^ Aerodynamic simulation determines vortex velocity.

> "The NUI Galway team expects to reduce the car's weight by about 10–15 kg as it attempts to match weight with the best cars in the competition."

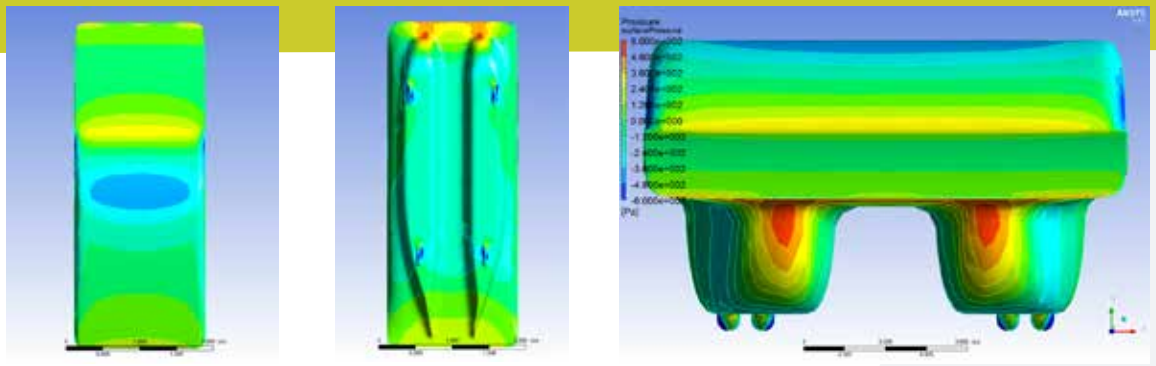
v Eco-car team from NUI Galway

2 SOLAR-POWERED REVOLUTION

The **University of Minnesota Solar Vehicle Project** is one of many teams competing to harness the power of the sun to drive wheels, with the premise that efficiency is critical in all aspects of a solar-powered car. Using ANSYS Mechanical, the team ensures that the car has appropriate material distribution for durability and safety, while keeping the car as light as possible through excess material removal. Students are working on a new multiperson car that will be introduced in late 2017. They employed ANSYS software in the early design stages to maximize efficiency for almost every component. Simulation gives the team confidence that the car will perform as expected so that they can focus on efficiency in all aspects of the design.

Information courtesy David Sorenson.

The University of Minnesota Solar Vehicle Project two-seater cruiser-class car and team



TOP BOTTOM FRONT

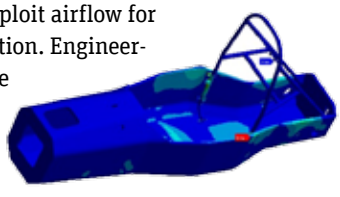
Surface pressure aerodynamic simulations

3 DESIGNING WITH NEW MATERIALS

MUR Motorsports team

A major goal in 2016 for the **University of Melbourne Australia Motorsports team** was to reduce the mass and weight of its FSAE car. Lightweight solutions were required for many of the car components, and ANSYS structural software was vital to achieving this goal. With a new monocoque design for the 2016 car, ANSYS Composite PrepPost allowed the team to check for the presence of internal carbon fiber delamination in response to excessive loads. The aerodynamics sub-team used fluid dynamics simulation with ANSYS CFD to gain insight into how to exploit airflow for downforce production and drag minimization. Engineering simulation made it possible to estimate fuel consumption (by determining the drag coefficient) so that the team could determine trade-offs between fuel efficiency and other areas affecting competition.

Information courtesy James Hancock.



Stress simulation on chassis

> “These tools simplified and ***streamlined the process*** of composite layup design iteration to reduce mass and maximize stiffness.”

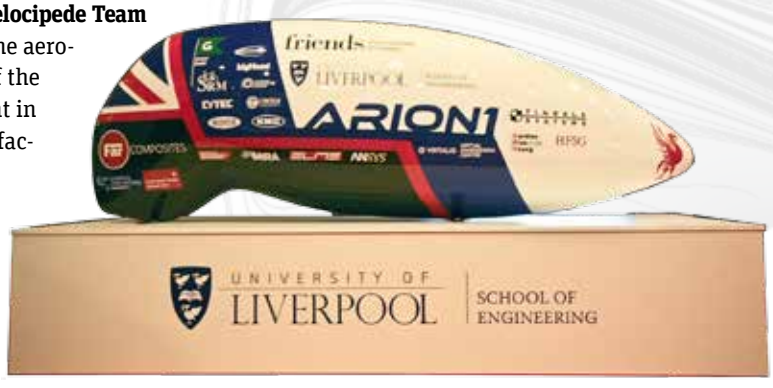
4 THE POWER OF PEOPLE

At the World Human Powered Speed Challenge, students from around the world compete to design and power a bicycle using only energy generated by the rider’s body. At 80 mph, 70 percent of the retarding forces acting upon a high-speed bike result from drag upon the carbon-fiber monocoque shell, so aerodynamic optimization is critical to a successful world record attempt. Employing the easy-to-use ANSYS CFD software featuring the transition SST turbulence model, the **University of Liverpool Velocipede Team** optimized the aerodynamics of the shell upfront in the design process so that it could be manufactured in time for testing and competition.

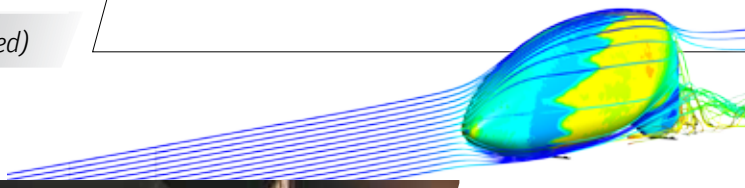
^ ANSYS mesh of ARION1

The University of Liverpool Velocipede (ULV) Team was sponsored by Rathbone Investment Management and supported by Friends of the University of Liverpool.

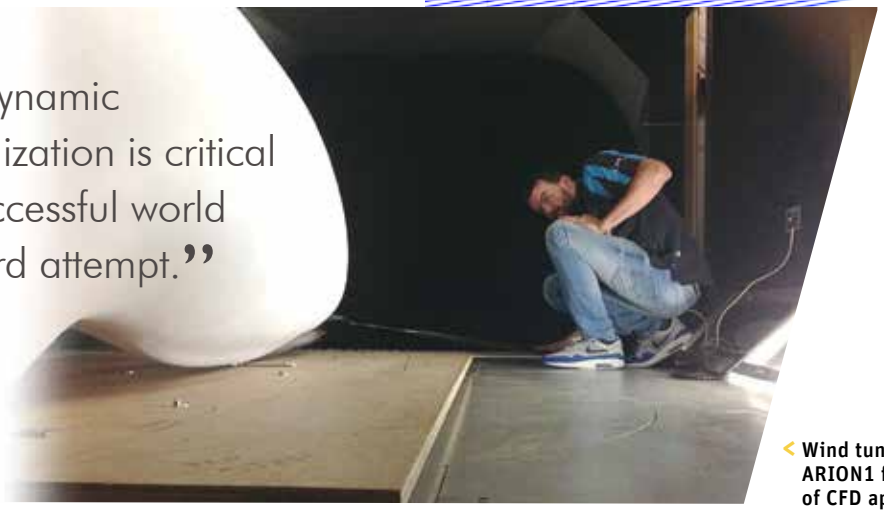
Information courtesy Toby Dafforn-Jones, Michael Francis Kelly and Mark D. White.



^ ARION1 human-powered vehicle

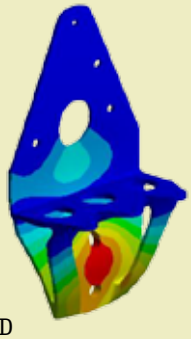


"Aerodynamic optimization is critical to a successful world record attempt."



< Wind tunnel testing of ARION1 for validation of CFD approach.

5 DASHING THROUGH THE SNOW



The SAE (Society of Automotive Engineers) Clean Snowmobile Challenge tasks college and university students to modify a snowmobile to make it acceptable for use in environmentally sensitive areas. A team of students from **Rochester Institute of Technology** (RIT) took on the challenge. Much of the design work involved modifying the internal combustion engine to reduce noise and decrease emissions. The team used ANSYS SpaceClaim to prepare CAD models for simulation with a special-purpose engine design program, and ANSYS Forte to simulate tailpipe emissions during the design stage, even before any camshafts were fabricated. Students used ANSYS Mechanical to ensure that a new, lighter part would work safely and reliably, while meeting competition rules. The team will employ the software for further development of emission control strategy.

Information courtesy Eric Oswald.

"ANSYS Forte allowed the team to simulate tailpipe emissions during the design stage."

6 AROUND THE WORLD IN 80 DAYS... ON A MOTORCYCLE

A team of 23 students at the **Eindhoven University of Technology** in the Netherlands has designed and built the Storm Wave — the first long-distance electric touring motorcycle. The team will attempt to ride around the world (26,000 km) in 80 days using only the existing electric power grid. By designing the motorcycle with batteries that can be swapped out, the team reduced the weight of the bike so that it can travel farther on a single charge. In addition, students used ANSYS CFD to optimize the aerodynamics of the motorbike as much as possible. The around-the-world tour will show the promise of electric vehicles.

Information provided by Bas Verkaik.

The STORM team from TU Eindhoven

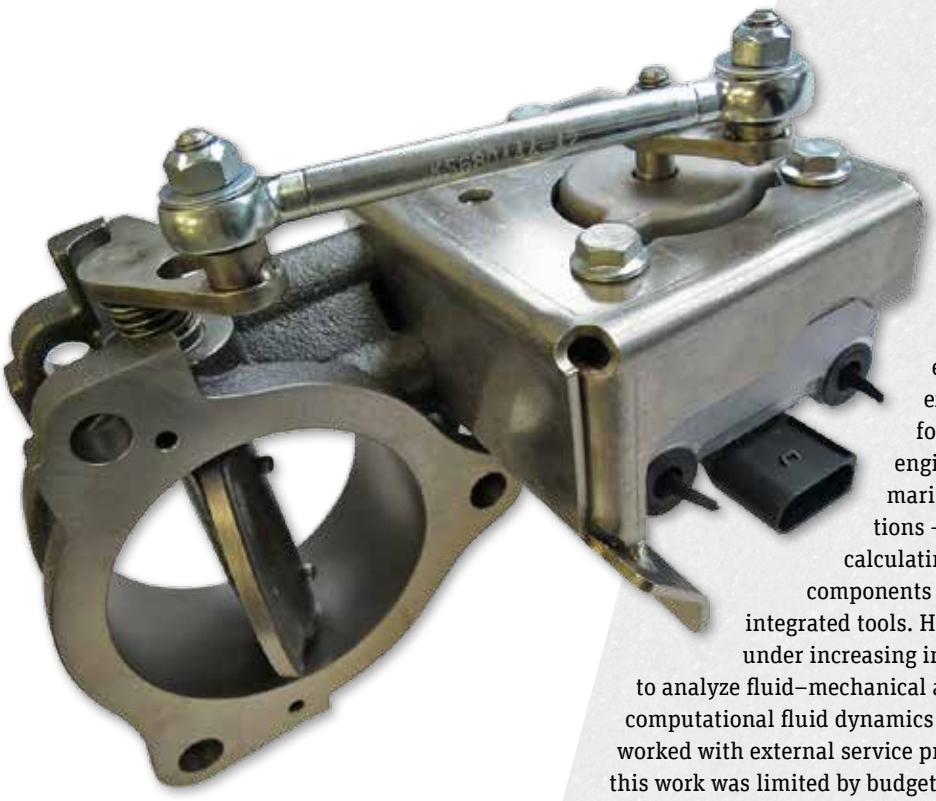
The Storm Wave with removable battery cells

For more information on these teams, see the Web Exclusives for this issue of *ANSYS Advantage*.

TAKING OFF THE **BRAKES** TO **PRODUCT DEVELOPMENT**

Moving from CAD-centric design to the world of high-fidelity simulation was made easy for designers at a leading automotive and marine supplier by using ANSYS AIM. Access to multiphysics tools has reduced costs, improved productivity and provided insights that were previously unavailable.

By **Richard Krellner**, Construction Manager, and **Tobias Dörres**, Development and Test Engineer, Klubert + Schmidt GmbH, Pottenstein, Germany



^ Klubert + Schmidt exhaust control flap for commercial vehicles

CAD designers at Klubert + Schmidt GmbH — a reliable developer and supplier of exhaust flaps and hot-side exhaust gas recirculation valves for heavy- and medium-size engines used in on-road, off-road, marine and industrial applications — had, for several years, been calculating the mechanical loads of components and subsystems using CAD-integrated tools. However, the design team was under increasing internal and external demand to analyze fluid–mechanical aspects of their products with computational fluid dynamics (CFD). In the past, they had worked with external service providers, but contracting out this work was limited by budget, the time required to explain the task to an outside engineer and effectively communicating the meaning of the results obtained.

The company started looking for CFD software and met with CADFEM, an ANSYS elite channel partner. CADFEM recommended the new multiphysics simulation software ANSYS AIM, which integrates analysis of fluid flow, structural mechanics and thermal behavior in a single software tool.

However, the team at Klubert + Schmidt was very skeptical. Designers initially believed that this comprehensive tool was oversized for their company, but they were encouraged that AIM provided an integrated solution that enabled designers to work with native CAD data.

ANSYS AIM: Try It Now for Free
ansys.com/tryAIM

FROM CAD TO SIMULATION MODEL

After a demonstration, the designers were impressed by ANSYS SpaceClaim, the integrated geometric modeler within ANSYS AIM. SpaceClaim enables

designers to apply the changes required for simulation to already existing CAD models. Often, preparing a CAD model for simulation involves numerous modifications. Transferring an optimized geometry from the simulation tool back to the CAD modeler was, in many cases, problematic even with native CAD models. AIM with SpaceClaim overcomes these difficulties.

Klubert + Schmidt decided to use ANSYS AIM because of the ease of use and consistent user interaction for all physical domains. Another decisive factor was the wide distribution of ANSYS software and the attractive value/price ratio of AIM.

In September 2015, the company introduced ANSYS AIM simulation software by training six designers. The company chose training based on a real development project — a Klubert + Schmidt engine brake. Because this product had already been measured on the test bench, the simulation process and results could be, and were, validated.

After the training, a test engineer focused on flow simulations and temperature field calculations — certain products reach up to 700 C — so that this user could build up expertise in the use of ANSYS AIM. Flow analysis was new to him and he was eager to learn to use CFD results as a basis for further calculations.

DETAILED UNDERSTANDING OF PRODUCT BEHAVIOR

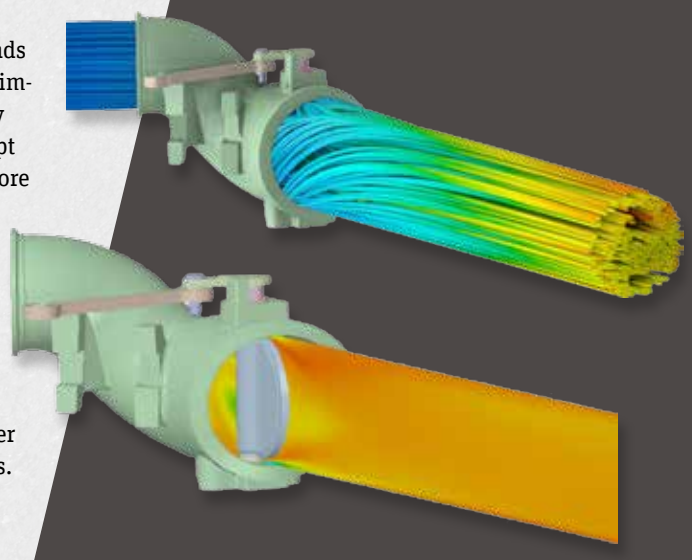
Later, for a new customer project, the developers at Klubert + Schmidt were able to present technical details well before the design freeze. In addition to the first prototype demonstration, the customers were impressed with ANSYS AIM simulation results that vividly illustrated the basic functionality of the new product technology. This extended use of simulation emphasized the company's technical and innovative competence.

The CFD simulations performed so far have helped the developers understand details of the behavior of their products that were previously unknown. Bench testing variants of components had been a common practice. But, for some of the measurements, the team was unable to determine a sound physical explanation. With simulation, this is much easier and faster. By not relying on a single measurement, unexpected behavior often can be better understood. Problems that used to be discovered only on the test stands during the final stages of the development can now be eliminated almost completely with early simulation. Not only are functional impairments identified early in the concept phase, initial vulnerabilities can be fixed, leading to a more robust design. Additional parameter studies help to identify the limits of safe operation.

Multiphysics simulations with ANSYS AIM are becoming a part of the standard development process at Klubert + Schmidt. Simulation will be a future element of the quality gates through which development passes to enter the next phase of the project. Many test rig setups and some physical prototypes are no longer needed, since simulations provide the necessary answers. This saves the company time and money. In addition, simulation allows the design team to check ideas that were previously too costly or too difficult to follow up.

Exploring new possibilities with multiphysics simulation enables Klubert + Schmidt to better meet the ever-rising demands for product development and innovation.

> "Klubert + Schmidt decided to use ANSYS AIM because of the ease of use and consistent user interaction for all physical domains."



Flow and temperature simulation of a typical exhaust flap

ENSURING A GOOD BOND

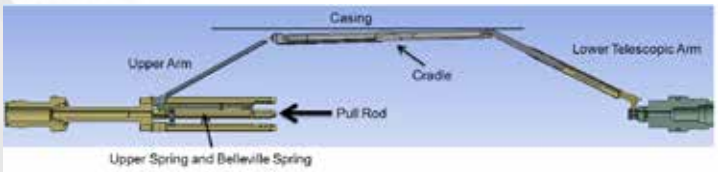
New **Baker Hughes** electromagnetic–acoustic transducer (EMAT) technology provides a more accurate method of assessing the bond between the cement and casing in an oil well. Baker Hughes engineers used ANSYS Mechanical to maximize the reliability and reduce the time to market for this technology by 20 percent; the key was designing the complex linkage assembly to successfully deploy the transducers inside the wellbore for measurements.

By **Saeed Rafie**,
Senior Reliability Manager,
Baker Hughes
Wireline Services,
Houston, USA

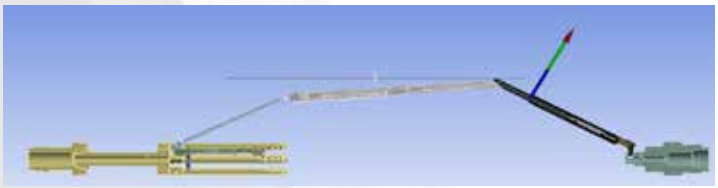
One of the most critical steps in preparing a well for production involves pumping cement into the annulus between the drill bore and the casing — the tubing that is set inside the drilled well to protect and support the well stream. Cementing the well provides a number of benefits, the most important of which is isolating the surrounding geological environment from flowing production fluids. The integrity of the cement, and particularly the strength of the bond between the cement and the casing, is critical for delivering the highest-possible level of protection to ensure reliability and safety of the well.

Traditionally, measuring cement bond integrity relied on acoustic transducer instruments that are lowered into the well. The instruments sent an acoustic energy (pulse) into the casing. The amount of acoustic energy that leaked into the cement indicated the cement's compressive strength. A solid bond resulted in less energy being reflected back to the instrument. In challenging conditions, such as when the cement becomes contaminated with borehole mud or when low-density cement is used, acoustic impedance of the cement is decreased and traditional acoustic-based cement evaluation is not possible. However, using a new capability called the Baker Hughes Integrity eXplorer™ wireline cement evaluation service, electromagnetic–acoustic instruments are lowered into the casing to assess the integrity of cement bonds. The acoustic shear wave generated by the electromagnetic-acoustic transducers (EMATs) provides a new foundation for cement evaluation by responding to the cement's shear modulus, which is a true indicator of solid cement behind the casing.

When developing this new instrument, engineers needed to design the linkage, a mechanism that carries the sensors downhole so the device would open to deploy the instruments and then close to move the linkage to a new location. Repairs cannot be performed downhole, so if the linkage does not operate properly when deployed, it must be brought to the surface for repairs. This is an expensive process. In a worst-case scenario, the linkage becomes lodged in the hole, which requires even more expensive

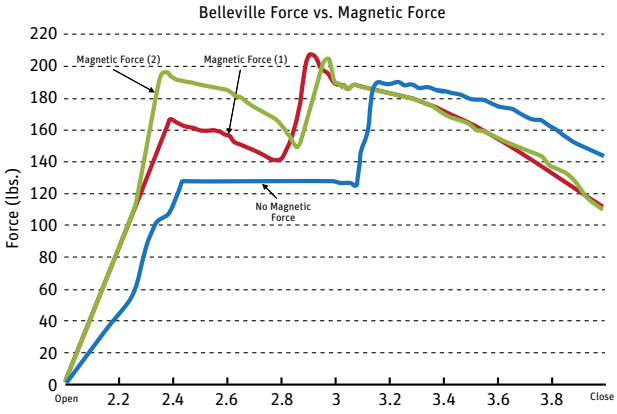


^ Schematic of linkage of Baker Hughes electromagnetic–acoustic transducer



^ Contact force due to friction between telescopic parts

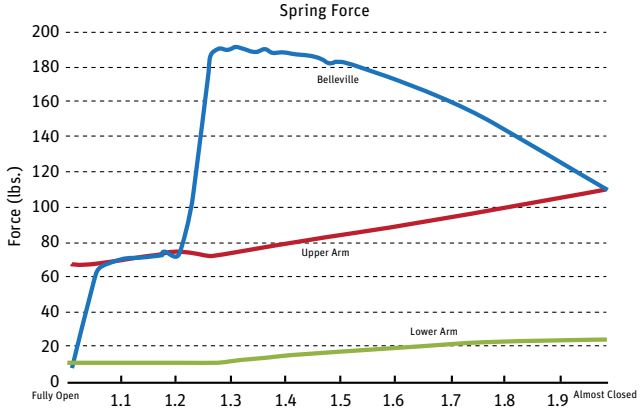
remedies. The design of the linkage is particularly challenging because, for accurate measurements, the sensors must maintain the contact between instrument and casing. The hole and casing are not always straight, so to ensure that the sensors maintain contact when a bend occurs, each sensor is attached to a spring-loaded telescopic arm that extends to maintain contact with the casing. The linkage also must operate properly in a wide range of wellbore diameters. It was critical that engineers optimize the design of the linkage because of the high cost of building and testing prototypes, and the even higher cost of a downhole failure in a well being readied for production. Baker Hughes engineers estimated that traditional design methods, which rely on building and testing prototypes to evaluate the performance of proposed linkage designs, would have required three or four prototype iterations to meet the design specifications. This approach would have taken about three years to develop the product.



Resultant forces from simulation when the linkage is fully open and fully closed

Instead, Baker Hughes engineers used ANSYS Mechanical to evaluate multiple linkage designs and simulate their performance under a wide range of conditions. Engineers began with a SolidWorks® model of the initial concept design. They defined several types of joints and springs used in the linkage, the spring constants, the torque function of the motor driving the linkage, and the coefficients of friction of the telescopic arm and joints in the linkage. Concern about the friction between telescopic parts due to potential deflection of the lower telescopic arm led Baker Hughes engineers to define the arm as a flexible model by incorporating a finite element mesh, material assignment and solver setup.

ANSYS Mechanical provided a complete picture of the linkage's performance, including contact friction forces, flexing, twisting and deforming of the lower telescopic arm. For example, it calculated the frictional forces generated as the telescopic arm was deployed and the joints in the linkage rotated, and the forces generated by the springs as they were displaced. The simulation results showed several opportunities for improvement. For example, in the initial concept design, the springs that retract the telescopic arm did not generate enough force to overcome the frictional forces.



Spring forces when the linkage is fully open and fully closed

Baker Hughes engineers addressed these and other issues revealed by the simulation by changing design parameters in the model and rerunning the simulation. They solved the telescopic arm problem by adjusting the spring constants. Engineers used ANSYS Mechanical to evaluate many different alternative linkage designs. This ensured that the linkage would successfully perform the critical deployment and withdrawal operations. By calculating component loading, the team optimized components and ensured that they would survive downhole.

Based on the simulation results, Baker Hughes engineers identified a design that met the requirements and commissioned a prototype. Testing of this prototype showed it performed exactly as predicted by the simulation, so the company began production without additional prototype iterations. Baker Hughes engineers estimate that the use of simulation on this project saved about 6 months or about 20 percent of the total of 30 months required for this project from concept through completion.

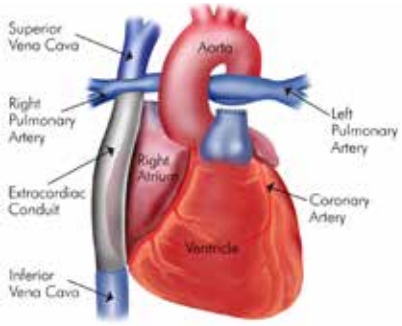
The Baker Hughes Integrity eXplorer™ wireline cement evaluation service was introduced in 2015. The service earned an Offshore Technology Conference (OTC) Spotlight on New Technology award for 2016. This OTC award is presented to selected exhibitors for the most innovative hardware and software technologies for offshore exploration and production.

"Using accurate simulation in conjunction with skilled surgery will increase the effectiveness of these procedures and provide the young patients with better quality of life."

Hearts Content

The long-term prognosis for babies born with single-ventricle heart defects can depend on the location of vascular connections made during corrective surgery. **Shanghai Children's Medical Center** uses simulation to individualize this surgery. Researchers employ computational fluid dynamics to determine the optimal connection points based on the patient's cardiovascular anatomy, improving surgical effectiveness and resulting in a better quality of life for these children.

By **Liu Jinlong**,
Associate Professor,
Department of Cardiothoracic Surgery,
Shanghai Children's Medical Center, Shanghai, China



^ Heart after correction with Glenn and TCPC surgical procedures

A New System for Surgery
ansys.com/surgery

In a normal heart, the left ventricle pumps oxygenated blood to the body and the right ventricle pumps deoxygenated blood to the lungs. But about two out of every 1,000 babies are born with only a single ventricle. The oxygenated and the deoxygenated blood blend in the ventricle, and the mixture is pumped throughout the body, causing symptoms that include shortness of breath, low energy and a blue color in the extremities. This condition places such a heavy burden on the single ventricle that, without surgical correction, most children born with this disorder will die from heart failure within one year.

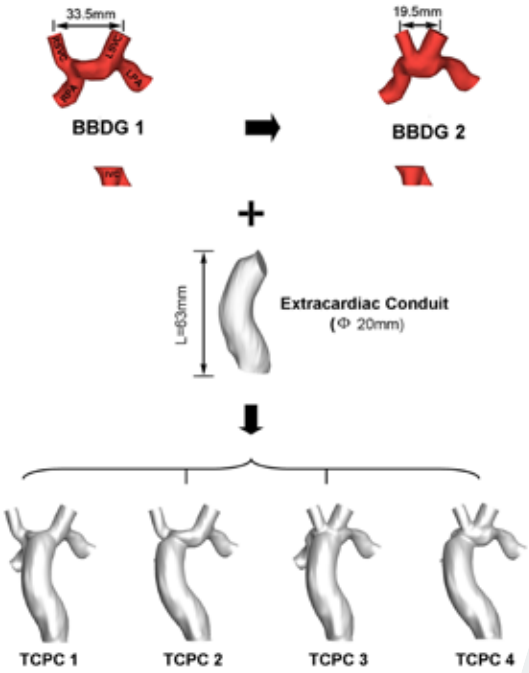
Correcting single ventricle defects normally involves reconfiguring the circulatory system to ease the burden on the ventricle. The ventricle still pumps blood to the body, but the blood returning from the body travels directly to the lungs via blood vessel connections for reoxygenation, before reaching the malformed heart's single pumping chamber.

The surgery is performed in a staged approach. The first stage, required in most but not all babies born with this defect, balances the blood flow so that an equal amount of blood travels to the body and lungs (Norwood procedure). In the second stage, called a bidirectional Glenn procedure, the vessels that drain blood from the head and upper body — the left and right superior vena cava (LSVC and RSVC) — are disconnected from the heart and sutured directly to the pulmonary artery (PA), which provides blood to the lungs. This removes some of the work done by the single ventricle. In the third stage, called the total cavopulmonary connection (TCPC) or Fontan procedure, the vessel returning blood from the lower half of the body — the inferior vena cava (IVC) — is disconnected from the heart and connected directly to the PA.

Clinical studies have found a wide variation in the long-term survival rate and post-operative quality of life of patients receiving these procedures. One reason for this disparity is that surgeons have the flexibility to connect the LSVC, RSVC and IVC to different points on the PAs. Which connection point will work best for a particular patient depends on the patient's heart structure and other variables that affect flow patterns in the veins. These flow patterns, in turn, have a major impact on the efficiency of the pulmonary system and the burden that is placed on the single ventricle.

TAKING PATIENTS' INDIVIDUAL ANATOMY INTO ACCOUNT

Until recently, surgeons have not had a method to determine the effects of different connection points on the patient's long-term health. Researchers at Shanghai Children's Medical Center use computational fluid dynamics (CFD) to perform virtual operations that take each patient's unique heart and blood vessels into account while evaluating



Connection points from the large veins to the pulmonary artery were varied in the four different models developed for this study (front view shown on top and back view on bottom).

different sites to connect vessels to the PA. Researchers can then compare power losses and energy efficiency across the flow domain to determine the configuration that will maximize energy efficiency for that specific patient. To calculate energy efficiency, clinicians divide the total energy leaving the flow domain across the two outlets by the sum of energy entering the system across the three inlets.

In a recent example, researchers performed magnetic resonance imaging (MRI) on a five-year-old boy who had been born with a single ventricle, and had already undergone the Glenn procedure, and whose doctors were planning to perform a TCPC. His MRI images were imported into Materialise Mimics® software for 3-D reconstruction of his vascular anatomy, which was used to perform virtual operations based on these configurations. First, a second Glenn procedure was performed virtually by moving the connection sites of the LSVC and RSVC vessels closer to the PA. Then two different virtual TCPC operations were performed on each of these two models with different connection points for the IVC. The result was four different geometrical models, each based on different attachment alternatives, for the Glenn and TCPC surgeries.

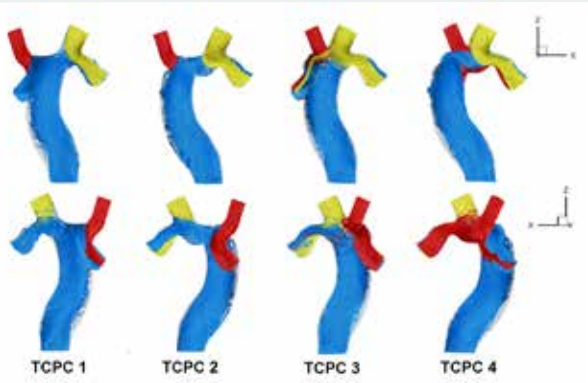
The researchers exported the four models as STL files that they then imported into ANSYS Workbench. A tetrahedral mesh was generated in the central connection area, and five boundary-fitted prism layers were created at the near-wall regions to improve the resolution with which fluid motion could be determined in this critical area.

Mass flow rates measured by the MRI on each vessel entering the PA were imposed as boundary conditions. They set a static pressure boundary condition at the outlet of the left pulmonary artery (LPA) and set five different static pressures at the outlet of the right pulmonary artery (RPA) to vary the relative flow rates between the LPA and the RPA. The flow rates were varied (40:60, 45:55, 50:50, 55:45 and 60:40) because different levels of vascular resistance of the LPA and RPA lead to this degree of disparity in patients. The team calculated the power loss in each simulation based on the static pressure, velocity and flow rate on the cross-sections of each of the five vessels.

DETERMINING THE IDEAL SURGERY

The results showed that the power loss was the lowest and the energy efficiency the highest in the TCPC 2 configuration, while TCPC 4 provided the largest power loss and the lowest energy efficiency. TCPC 1 and TCPC 3 fell in between. The variation in the relative pulmonary flow rates (LPA:RPA) did not affect the relative ranking of the different TCPC surgical options; however, it did significantly affect the relative differences between these options. For example, the value power loss reached its lowest level at an LPA:RPA flow ratio of 50:50 in TCPC 1 and TCPC 3. However, in TCPC 2 and TCPC 4, the flow domain power loss kept decreasing as the RPA flow ratio increased.

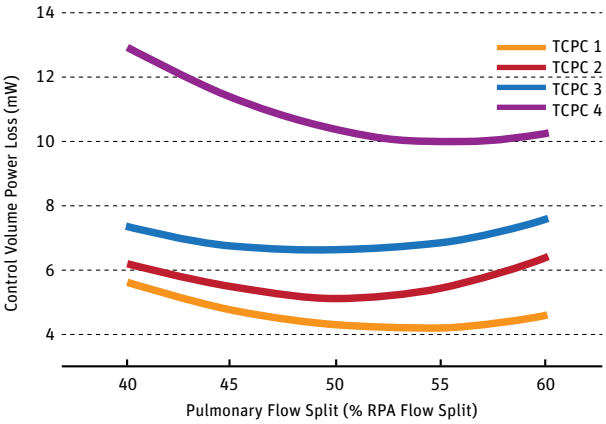
The flow pattern results helped explain why the different designs performed as they did. For example, the results



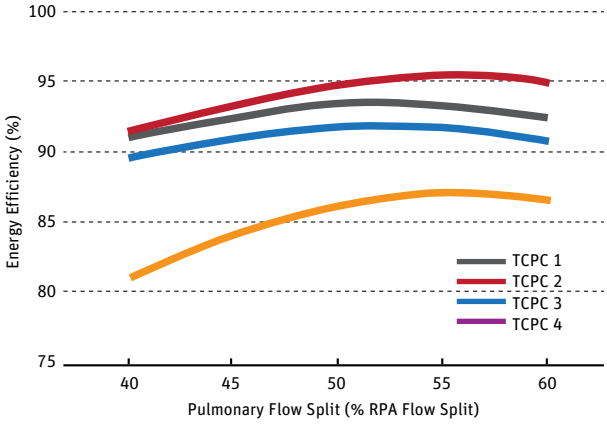
Velocity flow streamlines for the four different models (front view shown on top and back view on bottom) showing direction of flow. Each inlet is represented by a different color.

showed that the particular connections used in TCPC 1 caused interaction between the IVC and LSVC streams, producing turbulence that wasted power. However, when the connection was moved in TCPC 2, there was no turbulence in this area. Researchers concluded that the streams from the LSVC and RSVC should not interact with the IVC in order to avoid turbulence and resulting power loss.

Based on the simulation results, Shanghai Children's Medical Center researchers recommended that the TCPC 2 surgical configuration be performed on the patient. Once researchers optimized the surgical procedure, they printed a



Power loss for the different geometries and flow splits



Energy efficiency for the different geometries and flow splits

3-D model of the recommended surgical geometry as a guide for the surgeon. At the patient's 5-year and 10-year folow-ups, he showed no signs of cardiac failure and displayed normal physical capacity. The latest echocardiogram showed no obstructions in the reconfigured areas of the circulatory system and normal cardiac function. Both sides of the branch pulmonary arteries were well developed. While it is not yet practical to perform simulation on every patient, one of the goals of the research team is to reduce the time and effort required for simulation in order to make this possible in the future. Another goal is improving the accuracy of the simulation by using a transient simulation with boundary conditions controlled by a user-defined function (UDF) to model the variations in inlet flow and velocity during the pulmonary cycle.

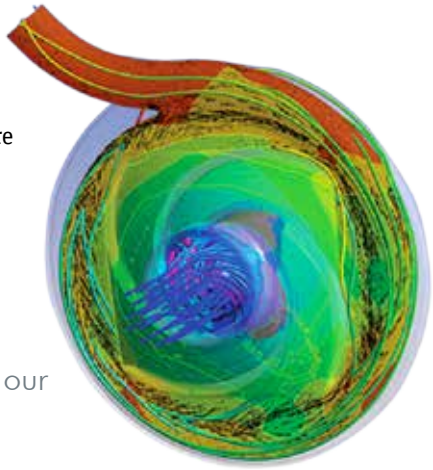
It is hoped that using accurate simulation in conjunction with skilled surgery will increase the effectiveness of these procedures and provide these young patients with better quality of life.

Simulation in the News

FLOWSERVE EMBRACES "DIGITAL TWIN" FOR REAL-TIME ANALYSIS

Develop3D, August 2016

Flowserve signed an enterprise license agreement to use ANSYS simulation software to produce "digital twins" — complete system virtual prototypes — of its products, enabling real-time monitoring of industrial pumps, seals and valves. Digital twins enable Flowserve to analyze true operational data to determine the performance of products in real-world operating conditions.



> "We expect our utilization of IoT technology and ANSYS simulation capabilities to increase the value we provide when our customers need advanced flow solutions."
>
> — **Keith Gillespie**, *Flowserve CSO*

ANSYS 17.2 RELEASED

ANSYS, August/September 2016

ANSYS 17.2 has been released to add a wealth of new functionality to this simulation tool platform, including enhanced multiphysics coupling, new workflows for antenna design, and automated temperature characteristics

for electric machines. This latest release also adds GT-Suite 2016 plug-in compatibility for better combustion modeling; a complete workflow for software design, code generation, and software testing and verification; and a new SCADE Test environment capability to automate the testing of embedded displays. ANSYS AIM 17.2 enhances engineering simulation for thermal management, extends collaboration between designers and analysts, and brings upfront simulation to Japanese engineers in their own language.

ANSYS, GE PARTNER TO BRING SIMULATION TO THE INDUSTRIAL IOT

Design World, September 2016

An agreement between ANSYS and General Electric expands GE's use of ANSYS engineering simulation solutions to pilot new simulation-as-a-service applications built on Predix, the operating system and platform for industrial internet applications. These applications focus on gathering big data analytics across industries such as aviation, transportation, healthcare, and oil and gas. Businesses will be able to analyze the performance of smart, connected machines, predict future performance, avoid unplanned downtime, and accelerate product development.

LEADERSHIP SUCCESSION PLAN

Digital Engineering, August 2016

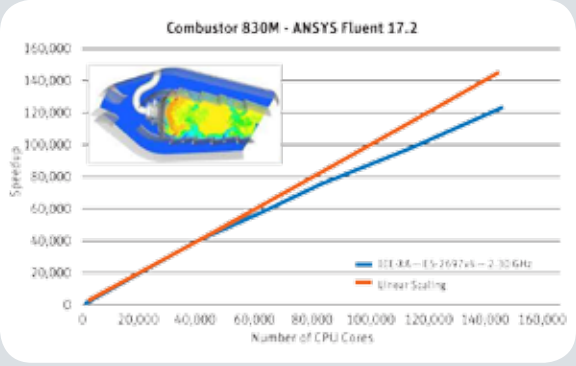
A leadership succession plan for ANSYS has been announced. James E. Cashman, who has served as ANSYS' chief executive officer since 2000, will become chairman of the board of directors effective January 1, 2017, and will be succeeded by Ajei S. Gopal, a technology industry veteran and member of the board since 2011.

Gopal, has been appointed president and chief operating officer until January 1, and will continue to serve on the Board.

SGI AND ANSYS ACHIEVE NEW CFD WORLD RECORD IN HPC

ANSYS Blog, September 2016

ANSYS and SGI achieved a new world record for scaling ANSYS Fluent on an SGI® ICE™ XA, one of the world's fastest commercial distributed-memory supercomputer platforms. This joint project breaks last year's 129,024 core record by more than 16,000 cores.



> "The new world record benchmark reduces the total solver wall clock time to run a single simulation from 20 minutes for 1,296 cores to a mere 13 seconds using 145,152 cores."
>
> — **Tony DeVarco**, *Director of Virtual Product Development Manufacturing Solutions, SGI*

ANSYS HELPS STARTUP COMPANIES CREATE TOMORROW'S PRODUCTS

Industry Today, September 2016

Startup companies can speed their innovative products to market by leveraging the same cutting-edge engineering simulation solutions used by larger, more established industry leaders. The ANSYS Startup Program provides small companies around the world with virtually free access to ANSYS' leading suite of engineering simulation products

ANSYS HONORED AS 2016 CONFIRMIT ACE AWARDS JUDGES' CHOICE WINNER

Mechspot, June 2016

ANSYS has been named a Judges' Choice winner of the 2016 Confirmit ACE (Achievement in Customer Excellence) Awards in the Innovation in Customer Engagement category. The ACE Awards program was established in 2005 to recognize outstanding achievement in customer experience.

RESEARCH LAB TO IMPROVE METAL 3-D PRINTING

NextPittsburgh, June 2016

The University of Pittsburgh and ANSYS have formed a partnership to establish a new additive manufacturing research laboratory at the Swanson School of Engineering. The partnership will ensure collaborative research between Pitt faculty and students and ANSYS, and further education and research to develop software tools that solve some of the industry's toughest additive manufacturing problems. As part of the partnership, the university is opening a 1,200-square-foot additive manufacturing lab that will enable students and faculty to perform their own research.

BRINGING SIMULATION TO STEM EDUCATION

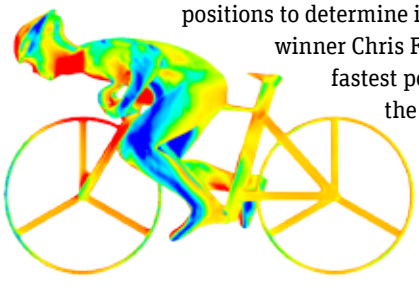
Engineering.com, June 2016

ANSYS's academic program manager, Paul Lethbridge, reveals the challenges of incorporating simulation into education programs, especially at the K–12 level. ANSYS helps teachers and students gain access to simulation by offering educational editions of its software and through free classes such as Cornell's massive open online course (MOOC).

WHY CHRIS FROOME'S UNUSUAL DESCENDING STYLE ISN'T AS FAST AS IT LOOKS

Cycling Weekly, July 2016

Bert Blocken of the Eindhoven University of Technology and other researchers at KU Leuven, the University of Liège in Belgium and ANSYS used ANSYS computational fluid dynamics (CFD) to compare four different bicycle riding positions to determine if Tour de France winner Chris Froome used the fastest position to descend the Col de Peyresourde, one of the most unexpected moments of the race.



ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA U.S.A. 15317

Send address corrections to
AdvantageAddressChange@ansys.com