

www.ozeninc.com
info@ozeninc.com
(408) 732 4665
1210 E Arques Ave St 207
Sunnyvale, CA 94085

Reliable World Class Insights

Your Silicon Valley Partner in Simulation

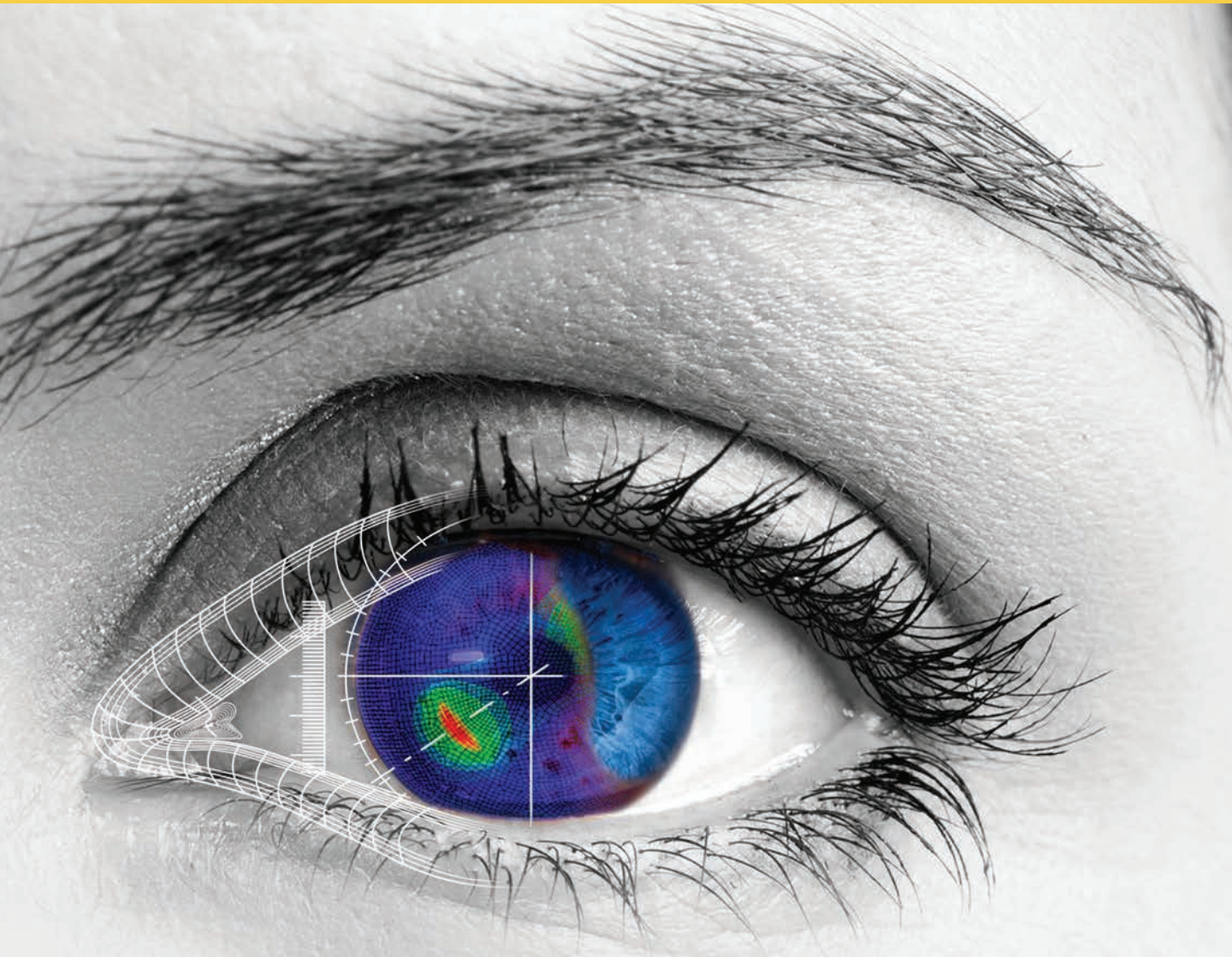
ANSYS Sales, Consulting, Training & Support

ANSYS®

ADVANTAGE™

Excellence in Engineering Simulation

VOLUME IX | ISSUE 1 | 2015



FOCUS ON HEALTHCARE

1 Entering the In Silico Era **11** A Perfect Vision **14** Charged Up



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

Senior Editor
Tim Palucka

Editors
Ashleigh Chicko
Judy Cooper
Erik Ferguson
Verly Flores
Kara Gremillion
Thomas Matich
Mark Ravenstahl
Ravi Ravikumar
Walter Scott

Editorial Adviser
Tom Smithyman

Editorial Contributor
ANSYS Customer Excellence North America

Art Directors
Ron Santillo
Dan Hart

Graphics Contributor
Gregg Weber

Design
Agency 1903

Join the simulation conversation
ansys.com/Social@ANSYS

© 2015 ANSYS, Inc.

Realize Your Product Promise®
ANSYS is dedicated exclusively to developing engineering simulation software that fosters rapid and innovative product design. ANSYS technology enables you to predict with confidence that your product will thrive in the real world. For more than 40 years, customers in the most demanding markets have trusted our solutions to help ensure the integrity of their products and drive business success through innovation.

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

Subscribe at
ansys.com/magazine

For ANSYS, Inc. sales information, call 1.866.267.9724.

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

For address changes, contact AdvantageAddressChange@ansys.com.

Neither ANSYS, Inc. nor Agency 1903 guarantees or warrants accuracy or completeness of the material contained in this publication.
ANSYS, ALinks, Ansoft Designer, Aqwa, Asas, Autodyn, BladeModeler, CFD, CFX, Chip Power Module (CPM), Chip Thermal Model (CTM), DesignerRF, DesignerSI, DesignModeler, DesignSpace, DesignXplorer, Engineering Knowledge Manager (EKM), Explicit STR, Fatigue, Fluent, FORTÉ, Full-Wave SPICE, HFSS, ICEM CFD, Icepak, Icepro, Maxwell, Mechanical, Mesh Morpher, Multiphysics, Nexxim, Optimetrics, ParICs, PathFinder, PExprt, Polyflow, PowerArtist, PowerArtist Calibrator and Estimator (PACE), Professional, Q3D Extractor, QuickEye, Realize Your Product Promise, RedHawk, Rigid Dynamics, RMxprt, RTL Power Model (RPM), SCADE Display, SCADE Lifecycle, SCADE Suite, SCADE System, Sentinel, SIwave, Simplorer, Simulation-Driven Product Development, Solver on Demand, SpaceClaim, Structural, Super-Compact, TGrid, Totem, TPA, TurboGrid, Vista TF, VerifEye, WinIQSIM, Workbench, and any and all ANSYS, Inc. brand, product, service, and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries.
ICEM CFD is a trademark licensed by ANSYS, Inc. LS-DYNA is a registered trademark of Livermore Software Technology Corporation. nCode DesignLife is a trademark of HBM nCode. All other brand, product, service, and feature names or trademarks are the property of their respective owners.



ENTERING THE IN SILICO ERA

The healthcare industry needs solutions now. The world requires rapid-fire innovation to treat an aging population, reduce healthcare costs and ensure the reliability of treatments. Medical and pharmaceutical leaders are widely adopting engineering simulation to create significant opportunities from these challenges.

By Thierry Marchal, Director Healthcare Industry Marketing, ANSYS

Innovation in healthcare is not just a question of business; often it is a matter of life and death. Medical and pharmaceutical companies must turn a profit, but they also have an ethical duty to improve lives — addressing common chronic illnesses and providing effective treatments for rare conditions — no matter the lack of profitability. Healthcare companies, therefore, face the dilemma of how to minimize cost and maximize efficiency.

Good healthcare should be the right of every person on Earth, but, as consumers, we expect healthcare costs to be as low as possible. The combination of these expectations with the skyrocketing cost of healthcare places huge pressure on companies to quickly come up with affordable but reliable and safe solutions. A technology shift is needed to ramp up medical research while slashing costs.

Medical and pharmaceutical companies face another quandary: They could save or improve millions of lives with one device or drug, but if a single patient's life could be endangered, companies could face plummeting stocks, fines, potential litigation and bad press. This need for extreme safety — far beyond the tolerance in any other industry — requires organizations to perform a delicate balancing act to provide inexpensive solutions.

To provide safe, reliable and cost-effective medical solutions, consumers and the medical field alike are turning to treatments that are personalized, participative, predictive and preventive, called P4 medicine.

This approach recognizes human variability, distinctions between individual patients, and the need for each of us to listen to our bodies to detect early signs of disease or distress. It considers that researchers must predict how medical conditions and diseases evolve so they can prevent pathology; it incorporates the use of minimally invasive actions that reduce the impact and cost of treatment. P4 medicine won't be possible without huge sets of data and the ability to continually monitor key parameters for each patient. To compile, manage and interpret such big data, researchers need computer models and simulation (CM&S) that describe healthy and diseased states in humans, which is also known as in silico medicine. This groundbreaking technology properly captures the vital conditions within and surrounding implantable or wearable devices, including their sensors and monitors; shares data with other devices, doctors and/or patients; and suggests action plans that will bring the patient's body back to a more healthy state.

To provide safe, reliable and cost-effective medical solutions, consumers and the medical field are turning to treatments that are personalized, participative, predictive and preventive.

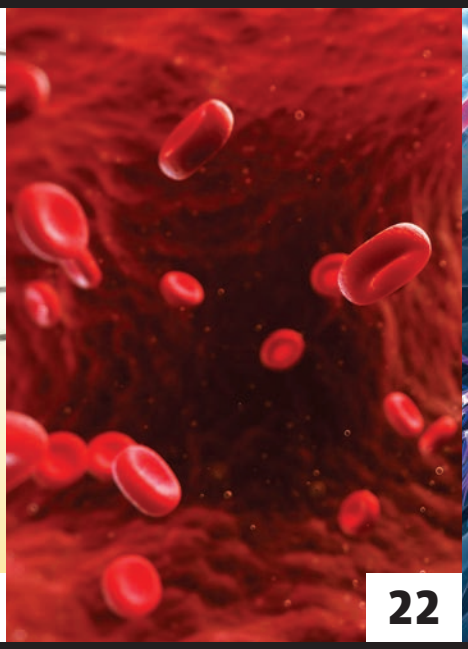
Combining fluid, structure and electrical physics that drive the human body — a body that now interacts more than ever with the connected devices that constitute the medical IoT — calls for advanced solutions in the healthcare field. At ANSYS, researchers progressively create and develop technologies to provide the medical and pharmaceutical worlds with a platform that allows development of a virtual physiological human (VPH).

New concepts now can be tested on large groups of virtual patients in very early stages of product and treatment development. By modeling the entire human body using a systems approach, healthcare companies can accurately predict the behavior of their solution for a wide range of human conditions and activities. The in silico revolution, in turn, will change the rules of the healthcare world, just like the in vivo and in vitro revolutions did centuries ago.

© 2015 ANSYS, INC.



14



22



25

FEATURES

6
BEST PRACTICES

In Vivo, In Vitro, In Silico!
Four best practices help ensure a smooth technology shift to computer modeling and simulation for medical device and pharmaceutical organizations.

11
OPHTHALMOLOGY

A Perfect Vision
To dramatically improve vision through customized patient-specific surgery, Optimeyes employs structural simulation in clinical ophthalmology.

14
WEARABLE DEVICES

Charged Up
Medtronic ensures the safety of recharging subcutaneous medical devices through simulation.

18
HEARING AIDS

I Hear You
Simulation improves hearing aid performance while saving time and money by quickly iterating through design alternatives.

22
HEALTHCARE REGULATION

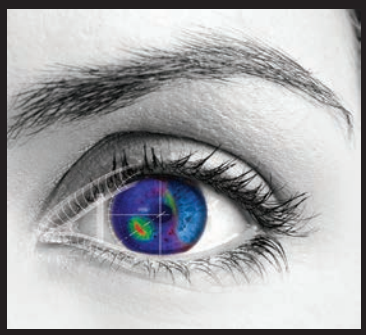
Simulation for Regulation
The Medical Device Innovation Consortium advances the use of simulation in the development and regulation of medical devices.

25
CARDIOLOGY

Decision Under Pressure
A non-invasive computational method for determining pressure drop across an arterial blockage could help cardiologists make more-objective and accurate treatment decisions.

29
PHARMACEUTICAL

The Right Mix
CFD simulation saves time and money by validating the ability of a single-use mixer design to scale to 5,000 liters.



ABOUT THE COVER

Healthcare companies must accurately predict the behavior of solutions for a wide range of human conditions and activities. Scientists and engineers at Integrated Scientific Services AG are helping physicians to improve surgical outcomes using engineering simulation for personalized medicine.

© 2015 ANSYS, INC.





ANSYS ADVANTAGE
Excellence in Engineering Simulation

Available in print or on your iPad

ansys.com/magazine



SIMULATION@WORK

33
CONSTRUCTION

Energy-Efficient Tunnel Ventilation
A leading ventilation company develops jet fans that are smaller and up to 30 percent more energy efficient.

36
FLUID–MECHANICAL SYSTEMS SIMULATION

Shaking All Over
Multiphysics simulation solves a vibration issue in a Francis turbine.

40
FLUID–MECHANICAL SYSTEMS SIMULATION

Knowing the Score
Analyzing a World Cup stadium with ANSYS multiphysics tools takes one-tenth the time of wind-tunnel testing.

DEPARTMENTS

44
ACADEMIC

Developing Reliable Antennas for Smartphones
A team of researchers used ANSYS technology to design a small multiband internal antenna to successfully operate across eight different frequency bands.

47
ACADEMIC

Magnetic Nanoparticle Simulation
Research using simulation will help to develop biomagnetic beads for disease treatment.

51
BEST PRACTICES

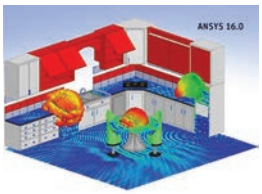
Accelerating ANSYS Fluent Simulations with NVIDIA GPUs
ANSYS Fluent supports GPUs so engineers can meet project schedules and get robust products to market faster.

© 2015 ANSYS, INC.

Simulation in the News

ANSYS LAUNCHES RELEASE 16.0

Desktop Engineering
deskeng.com, January 2015



ANSYS 16.0 delivers major advancements across the company's entire portfolio, including structures, fluids, electronics and systems engineering solutions — providing engineers with the ability to validate complete virtual prototypes. This release introduces an electronics desktop, provides users with the opportunity to create 3-D components and integrate them into larger electronics assemblies, reduces fluid dynamic simulation time for complex models by up to 40 percent, and launches ANSYS AIM, an innovative, immersive simulation environment that lowers the barrier to entry for multiphysics simulations.

"Validating prototypes on the computer using ANSYS simulation software, in addition to physical tests, drives us to higher levels of innovation while getting our products to market faster and more cost effectively."

– Robert Terhune, Mechanical Engineer, 4moms

ANSYS RELEASES NEW VERSION OF SPACECLAIM

Develop 3D
develop3d.com, January 2015

The newly released version of ANSYS SpaceClaim enables engineers to manipulate product geometries more easily than ever, as well as to print fast and flexibly in 3-D. Additional improvements include imprint and wrap tools for easier simulation edits, clean and detection functionality, and improved integration with ANSYS Workbench.

MEET THE 2015 ANSYS HALL OF FAME WINNERS

Engineering.com
engineering.com, January 2015

The five ANSYS Hall of Fame winners for 2015 include a company developing new spinal instruments to reduce the risks of scoliosis surgery and a university that is exploring how leatherback turtles might survive global climate change. The contest, which highlights some of the most complex scientific and engineering challenges, gives ANSYS users the opportunity to showcase their simulation work by producing eye-grabbing images and animations. For the second year in a row, submissions were divided into two categories — corporate and academic — and allowed ANSYS to select multiple "best-in-class" winners from each category.



▲ Simulation of a spherical valve for a hydroelectric plant inlet. ANDRITZ Hydro, whose entry was named to the Hall of Fame, reduced the weight of the valve inlet with the help of simulation.

SCALING ANSYS HPC IN THE CLOUD

insideHPC
insidehpc.com, October 2014

A newly published report by Techila Technologies and Cargotec's MacGregor business takes a look at how ANSYS simulation users can reduce the total cost of high-performance computing ownership by integrating cloud services with IT. Because MacGregor's marine structural components are custom-designed for each specific application and must be simulated quickly, the role of HPC is critical.

ANSYS BOOSTS COMPUTATIONAL FLUID DYNAMICS OFFERING WITH ACQUISITION

Financial News
financial-news.co.uk, February 2015

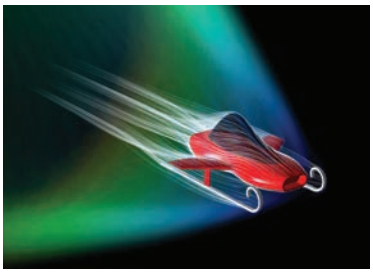
ANSYS has acquired the assets of Newmerical Technologies International (NTI), a premier developer of in-flight icing simulation software and associated design, testing and certification services. NTI's suite of state-of-the-art specialized software can solve problems in aerodynamics, in-flight icing, heat transfer, fluid–structure interaction and wind engineering.

© 2015 ANSYS, INC.

SANTA'S SLEIGH UPGRADED WITH COMPUTATIONAL FLUID DYNAMICS SIMULATION

3D CAD World
3dcadworld.com, December 2014

ANSYS celebrated the holiday season by attempting to improve the deer-energy efficiency of Santa Claus's sleigh and thus protect the Christmas figure from the harsh elements. CFD simulation revealed that the drag on Santa's traditional sleigh was very high. The newly designed sleigh, which resembles a small aircraft, reduces drag by 90 percent and now, hypothetically, can reach even supersonic speeds.



▲ Santa Claus's new sleigh, designed with simulation, is more aerodynamic and built to handle supersonic speeds.

NORTHROP GRUMMAN TEAMED WITH ANSYS ON BLACK HAWK

Avionics Today
aviationtoday.com, October 2014

Northrop Grumman is developing the new Black Hawk cockpit display system using ANSYS simulation. ANSYS SCADE Suite and Display are being used to upgrade to the digitized UH-60V cockpit and help meet new safety standards.

CFD SIMULATIONS ARE SAVING INFANTS FROM DEADLY HEART DEFECTS

Engineering.com
engineering.com, December 2014

Approximately 1,000 children are born in the United States each year with a heart condition known as hypoplastic left heart syndrome, which is characterized by underdevelopment of the left side of the heart; the diagnosis affects newborns worldwide. Three surgeries are required within a child's first two to three years to ensure his or her health and survival. Doctors now can gain more information to select optimal surgical options by testing scenarios using ANSYS Fluent CFD simulations. By employing simulation, medical professionals save lives and cut down on unnecessary or excessively dangerous surgeries on very young patients.



▲ ANSYS simulation of velocity magnitude at a total cavopulmonary connection (TCPC) surgery site
Courtesy Politecnico di Milano.

COMPUTERS IN THE SKY: THE DESIGN OF NEXT-GEN AIRCRAFT

Aerospace Manufacturing and Design
onlineamd.com, November 2014

Robert Harwood, director of aerospace and defense at ANSYS, breaks down the three main considerations for designing next-generation aircraft: economic efficiency, environmental impact and improved passenger experience. A suitably next-generation digital design process, he writes, is necessary for the work but has not been fully realized. The article features Boeing, Airbus, Lockheed Martin and Northrop Grumman advancements.

SIMULATION HELPS MOTORBIKE RACER BREAK DOWNHILL SPEED RECORD

The Engineer
theengineer.co.uk, November 2014



▲ The team from Sheffield Hallam University's Centre for Sports Engineering Research with Guy Martin (center)
Courtesy Channel 4.

A team from Sheffield Hallam University used ANSYS CFD technology to design a gravity-powered sled that broke the world record for downhill speed. The vehicle was created around — and built to accommodate — a virtual prototype of Guy Martin. Martin, a mechanic, motorbike racer and TV presenter, then raced the vehicle downhill in France at a new world record speed of 85.61 mph, breaking his own 2013 world record of 83.49 mph.

HOW GERMS TRAVEL: THE SNEEZE-ON-A-PLANE EDITION

Popular Science
popsci.com, October 2014

A simulation generated by the FAA's Center of Excellence at Purdue University used ANSYS fluid dynamics solutions to demonstrate a passenger sneezing in a crowded airplane and dispersing influenza particles throughout the cabin. The Federal Aviation Administration has been interested in determining how flu particles spread on airplanes since the outbreak of SARS in 2002. The simulation shows that airborne diseases can be spread quickly in flight.

© 2015 ANSYS, INC.



IN VIVO, IN VITRO, IN SILICO!

Simulation courtesy Foundation of Cardiac Surgery Development.

Four best practices help ensure a smooth technology shift to computer modeling and simulation for medical device and pharmaceutical organizations.

By Thierry Marchal, Director Healthcare Industry Marketing, ANSYS

Following early engineering simulation adopters such as the aeronautic, automotive and nuclear industries, biomedical and pharmaceutical companies have started to widely embrace computer modeling and simulation (CM&S) to accelerate their product development processes and reduce the huge cost of bringing a new drug to market. (That cost could be up to $2 billion U.S.) Yet, some healthcare practitioners and organizations remain hesitant to adopt this unfamiliar approach and technology.

Medical history shows that those who embraced true technological revolutions early emerged as new market leaders. Others, including some who were previously dominators, simply disappeared. The medical world experienced this shift a few centuries ago by following the example of Leonardo da Vinci, adopting an in vivo approach to understand how the human body works. In other words, researchers conducted tests and experiments on living organisms as well as cadavers. This led to tremendous innovation, such as the development of modern surgery, which saved many lives. Later in the 19th century, innovators developed in vitro testing, which was much faster than in vivo experiment but did not replace it completely. In these instances, researchers conducted tests and experiments in test tubes and on petri dishes. This led to a new wave of innovations, including the emergence of a large pharmaceutical industry.

Today, to maintain the exponential growth of innovation, the medical and pharmaceutical worlds are entering an era in which a growing number of experiments will be done on the computer — known as in silico — to complete and accelerate in vivo and in vitro approaches. This has the potential to revolutionize science and medicine. Some companies still wonder, however, how to navigate through this intimidating transition to a new style or stage of testing.

© 2015 ANSYS, INC.

Medical and pharmaceutical worlds are entering an era in which a growing number of experiments will be done on the computer.

THE ROLE OF NUMERICAL SIMULATION IN MEDICAL DEVICE NPD: PAST, PRESENT AND FUTURE
ansys.com/91heart

As more technology advocates interact with market leaders and innovators, four best practices emerge.

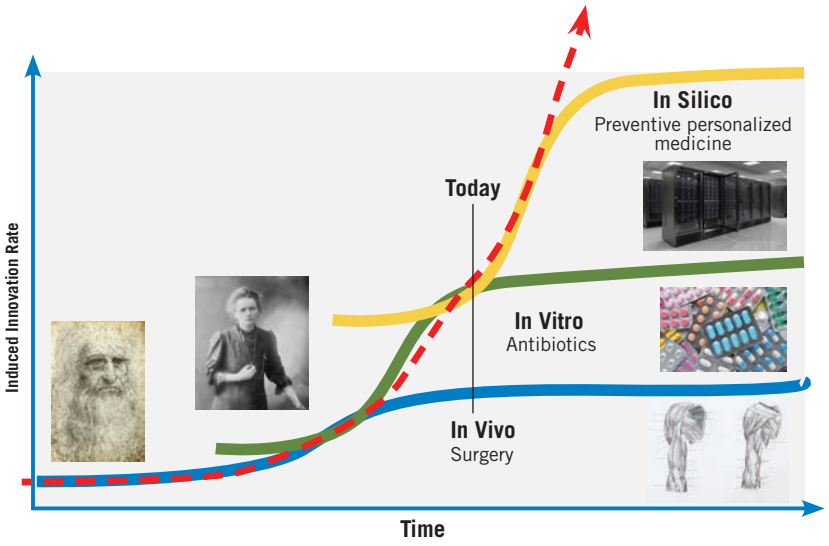
VIRTUAL HUMAN LABORATORY

Many medical products, and, of course, all drugs, directly interact with the body. Testing them in their working environment is a necessary but also challenging task. It can be difficult or impossible to find volunteers to test the performance of a given treatment without endangering them. Thanks to advances in medical imaging, computational power, and numerical algorithms and models, medical and pharmaceutical companies now can reproduce human environments. If necessary, researchers can extend them with great geometrical and operating condition accuracy, potentially modeling the entire human body through a systems approach.

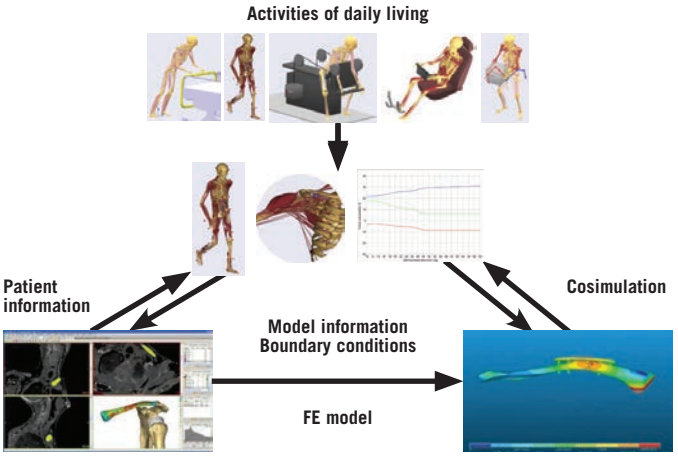
Best-in-class companies develop a detailed model of both the device and the part of the body interacting with it. They predict the complex thermal, structural, fluid and/or electromagnetic behavior of the natural components — such as soft tissues, bones and blood — by using advanced models previously validated through experiments. Proper boundary conditions mimicking different patient activities and pathologies are considered through a combination of 3-D modeling and systems-level simulation to perform full-body simulation. Understanding in detail how the device might work is crucial in the early concept development stage of the product development process. It reveals possible failures and opens the door to optimization.

Developing such advanced models of the human body is usually done step by step — with each adoption level providing valuable insight.

As a first step, a company using engineering simulation adds the components of the human body that directly interact with the device, a first-level adoption of the virtual human interaction laboratory best practice. Researchers can then add more to mimic the complexity of the human body — by



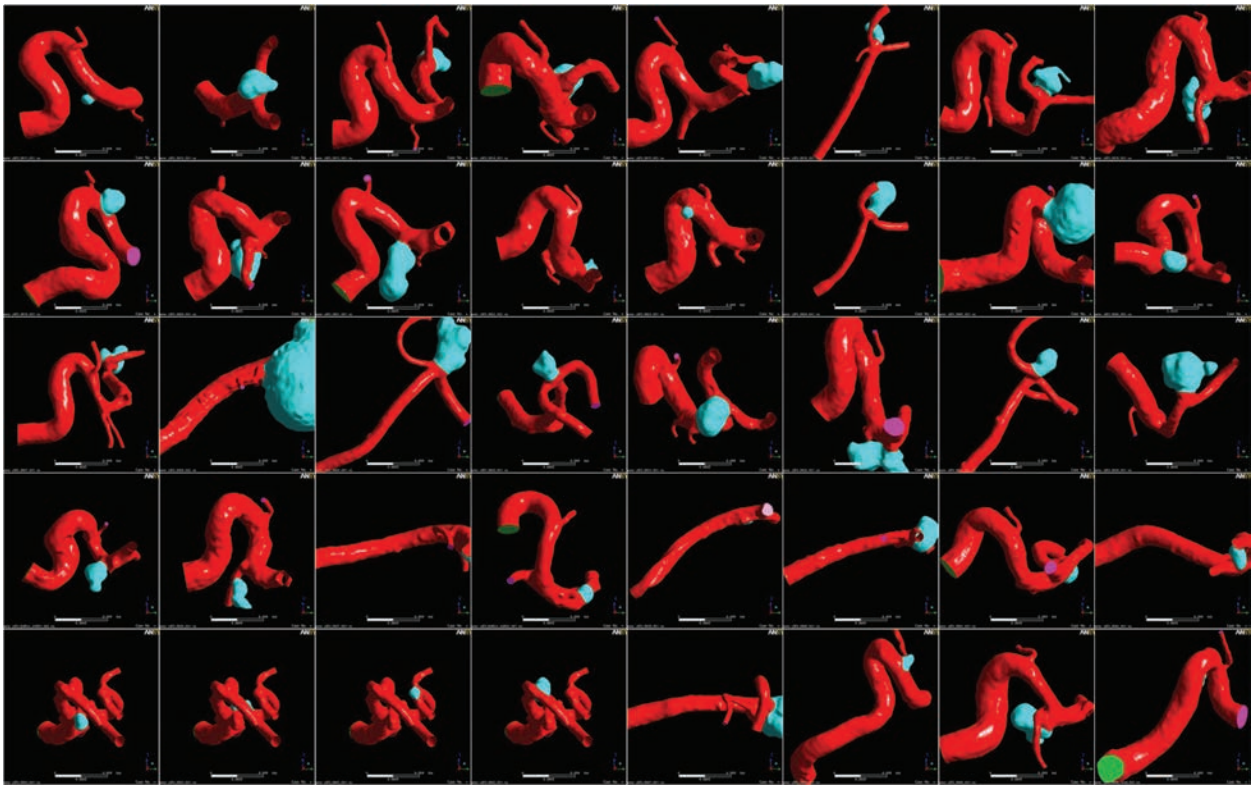
▲ The in silico approach will complement in vivo and in vitro testing to boost the innovation rate.



▲ Testing new devices in the working environment of the human body during various activities minimizes the risk of post-release failure. *Courtesy Hochschule Regensburg University of Applied Science.*

The in silico approach has the potential to revolutionize science and medicine.

© 2015 ANSYS, INC.



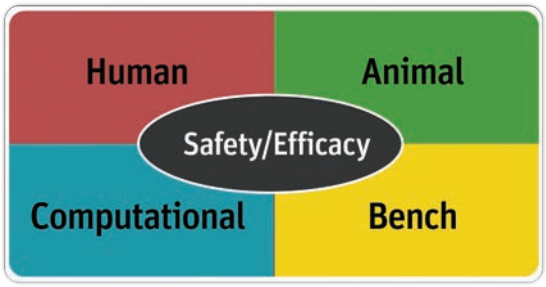
▲ Testing new cerebral aneurysm treatment on more than 300 patient-specific aneurysms was the goal of the @neurIST project. *Courtesy the @neurIST project.*

considering more-advanced nonlinear models or by adding more physics — to understand body–device interaction with greater fidelity. They also should progressively expand the number of body parts until they encompass the full body, possibly by using a reduced-order model approach. Through this progressive approach, innovative companies continually improve their virtual testing processes.

For example, the VIRTUheart™ project featured on page 25 helps doctors determine the best form of treatment. It uses simulation to improve the diagnosis of the severity of coronary artery disease in a given patient. Starkey Hearing Technologies employs simulation to ensure that hearing aids and their controllers perform reliably by taking into account both the device and the wearer. (See page 18.)

IN SILICO TESTING

The medical community faces a unique challenge from human variability; it is not enough to prove that a new device works well for a single person. Device manufacturers need to state with confidence that medical equipment will work as expected for the entire target population. Typically, new devices are tested on hundreds or thousands of patients in clinical trials to confirm that the prototype does not endanger the patient and that it provides the expected results despite the physiological and pathological variability of the target population.



▲ Regulatory authorities today recognize the importance of the computational approach to complement human, animal and bench testing.

The similarity and repeatability of these clinical tests is an obvious opportunity to use computer models and simulation. By doing the same simulation on large databases of patient-specific geometries and material properties, researchers can develop in silico clinical trials based on just a few key parameters and arrive at conclusions that are valuable to clinicians. Furthermore, as testing proceeds on virtual patients, there is no risk of harm or threat to safety. It is possible, therefore, to push the test to the extreme and determine the actual operating window for a given solution.

Developing such in silico clinical trials is a time-consuming task, as it requires accumulating large databases of patient-specific geometries, material properties and operating conditions. Modeling the interactions of the device

© 2015 ANSYS, INC.



▲ As the number of implantable and wearable devices increases, medical product developers must guarantee safe interaction with the body and minimize the risk of interference.

The in silico revolution will boost medical and pharmaceutical innovation and provide solutions so we can all live longer and better at an affordable cost.

with a single patient's body, however, is a first step toward an in silico testing approach. When medical device manufacturers collaborate with clinical partners and simulation software providers such as ANSYS, it is possible to progressively add more patient-specific geometries (and material properties) to the database. This progressive adoption and amplification of in silico clinical testing provides increasing confidence that new treatments will sail smoothly through the actual clinical testing.

This issue of *ANSYS Advantage* presents an article from Integrated Scientific Services AG on page 11. The company is developing Optimeyes — a clinical tool for ophthalmologists that runs ANSYS Mechanical in the background to produce patient-specific cataract surgical strategies.

PREDICTING MRI COMPLIANCE OF IMPLANTED DEVICES
ansys.com/91vivo

SIMULATION-DRIVEN FDA APPROVAL

Recently, regulatory authorities such as the U.S. Food and Drug Administration (FDA) have recognized the value of computer modeling and simulation within their approval processes when simulation is properly applied. Authorities encourage the adoption of simulation to complement other approaches such as bench, animal and human testing. Although computer modeling is still a tiny fraction of the testing, the Medical Device Innovation Consortium (MDIC) said the approach could represent more than 50 percent of clinical trials in the future. (Read more about MDIC on page 22.)

Building on the willingness of various regulatory authorities to accept burgeoning technology, engineering simulation can be an essential part of this approval process. Some medical device companies, including a few featured in this issue of *ANSYS Advantage*, have reported modifications of previously approved devices (and electromagnetic interaction with the body) approved through the exclusive use of simulation results. But, as some regulatory authorities might be less familiar with this technology or require results in a specific format, any organization modifying or developing medical devices should communicate early and often with regulatory authorities to determine expectations for validation and reporting.

Through these interactions, the recommended in silico approval strategy will be as clear for the company as for the authorities and increase the number of cases in which simulation can play an important role to reduce process time and minimize costs.

A NEW SYSTEM FOR SURGERY
ansys.com/91vivo2

VIRTUALLY CERTIFIED BODY-AREA NETWORK

As wearable and implantable devices connected to the Internet or each other multiply (as the Internet of Things becomes pervasive), individuals are fast becoming a complex network known as a body-area network (BAN). Even with an increasing number of devices directly interacting with the body on a regular basis, the human tolerance to absorb electromagnetic energy remains the same. So, product developers must

© 2015 ANSYS, INC.

ensure that the accumulated energy of implanted or worn devices will not exceed acceptable electromagnetic-field thresholds. It is equally important that these devices, especially those that have the potential to save lives, will not interfere with each other.

It is extremely difficult to test all configurations, and it is also challenging to obtain FCC approval for each new device. Clinical testing is time- and cost-prohibitive, if it were even possible. Market leaders such as Medtronic use engineering simulation to model both the device and the body to demonstrate that the specific absorption rate (SAR) is within safe levels for the wearer. (See page 14.) In silico testing has made it possible to virtually implant and/or wear different devices — and to model interactions between them or between the devices and the body — and even consider different body types (male, female, child, slim, average, overweight, etc.).

Although this approach is not expected to fully replace clinical trials in the foreseeable future — with a few exceptions — it already allows designers to identify potential failures in the very early stages of the product development process.

DO NOT FEAR THE IN SILICO REVOLUTION

This in silico revolution will boost medical and pharmaceutical innovation and provide solutions so we can all live longer and better at an affordable cost. It already is delivering huge business opportunities to organizations that adopt best practices such as the virtual human laboratory, in silico testing, simulation-driven FDA approval, and safe and reliable BAN. But this technology shift can be intimidating because of the uncertainty inherent in any new approach and the possible investment required.

As many of the articles in this issue of *ANSYS Advantage* reveal, it is important to start adopting these best practices as quickly as possible. Companies can gain significant value even with a minimal commitment to in silico testing. First, it is essential to identify which best practices could deliver the most immediate impact on your business. Next, adopt progressive, multilevel best practices that are most advantageous to you. Although organizations will not gain the full benefits of widespread deployment immediately, this approach will yield important results for a small investment.

ANSYS has extensive experience and a network of resources in this area and is willing to guide you in this important journey toward in silico medical product development.

Significant value can be gained even with a minimal commitment to in silico testing.



© 2015 ANSYS, INC.

A PERFECT VISION



To dramatically improve vision through customized patient-specific surgery, Optimeyes employs structural simulation in clinical ophthalmology.

By Harald Studer, Team Leader, Integrated Scientific Services AG, Biel, Switzerland

A cataract is a clouding of the lens of the eye that first blurs vision and can eventually lead to blindness if the lens becomes opaque. The World Health Organization estimates that nearly 18 million people worldwide are blind in both eyes because of cataracts, making this disease the cause of almost half of all cases of blindness. The main culprit is aging: Approximately half of all people older than 80 will suffer from cataracts to some extent.

The good news is that femtosecond laser surgery to remove the defective lens and replace it with an artificial intra-ocular lens is increasingly viable. In most cases, the surgery restores vision to the point that the patient has no need for glasses or contact lenses for vision correction. Still, about 30 percent of patients have to wear some form of corrective lens after the surgery. While this may seem like a reasonable outcome, the scientists and engineers at Integrated Scientific Services AG (ISS) knew they could do better.

Using knowledge of the structure and material properties of the various parts of the eye, especially the cornea, ISS teamed up with ANSYS to create Optimeyes — a clinical tool for ophthalmologists that runs ANSYS Mechanical in the background to produce patient-specific surgical strategies to improve results.

NONLINEAR MODELING FOR HEALTHCARE APPLICATIONS
ansys.com/91perfect

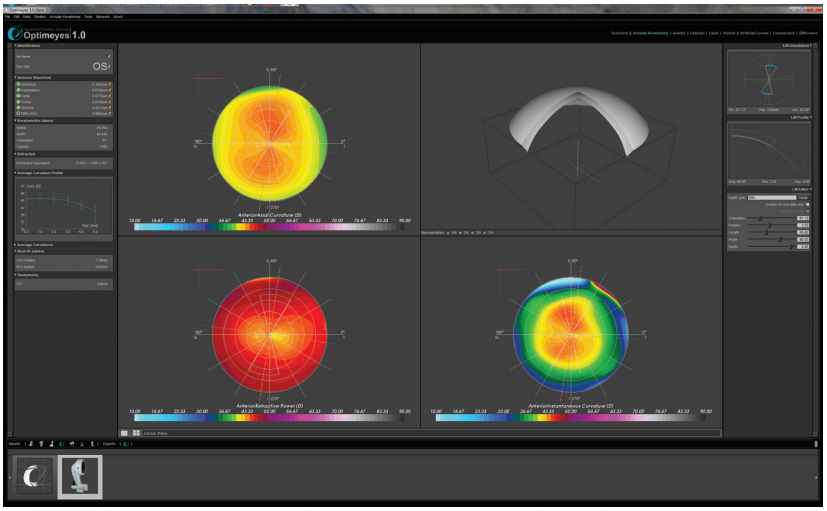
THE CORNEA IS THE KEY

The cornea is the transparent outermost layer of the front section of the eye, so virtually all eye surgeries, including the well-known LASIK procedure, involve cutting into the cornea to some degree. The details of where and how to make the laser incisions — the location, length, depth and angle of the cuts — are based largely on the doctor's experience, statistical information (so-called nomograms) and the properties of the average eye. The question is, what is the average eye?

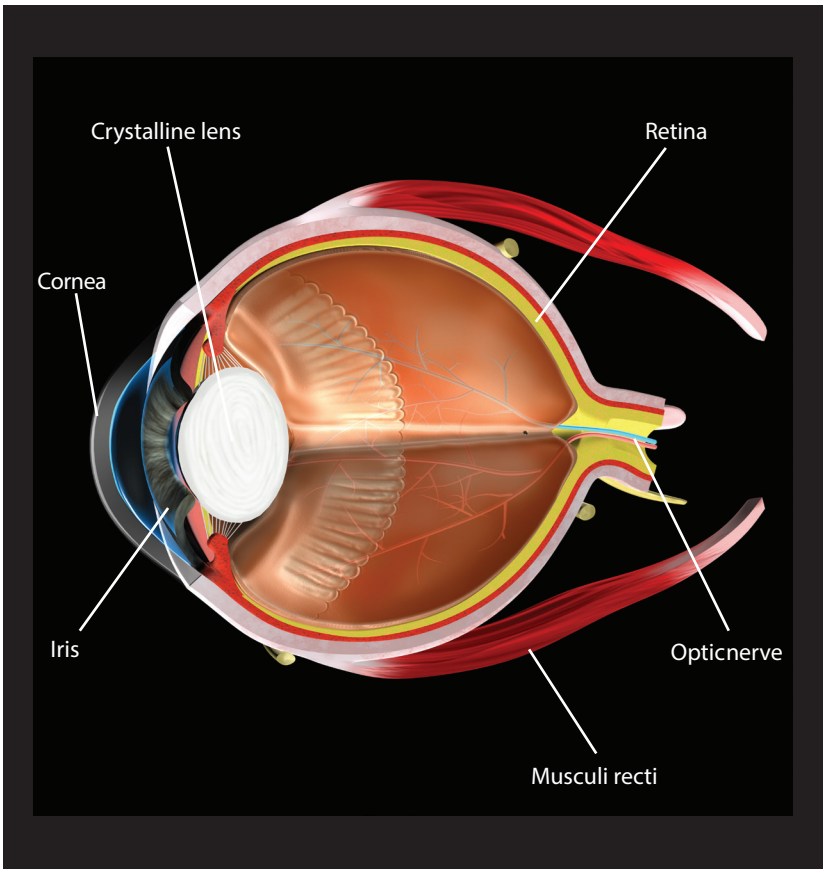
The shape of the cornea is determined to a large extent by its stiffness and the internal pressure of the eye, also known as intra-ocular pressure. The cornea is a composite comprising collagen fibers in various orientations in a matrix of polysaccharides and cells called keratocytes. Ultraviolet light emitted by the sun causes crosslinks to form between the collagen fibers, adding shear stiffness to the cornea. Because our eyes encounter more UV light with time, our corneas — as any other

A clinical tool for ophthalmologists runs ANSYS Mechanical in the background.

© 2015 ANSYS, INC.



Optimeyes software user interface



The human eye

ANSYS Mechanical calculates the deformations of a patient-specific cornea for a given laser treatment.

soft biological tissue — become stiffer as we age. To model corneal biomechanics realistically, ISS implemented these distinct material properties in an inhomogeneous, nonlinear and anisotropic user-material Fortran function (usermat) in ANSYS Mechanical.

Intra-ocular pressure is caused by the amount of a clear fluid called the aqueous humor in the eye. This fluid is formed in the ciliary process behind the iris, and it then flows through the pupil and fills the space between the iris and the cornea. The combination of this internal pressure with the biomechanical stiffness of the eye tissue determines the unique shape, or how much an eye deviates from the average. The amount of deviation is critical in determining where and how to make incisions in the cornea to optimize the surgical outcome.

GETTING PATIENT-SPECIFIC

Optimeyes is a physician-friendly software product with a user interface that looks like other diagnostic software that ophthalmologists use routinely. This ensures the doctor's level of comfort with the product. The patient simply looks into a Pentacam® camera and the doctor sees a standard image of the cornea's front (anterior) and back (posterior) surfaces on the screen.

Meanwhile, a script starts ANSYS Mechanical, which works unseen in the background, using the camera image as the geometry for its finite element analysis (FEA) simulations. ANSYS Mechanical uses between 60,000 and 80,000 hexagonal mesh elements, organized in five layers, to produce a 3-D model of the cornea and part of the sclera (the white of the eye). The mesh is morphed to match the geometry of the patient's eye, and boundary conditions are applied to the clamped border of the sclera. The thickness of the cornea at any given point is measured as the normal distance between its anterior and posterior surfaces in the model. In this first step, the model produces a complete topography of the patient's eye without any intra-ocular pressure applied.

To account for the initial stress distribution in the cornea, caused by the intra-ocular pressure inflating it like a balloon, ANSYS Mechanical uses an inverse nonlinear process to determine the theoretical stress-free shape of the cornea. The difference between the stress-free shape

and the measured shape yields the actual 3-D stress distribution.

MAKING THE CUT

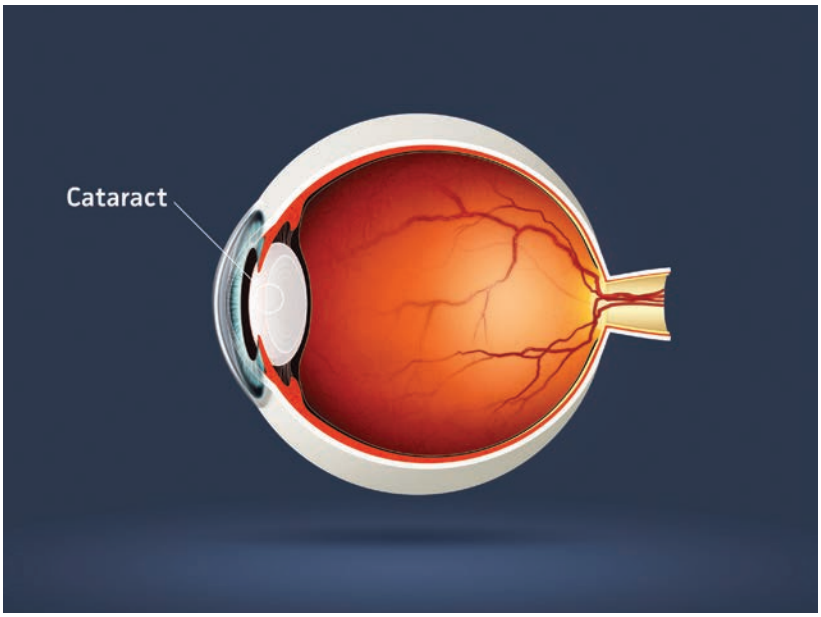
Even with all this modeling, another challenge remains: Laser surgery slightly deforms the cornea while cutting it, so this laser-induced deformation must be taken into account. Working together inside the clinical tool, ANSYS Mechanical calculates the deformations of a patient-specific cornea for a given laser treatment, and Optimeyes subsequently calculates surface curvatures and the length of the optical path of light rays passing through the cornea to predict the expected vision quality for the post-surgical eye.

To do this, ANSYS Mechanical simulates laser cuts in predefined areas of the mesh. When a standard intraocular pressure is applied to the model, the incision opens to reveal what kind of deformation will result from such a cut to the cornea. The result of the simulation appears on the Optimeyes screen for the doctor to review. An iterative, parametric workflow evaluates the impact of a given incision on the patient's post-surgical vision and adjusts the incision parameters to improve on the result. Ultimately, Optimeyes determines the optimized surgical procedure that will leave the patient with perfect vision. The complete process, from imaging the eye to optimized surgical model, currently takes about 20 minutes. The doctor then simply inputs the calculated optimized incision parameters — orientation, width and angle of incision — into the femtosecond laser instrument, and the instrument does the rest.

MOVING THE PRODUCT TO MARKET

As with any new medical instrumentation, Optimeyes has to go through very strict approval processes before it can be sold to ophthalmologists. ISS has started to perform clinical trials to demonstrate the effectiveness of Optimeyes in improving the post-surgical vision of cataract patients using this patient-specific approach compared to the currently used average-eye procedure. So far, Optimeyes has succeeded in correctly predicting surgical outcomes in all the cases studied. But more eye surgeries must be performed, and other steps taken, before approval can be granted.

The result of the simulation appears on the Optimeyes screen for the doctor to review.



▲ Image of eye with cataract: opacification of eye lens

ANSYS specialists have been integral players on the development team every step of the way. The flexibility of ANSYS solutions and the cooperation of ANSYS technical personnel have been critical in achieving the progress made to date. ANSYS staff continue to work with ISS to improve Optimeyes as the company pursues the ultimate goal of selling the product to clinicians.

In the meantime, ISS offers Optimeyes Professional Service to R&D departments of other companies. Typically, they want to implant a device in a cornea and have questions about how deep to implant it to make sure that the cornea's anterior surface does not deform. ISS performs parametric sensitivity analysis simulations so they can answer their own questions.

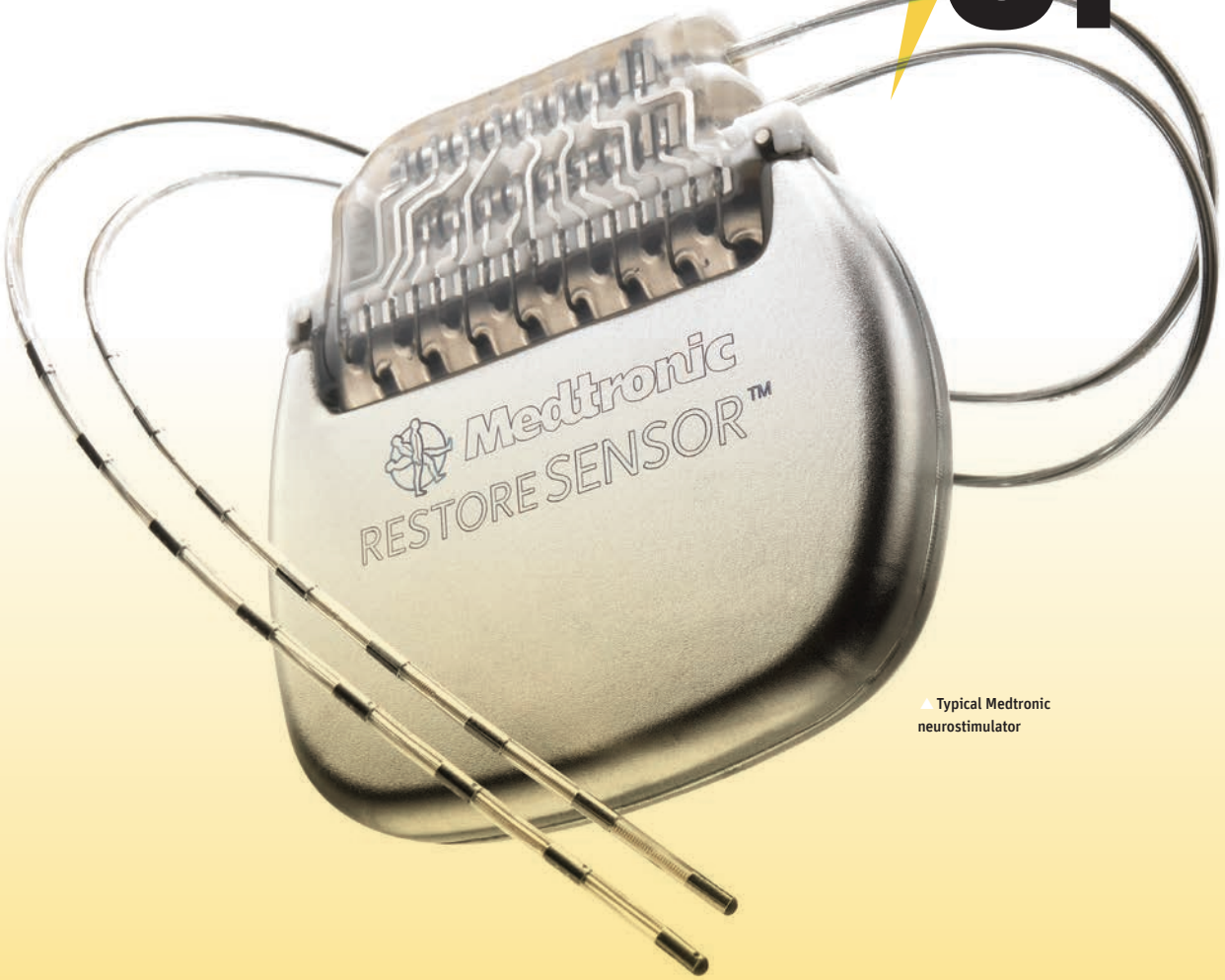
A long-term goal is a project called Optimeyes Embedded, which will involve installing the software directly into a laser device so it does not have to run on a separate computer. Instead of having to connect it to a laptop, the laser instrument itself would make proposals for the optimized incision for the patient. This will require partnering with an ophthalmology device company.

ISS is excited about Optimeyes and its potential to provide perfect vision for millions of people around the world. Though the focus is on cataracts right now, Optimeyes has the power and capability to be adapted to many different types of eye surgeries to improve the precious gift of vision for so many people whose eyesight is impaired.

Optimeyes has the potential to provide perfect vision for millions of people around the world.

© 2015 ANSYS, INC.

CHARGED UP



Typical Medtronic neurostimulator

Medtronic ensures the safety of recharging subcutaneous medical devices through simulation.

By Venkat Gaddam, Senior Electrical Engineer, Medtronic, Minneapolis, U.S.A.

Neurostimulators that are placed under the patient's skin deliver mild electrical signals to provide pain relief by blocking pain messages before they reach the brain. Unlike oral medications that circulate through the patient's body, the neurostimulator targets the precise area where pain is experienced. Patients can try a neurostimulator to see if it relieves their pain before committing to long-term therapy; the device can be surgically removed later if the patient decides to pursue a different treatment. The batteries of rechargeable neurostimulators are recharged by

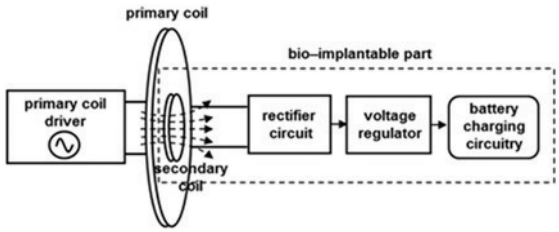
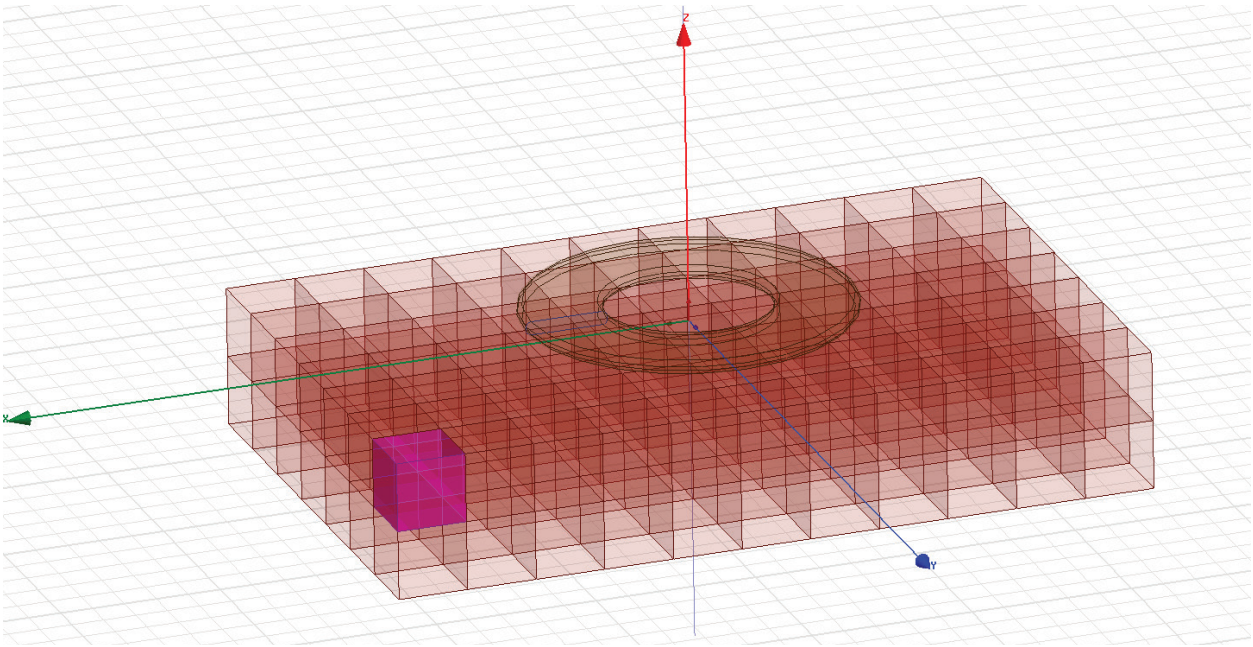


Diagram of recharger and neurostimulator

© 2015 ANSYS, INC.



▲ 10-gram tissue model in ANSYS Maxwell with recharger coil

low-frequency inductive energy transfer using a recharger that is attached to the patient's belt. The recharger emits a non-radiating magnetic field ranging from 3 kHz to 300 kHz that penetrates human tissue and the implanted device's sealed metal enclosure for communication and recharging.

Depending upon the operating configuration, wireless power transfer devices operating at frequencies above 9 kHz are subject to Part 15 and/or Part 18 of Federal Communications Commission (FCC) rules. Medical device manufacturers routinely file Office of Engineering and Technology Laboratory Division Knowledge Database (KDB) inquiries with the FCC to obtain further guidance for wireless power transfer compliance evaluations. As a result of one such inquiry, Medtronic — the world's largest medical technology company — was asked to demonstrate radio frequency (RF) exposure compliance for a wireless power transmitter.

The cost and time required to build a test rig capable of measuring specific absorption rate (SAR) — the rate at which energy is absorbed by the human body when exposed to an RF electromagnetic field — from the recharger is quite high. Medtronic was able to avoid these costs and delays in developing its neurostimulators by using ANSYS Maxwell electromagnetic field simulation software to simulate the operation of the recharger and predict SAR in local body tissues. Simulation showed that SAR generated by the recharger was far below existing FCC limits; the FCC accepted the simulation results for certification of the neurostimulator recharger.

MEDTRONIC AND ANSYS
ansys.com/91charged

TRANSCUTANEOUS RECHARGE INDUCED SAR

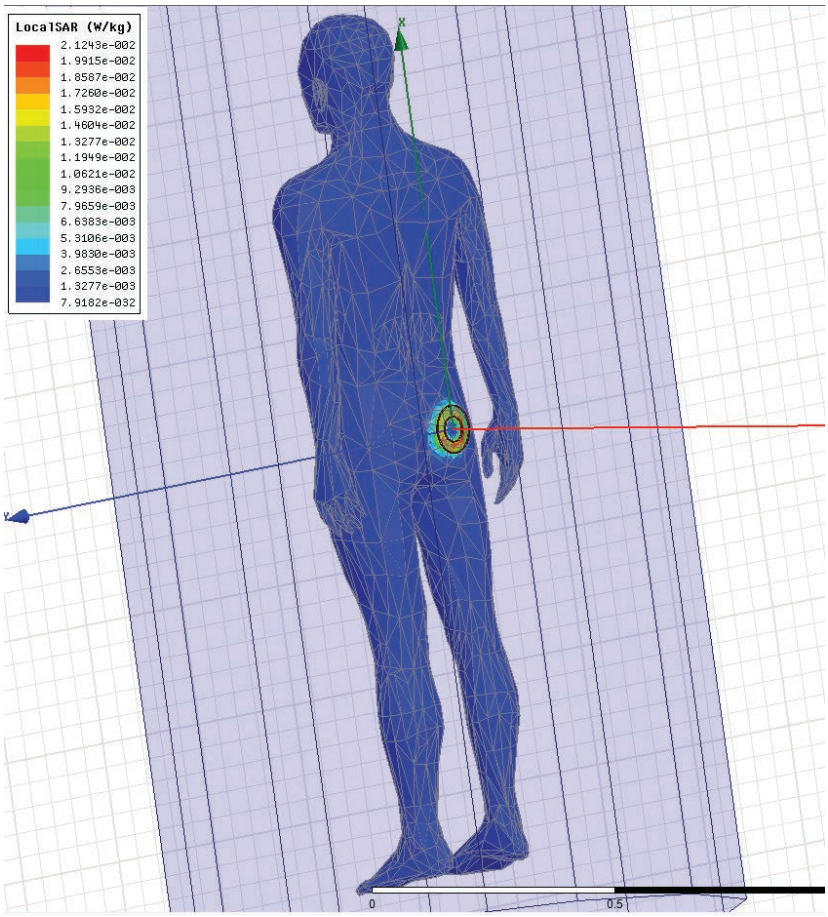
The existing FCC RF exposure requirement prescribed by §2.1093(d)(2) requires a SAR exposure limit of 0.08 W/kg as averaged over the whole body, and a spatial peak SAR not exceeding 1.6 W/kg as averaged over any 1 gram of cube-shaped tissue. SAR is the variable typically used to quantify the effects on tissue exposure to RF signals (defined as the time derivative of the incremental energy absorbed by an incremental mass contained in a volume of given density). Spatial peak SAR is determined by calculating the SAR values in the neighborhood of the electromagnetic source. The domain is then divided into cubes of a given size, and the average SAR value in each cube is evaluated. The peak spatial-averaged SAR is determined by the cube with the highest average SAR value.

Medtronic engineers were confident that their recharging system produced low levels of exposure but needed to measure these levels to obtain approval for a new product. There are a number

Medtronic was able to avoid costs and delays in developing its neurostimulators by using ANSYS Maxwell to simulate the operation of the recharger and predict SAR in local body tissues.

© 2015 ANSYS, INC.

Engineers were confident that the recharging system produced levels of exposure far below the limit specified in the regulation but needed to measure these levels to submit a new product.



▲ Coil positioning in ANSYS human body model

of FCC-certified testing organizations that perform SAR measurements on a contract basis, but Medtronic engineers soon learned that these organizations were not set up to run tests at frequencies as low as those used by the neurostimulator recharger.

ESTIMATING SAR WITH ANSYS MAXWELL

Medtronic engineers used ANSYS Maxwell to estimate the SAR values generated by the recharger coil, with the expectation that the FCC would accept accurate, validated simulation results in lieu of physical testing. They selected Maxwell because the tool makes it easy to set up the model and mesh, and solution times are relatively short. Medtronic engineers employed human tissue models that are available with Maxwell, including muscle sectioned into 10-gram cubes and muscle sectioned into 1-gram cubes. They also used a section tissue model containing skin, fat, fascia and muscle layers. The engineers specified the strength and geometry of the magnetic field generated by the charger. Maxwell adaptively

Model Type	Mass for averaging (g)	Peak spatial averaged SAR (mW/kg)	Peak local SAR (no averaging) (mW/kg)
Muscle	10	9.953	24.877
Muscle	1	15.63	24.965
Muscle (swept cube)	1	15.68	25.09

▲ SAR values based on the simulation results

© 2015 ANSYS, INC.

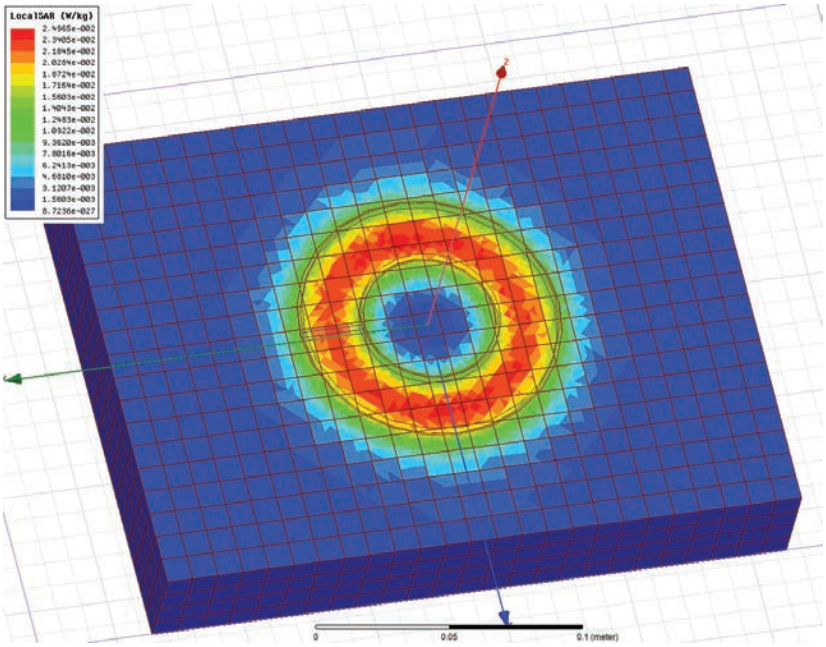
generated an appropriate mesh for solving the problem and used the finite element method to calculate the quasi-static electromagnetic field throughout the solution domain.

Medtronic engineers worked with ANSYS support engineers to create a simple Visual Basic script that calculated SAR values based on the results of the simulation. To calculate the peak spatially averaged SAR, the script calculated the SAR at every element in a 0.25-meter by 0.19-meter by 0.04-meter tissue section. Each 10-gram cube had 2.15-centimeter edges. The peak SAR values as predicted by Maxwell simulation were much lower than the current FCC limits.

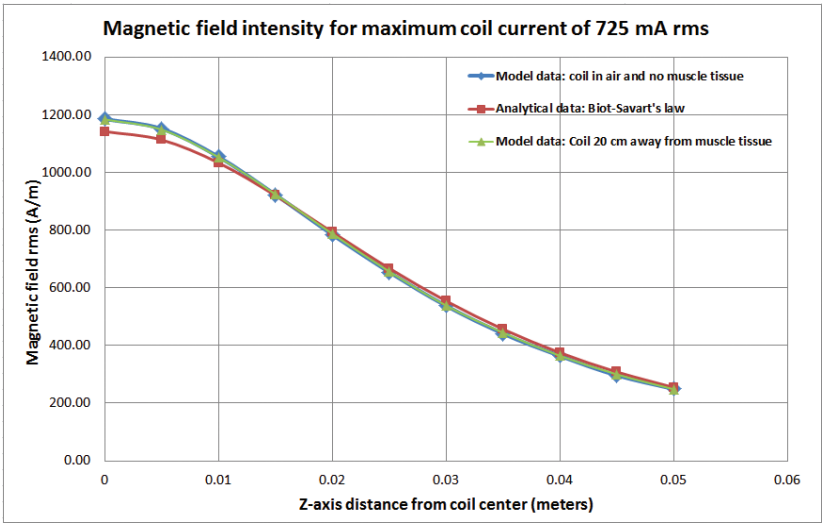
VALIDATING THE SIMULATION

Medtronic engineers used three different methods to validate the accuracy of Maxwell's predictions. First, they created a very simple model and hand-calculated the magnetic field with the Biot–Savart equation — which relates the magnetic field to the magnitude, direction, length and proximity of the electric current — and at the same time performed the calculation with Maxwell software. Second, they set up a simple physical test using a NARDA Safety Test Solutions® electric and magnetic field probe analyzer to measure the magnetic fields generated by the recharger, and then compared these measurements to a Maxwell simulation. Finally, engineers used ANSYS HFSS 3-D full-wave electromagnetic field simulator to simulate peak 10-gram–averaged and peak 1-gram–averaged SAR values. The Maxwell simulations generated results within a few percent of those produced by each of the validation methods.

Medtronic included the Maxwell simulations as part of its new product submission to the FCC. Medtronic engineers were further able to demonstrate that the SAR value would not have been any larger if the tissue geometry in the model had been divided into a different set of cubes. This was addressed by moving a cube around the tissue geometry in discrete step sizes and calculating the average SAR value at each possible position for the cube. Sweeping the cubic volume determined the average SAR value of every possible cube within the tissue volume of interest. The results showed that the maximum possible peak average SAR value for a 1-gram cube is 15.68 mW/kg, less than 1 percent higher than the value in the model that was arbitrarily partitioned.



▲ 1-gram muscle tissue Maxwell model



▲ Validating the model against analytical and physical testing results

Engineers used three different methods to validate the accuracy of Maxwell's predictions.

Medtronic determined that ANSYS Maxwell provides a fast and relatively simple method of measuring 10-gram and 1-gram peak spatially averaged SAR as required to comply with FCC regulations for recharging devices. The FCC accepted the simulation results, saving the company a considerable amount of time and money that would have been required to obtain the same data using physical testing.

© 2015 ANSYS, INC.

I HEAR YOU



Simulation improves hearing aid performance while saving time and money by quickly iterating through design alternatives.

By Casey Murray, Senior Radio Frequency Design Engineer, Starkey Hearing Technologies, Eden Prairie, U.S.A.

▲ Starkey Hearing Technologies' Halo™, a made-for-iPhone® hearing aid

Today, we usually think of smart watches and fitness devices as the premier wearable electronics, but hearing aids are wearable electronic devices that have long improved quality of life for millions (if not billions) worldwide. The hearing aid market has rapidly evolved recently with the introduction of wireless hearing aids that incorporate settings that can be changed while being worn; the aids also can communicate with electronic devices such as smartphones. Design of these products is complex because the antenna and other wireless components must be integrated inside a tiny package. Conforming the antenna to fit these constraints can degrade antenna performance and creates the potential for near-field coupling effects with other electronic components. Starkey Hearing Technologies RF engineers overcame these challenges by using an ANSYS HFSS simulation tool to optimize antenna design while considering the effects of other components and the wearer's body. The result is substantial improvement in hearing aid performance combined with reductions in product development time and expenses.

DESIGN CHALLENGES

With the advent of Starkey Hearing Technologies' 900 MHz wireless hearing aid technology, hearing aid wearers have new options to wirelessly connect with multimedia devices and easily change hearing aid settings. Accessories connecting hearing aids to electronic devices such as televisions and

Manufacturers are adding wireless technology while hearing aids are becoming smaller than ever. This creates major design challenges.

© 2015 ANSYS, INC.

Starkey Hearing Technologies increased the use of HPC licenses to reduce simulation time by over 90 percent.

smartphones are a few examples. These hearing aid products also utilize wireless accessories that enable users or medical professionals to make adjustments to the hearing aid without having to remove it from the ear. Beyond accessory communication capabilities, 900 MHz signals are used for communication between the hearing aids (ear-to-ear communication). This technology allows user controls on the left and right hearing aids to be configured for different operations (e.g., left is for volume, right is for memory) since the hearing aids will remain synced via ear-to-ear commands.

In addition to Starkey Hearing Technologies' 900 MHz products, Halo™, a made-for-iPhone® 2.4 GHz wireless product, was recently introduced to the market. Halo communicates directly with an iPhone using Bluetooth® wireless technology, so users can stream telephone calls and music to their hearing aids and control their experiences through an app on their phones.

Manufacturers are adding wireless technology and other features while hearing aids are becoming smaller than ever. This creates major design challenges. The hearing aid consists of many components, including microphones, a flexible printed circuit board, a battery, a receiver, an antenna and, in many cases, a telecoil. The flexible printed circuit board incorporates more than 60 different components and integrated circuits. The RF designer must manage all these components in a constrained space where the potential for the performance of the antenna to degrade is very high. Traditionally, antennas were designed based on textbooks and literature, but that approach addresses only very simple geometries and does not consider potential interactions generated by other components in the hearing aid and the wearer's body. The toughest challenge involves hearing aids that are designed to fit inside the ear because of their exceptionally small size. The outer package for these products is custom-designed to fit the wearer's ear. This means that the electronics must be small enough to fit into the smallest ear size that the hearing aid is designed to accommodate.

SIMULATION PROCESS

Starkey Hearing Technologies RF designers address these design challenges by using ANSYS HFSS to simulate a wide range of design alternatives that take into account the actual geometry of the antenna, components within the hearing aid, and the user's body. Engineers must generate concept designs for the antenna to fit within the packaging constraints. The greatest modeling obstacle is meshing large features, such as the human head, in conjunction with small features, such as hearing aid circuitry. Starkey Hearing Technologies engineers worked closely with ANSYS application engineers to develop a process that creates consistently meshable models. The Starkey Hearing Technologies team has also developed scripts for pre-processing, running simulations and post-processing that ensure simulation results are comparable across different designers and projects.

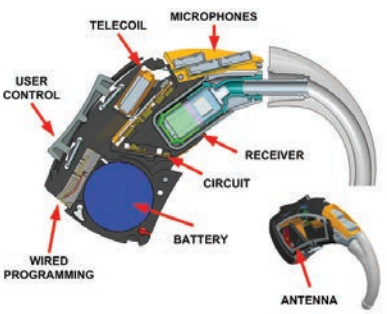
The simulation sequence normally begins with modeling the antenna alone as a subsystem, and then modeling the antenna with other hearing aid components. Finally, the antenna and hearing aid are simulated in place on the wearer's head as a full system to understand how the absorption of power by the wearer's body affects antenna performance. Engineers evaluate the ability of proposed designs to meet performance requirements despite variations in the wearer's head size, ear shape and position in which the hearing aid is worn. Simulation lets engineers explore the design's sensitivity to parameters such as antenna topology, the line width of the copper that makes up the antenna, and antenna excitation locations. Engineers can easily add and remove components of the hearing aid from the model to understand their impact on antenna performance.



▲ For hearing aids worn in the ear, all the components must fit within a package that accommodates the smallest ear size.



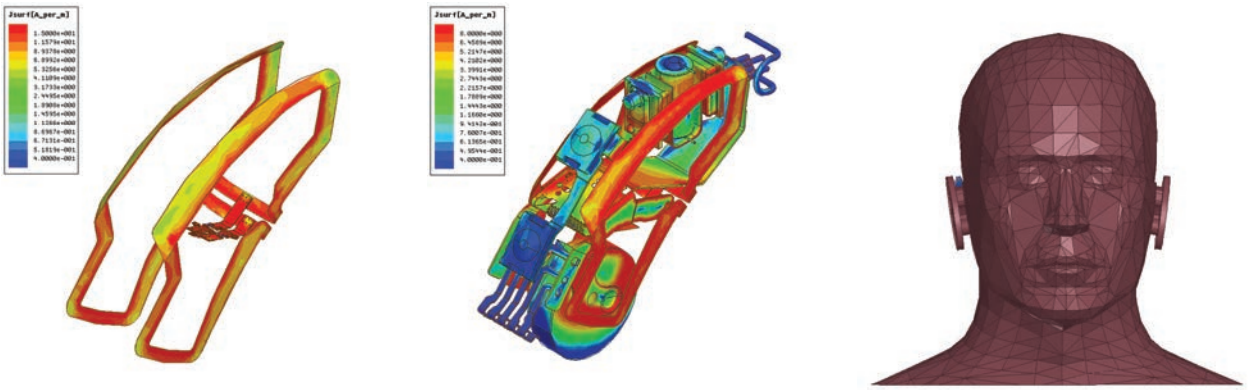
▲ Starkey Hearing Technologies' SurfLink® Mobile accessory device communicates with a hearing aid.



▲ Hearing aid components

STARKEY HEARING TECHNOLOGIES AND ANSYS
ansys.com/91hear

© 2015 ANSYS, INC.



▲ Typical simulation sequence (from left to right): antenna only, antenna with some hearing aid components, antenna with some hearing aid components on head

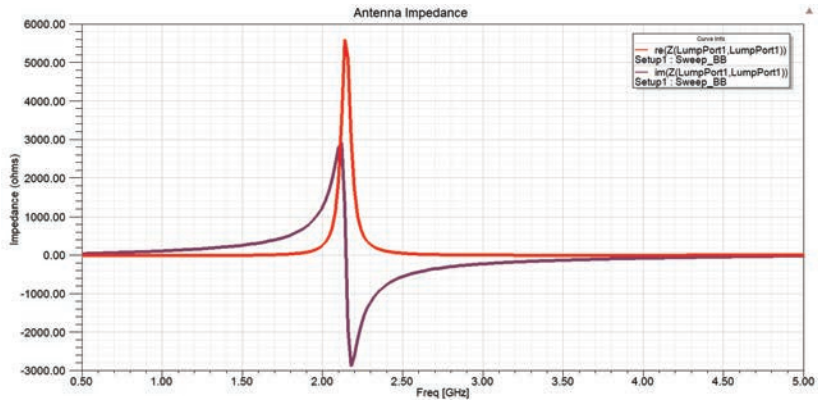
ITERATING TO AN OPTIMIZED DESIGN

The antenna design process typically starts with developing a link budget from which hearing aid performance requirements are derived. These are defined by two key metrics: total radiated power (TRP) (for hearing-aid-to-accessory communication) and receive sensitivity (for accessory-to-hearing-aid communication). With an understanding of the radio performance connected to the antenna, requirements are derived from these two metrics. Engineers use HFSS to calculate the radiation efficiency of the antenna, which is added to the power sent into the antenna to yield the TRP. There are other key antenna parameters that the designer needs to consider to understand antenna radiation properties. One example is effective isotropic radiated power (EIRP), which is the amount of power radiated at a single angle. This quantity is usually quoted in the direction of maximum antenna gain. The designer can obtain the hearing aid's predicted EIRP by using peak realized gain from HFSS and adding the power incident at the antenna. HFSS enables the designer to make comparisons between different antenna designs regardless of type, size or form based on these key metrics.

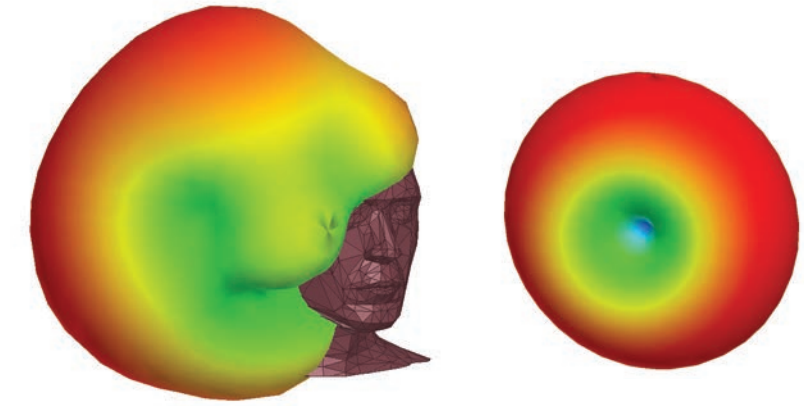
Simulation is also used to diagnose the performance of a proposed design and gain insight into how it can be improved. For example:

- Impedance plots predict the impedance of the antenna across a range of frequencies and are used to match the load impedance (antenna impedance) to source impedance to achieve maximum power transfer.
- Radiation patterns are used to tune the direction in which the antenna power is radiated to minimize power wasted by radiating into the body and maximize power directed toward the accessory or smartphone.
- Current density plots show the potential for interaction between each of the hearing aid components and the antenna.

Simulation results are verified by testing in an anechoic chamber (a room

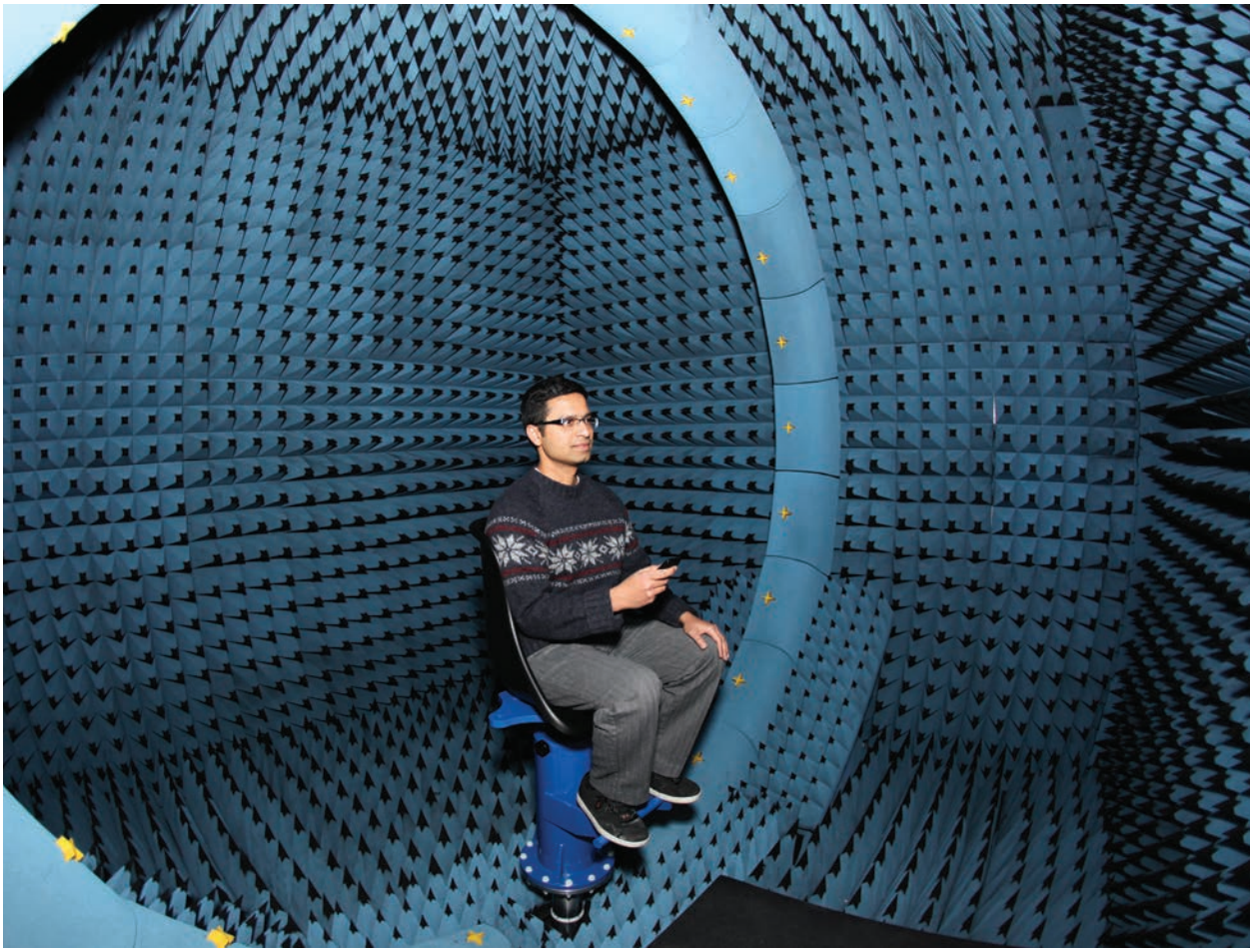


▲ Impedance plot



▲ Radiation patterns

© 2015 ANSYS, INC.



▲ Testing on a human in an anechoic chamber

with walls that completely absorb electromagnetic radiation), isolating the device under test from outside sources of energy. Measured results correlate well with simulation predictions, typically within 1 dB to 3 dB.

In the past, HFSS simulations were performed on high-performance computing (HPC) towers with 8 to 16 cores each taking about 11.5 hours to complete. Starkey Hearing Technologies recently transitioned to an HPC cluster that can be accessed easily by all the company's designers and provides efficient use of ANSYS HPC licenses. The HPC cluster hardware includes one virtual node for scheduling and three computational nodes, each with 48 cores and 192 GB of RAM. The HPC cluster uses 48 cores to reduce simulation time to less than one hour. This greater-than 90 percent reduction in simulation time enables Starkey Hearing Technologies designers to iterate through more design variations in a project time frame, ultimately resulting in a more robust product for the end user.

With the growing use of wireless technology in today's hearing aids, electromagnetic performance is becoming more and more critical to hearing aid performance and reliability. Simulation makes it possible to consider the impact of various antenna designs and component placement strategies in the early stages of the design process. Simulation also enables engineers to consider the effects of different head geometries and wearing positions on the performance of proposed designs prior to the prototyping phase. Simulation saves months of testing time and tens of thousands of dollars in resources for each design project by refining antenna options through virtual prototypes rather than physical prototypes. Using engineering simulation also reduces the risk of expensive mechanical tool iterations. Starkey Hearing Technologies has plans to increase its deployment of simulation by using a wider range of head models and incorporating full-body simulations.

Simulation saves months of testing time and tens of thousands of dollars in resources for each design project.

© 2015 ANSYS, INC.



SIMULATION FOR REGULATION

The Medical Device Innovation Consortium advances the use of simulation in the development and regulation of medical devices.

By Bill Murray, President and CEO, and **Dawn Bardot**, Senior Program Manager
Medical Device Innovation Consortium, St. Louis Park, U.S.A.

Computer modeling and simulation can revolutionize the field of medical devices, bringing innovative new treatments to patients faster and more safely. The Medical Device Innovation Consortium (MDIC) is working to advance the use of validated modeling and simulation in evaluation of regulated medical devices.

To provide state-of-the-art care for patients and keep pace with technology advancements, 21st-century medical device development and clinical trial design must leverage evidence from computer modeling and simulation. Historically, medical devices were evaluated based on three pillars of evidence: bench testing, animal studies and clinical trials. Computer modeling is now the fourth pillar of evidence, providing valuable insights early and often in device design and deployment. MDIC aims to identify and standardize modeling and simulation validation requirements so that computer simulations

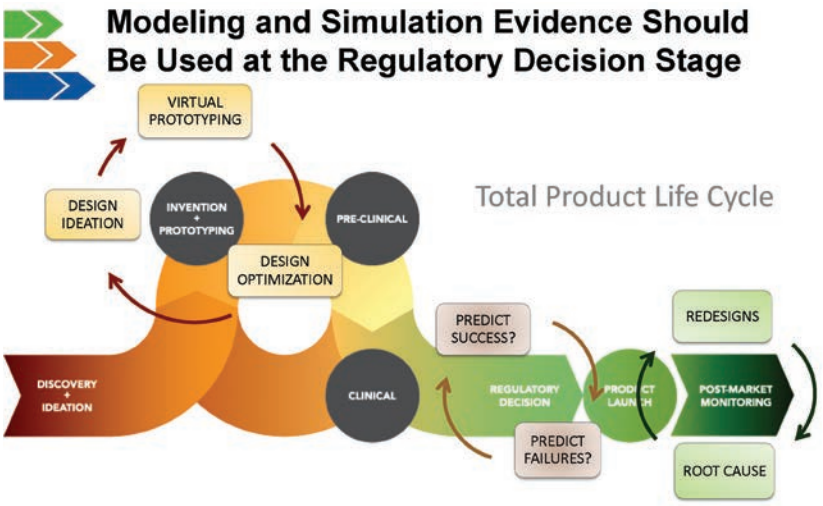
Twenty-first–century medical device development and clinical trial design must leverage evidence from computer modeling and simulation.

© 2015 ANSYS, INC.

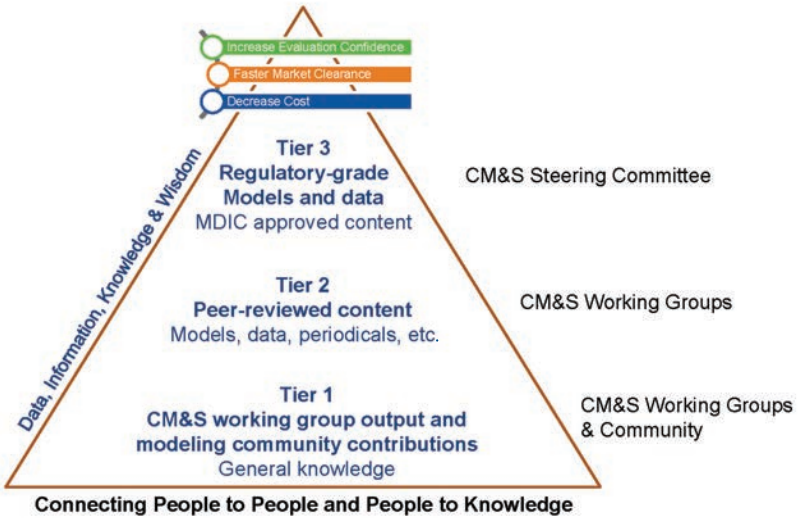
can become an integral source of evidence of medical device safety, efficacy and performance.

MDIC was formed in 2012 as a public–private partnership in which industry, government, nonprofit, patient groups and academia can come together to identify ways to bring medical devices to patients more safely, quickly and cost-effectively. To accomplish this overarching goal, MDIC established four project areas that focus on initiatives directly related to the clinical design, trial and process involved in bringing medical devices to market.

MEDICAL DEVICE INNOVATION CONSORTIUM AND ANSYS
ansys.com/91model



▲ Modeling and simulation evidence can be used to make regulatory decisions instead of just for proof-of-concept and virtual prototyping.



▲ Structure of MDIC repository for data and models

CURRENT ROLE OF MODELING AND SIMULATION

Currently, the medical device industry uses modeling and simulation tools in the initial proof-of-concept and prototyping phases. By creating these models earlier than ever before, engineers are able to evaluate candidate designs before building and testing prototypes. They can perform virtual tests early in the development process rather than waiting for prototype creation in a later development phase.

MDIC sees broad value in modeling and simulation. The consortium created the Computational Modeling & Simulation (CM&S) project to develop the tools and methods needed to extend the use of modeling and simulation throughout the total product lifecycle. The next big opportunity for computer models is at the regulatory decision phase of the lifecycle. MDIC builds confidence in these models' validated, predictive capabilities so that they can be reliably used to make regulatory decisions. This will bring a return on investment through reduced reliance on animal studies and bench testing. Predictive simulations are the gateway to personalized medicine.

The next big opportunity for computer models is at the regulatory decision phase of the lifecycle.

PRIORITIES FOR INCREASING USE OF MODELING AND SIMULATION

The CM&S project steering committee members share a vision of fostering medical innovation, assessing new and emerging technologies, and developing new ways of using clinical data to evaluate medical devices by conducting work in priority areas. This is achieved through working groups with member representation, including medical device manufacturers, the U.S. Food and Drug Administration (FDA), the National Institutes of Health (NIH), nonprofit groups and organizations with expertise in modeling and simulation. Priority areas include:

- Creating a framework to augment some clinical trials with virtual

© 2015 ANSYS, INC.

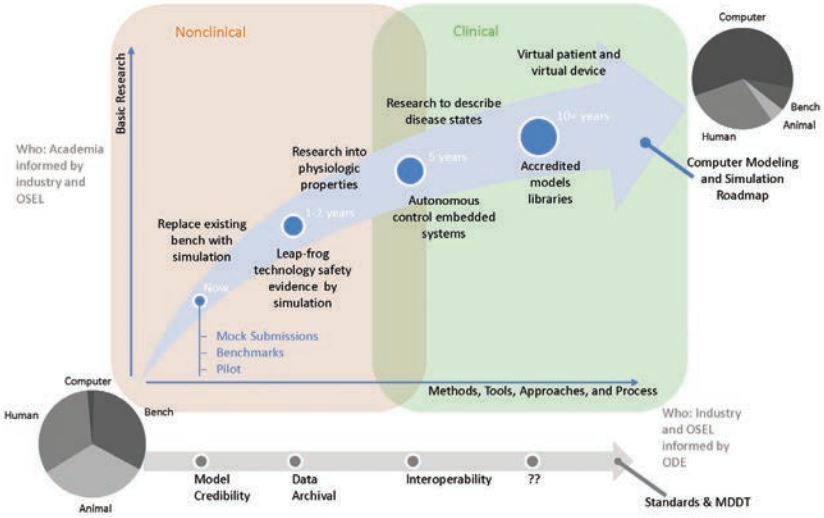
ANSYS and MDIC

ANSYS understands and embraces MDIC's vision and has been an actively participating member of the consortium since its founding. Evolving from a product and technology tool provider to an innovator in healthcare simulation looking at physiological models, ANSYS has played an important role in collaborating with MDIC's CM&S project. Thierry Marchal championed the participation of ANSYS in the MDIC. Marc Horner, ANSYS healthcare industry technical lead, is a member of the Computational Modeling & Simulation Steering Committee and leads the blood damage working group. Horner is also a member of the RF heating and clinical trials working groups. The work performed in these groups will enable medical device developers to generate more groundbreaking ideas, test with greater confidence at lower cost, and bring devices to patients more safely and quickly.

patients, paving the way for smaller, more cost-effective trials

- Identifying opportunities for simulation tools to gather data that cannot be captured by bench testing, particularly in the area of cardiovascular device design
- Developing models to establish the compatibility of implanted devices, such as pacemakers, with magnetic resonance imaging (MRI) by developing generic, regulatory-grade safety simulations that manufacturers can adapt to their specific needs
- Creating a simulation tool that can measure, quantitatively rather than qualitatively, the damage that can occur to blood cells when exposed to a medical device
- Constructing models of both basic healthy human physiology and diseased states to create, design and test new medical products to improve clinical outcomes
- Archiving data and resources on modeling and simulation to make them available to the entire medical device community

In addition to fostering project-specific initiatives and research, MDIC hosts events and gives presentations in national forums to build awareness of modeling and simulation and its importance. The organization is writing a white paper detailing the evolution of modeling and simulation in the medical device industry, current and future applications, barriers to using modeling and simulation in the regulatory process, and next steps. Simultaneously, the consortium is developing mock FDA submissions to produce representative examples of the use of computational modeling and simulation for actual regulatory submissions.



▲ Road map for increasing the use of modeling and simulation evidence

Computer simulations can become an integral source of evidence of medical device safety, efficacy and performance.

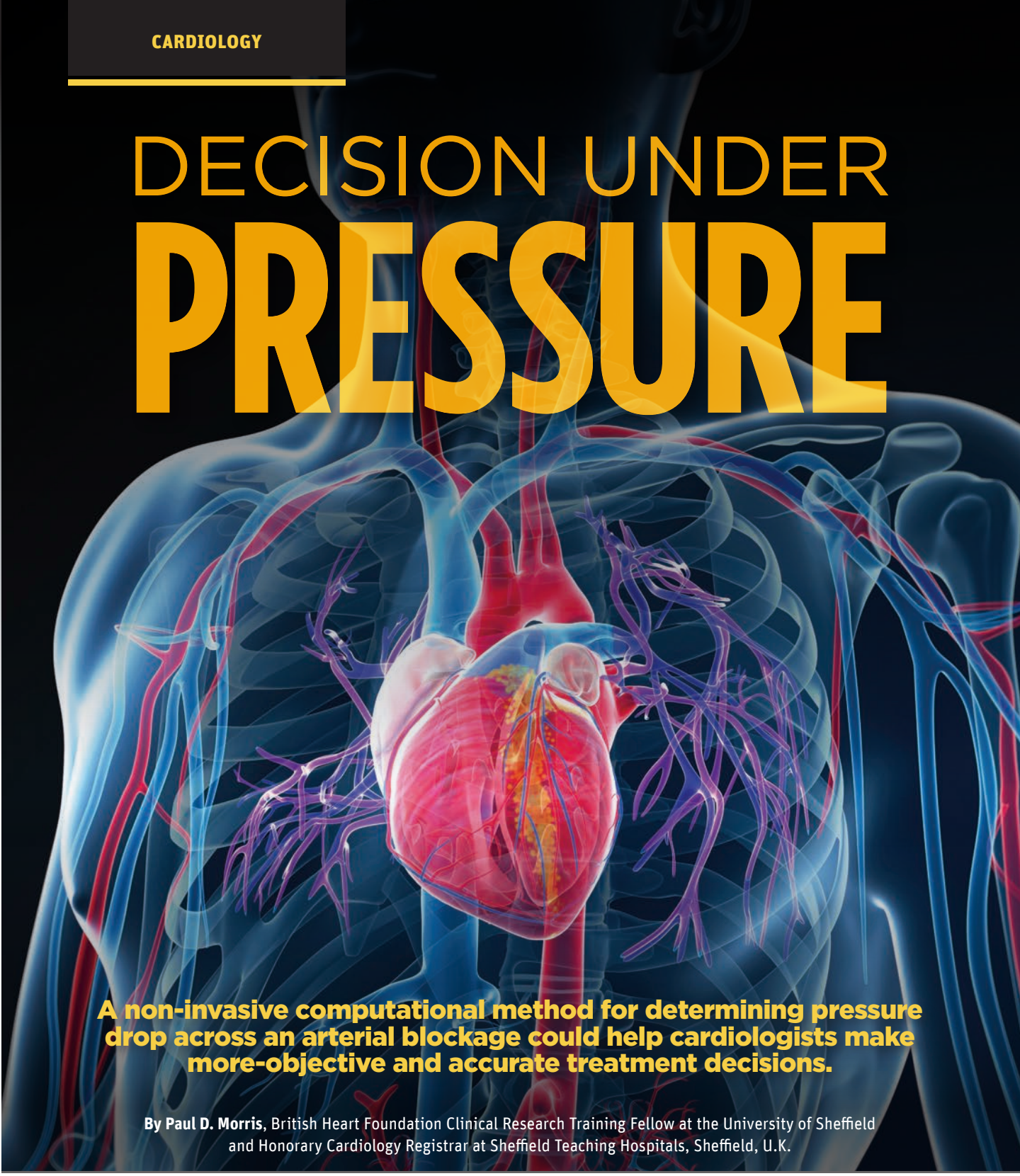
References

[1] MDIC *mdic.org*

[2] Q&A with Marc Horner *mdic.org/spotlight-on-members-marc-horner/*

© 2015 ANSYS, INC.

DECISION UNDER PRESSURE



A non-invasive computational method for determining pressure drop across an arterial blockage could help cardiologists make more-objective and accurate treatment decisions.

By Paul D. Morris, British Heart Foundation Clinical Research Training Fellow at the University of Sheffield and Honorary Cardiology Registrar at Sheffield Teaching Hospitals, Sheffield, U.K.

Coronary artery disease — which is the blockage of arteries that provide blood to the heart — is the leading cause of death in the world today. The World Health Organization notes that around 30 percent of all deaths worldwide are due to cardiovascular disease. [1] Occurrences of this disease are increasing both in numbers and in geographic range, making the development of new diagnosis and treatment options critical.

Thirty percent of all deaths worldwide in 2008 were due to some form of cardiovascular disease.

© 2015 ANSYS, INC.

Clinical cardiologists and simulation experts at the VIRTUheart™ project at the University of Sheffield in the United Kingdom are using ANSYS computational fluid dynamics (CFD) simulation to improve diagnosis of the severity of coronary artery disease in a given patient. Non-invasive modeling of the pressure drop across a lesion (blockage) gives clinical cardiologists a quantitative metric to help them decide whether to perform invasive procedures — insertion of stents or cardiac bypass surgery — or to simply medicate those patients with less severe conditions.

Reducing discomfort, invasiveness and risk, as well as increasing successful treatment outcomes, is of obvious value to patients. The potential monetary savings involved in simplifying the procedure, performing fewer stent procedures, and avoiding costly bypass surgeries are enormous to a world increasingly overwhelmed by healthcare costs.

TO TREAT OR NOT TO TREAT?

When a patient arrives at a cardiac catheterization laboratory complaining of chest pain, an angiogram is performed to get an anatomical picture of the coronary arteries. A constriction in an artery indicates a lesion. Two questions arise: How bad is the constriction, and what action should be taken?

The decision is a subjective one, dependent on the experience and judgment of the cardiologist. Ask several cardiologists for their assessment of the same lesion and you are likely to get several different opinions. A decision is needed quickly.

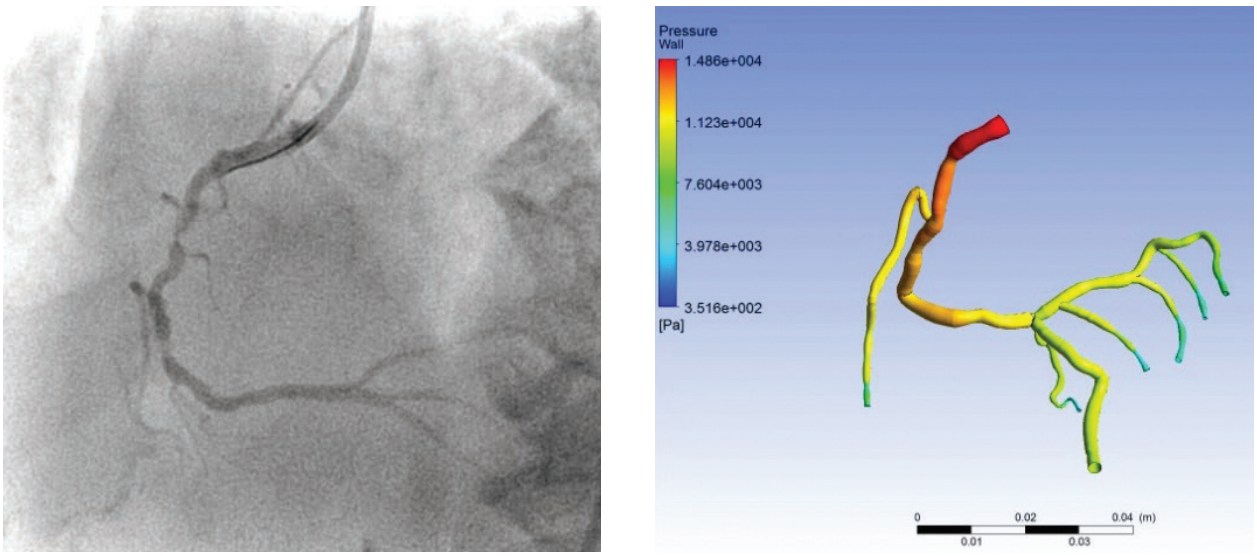
Physiological data — as opposed to the purely anatomical data shown in an angiogram — can be obtained by inserting a wire outfitted with a small pressure sensor into the artery in question and measuring the pressure on both sides of the lesion. The pressure drop across the lesion reveals the severity of the blockage. Dividing the lower pressure by the higher pressure yields a number between 0 and 1 that is known as the fractional flow reserve (FFR). If the FFR is above 0.80, the narrowing is unlikely to cause any clinical symptoms or problems. Values below 0.80 mean that blood flow can become restricted. A high FFR is good news for the patient, while patients with FFR values far below 0.80 are clear candidates for either stent insertion or cardiac bypass surgery. The tough decisions come when the FFR is close to the 0.80 threshold point.

Clearly, the measured FFR value (mFFR) derived from this technique gives the cardiologist an objective value to help in making treatment decisions, placing less reliance on the doctor's subjective judgment based on experience. Unfortunately, the pressure measurement procedure to obtain the mFFR value is performed in less than 10 percent of cases in which stents are inserted in the U.K. Because the procedure involves inserting a wire inside an artery, it adds expense and prolongs patient discomfort, so, although the pressure measurement procedure offers significant advantages, doctors around the world have not adopted this invasive technique in high percentages.

DETERMINING FFR VIRTUALLY

ANSYS solutions are used extensively at the INSIGNEO Institute for

Cardiologists and simulation experts at VIRTUheart are using ANSYS CFD to accelerate and improve diagnosis of the severity of coronary artery disease in a given patient.



▲ Angiogram of coronary artery and its branches (left); virtual model of branched coronary artery structure using ANSYS CFD (right). High pressures shown in red and orange; lower pressures in green and blue.

© 2015 ANSYS, INC.

In Silico Medicine at the University of Sheffield, so it was natural to turn to modeling and simulation of the coronary arteries to try to obtain a virtual, non-invasive FFR value (vFFR™). Such a solution, researchers hoped, would clear the way for higher adoption of this valuable measure in coronary treatment decisions. The potential benefit in improved patient outcomes was the driving force for this investigation.

Researchers conducted a pilot study involving 19 patients with stable coronary artery disease and a total of 35 constricted arteries using ANSYS CFD. [2] They obtained rotational coronary angiograms to give a 3-D picture of the coronary vessels. (Additional investigation following this clinical trial has been done to eliminate the need for rotational coronary angiography, as this instrumentation is less commonly available than standard 2-D angiography instrumentation. Results of this research are not yet available.) The team segmented the angiograms to produce 3-D arterial geometries. Customized software created a surface mesh and volumetric mesh comprising approximately 1.5 million cells. Researchers applied generic pressure and flow boundary conditions, as opposed to patient-specific ones, to the model. They then imported the meshed geometry into ANSYS CFX for CFD simulation of the pressure drop across the lesions and calculation of the vFFR.

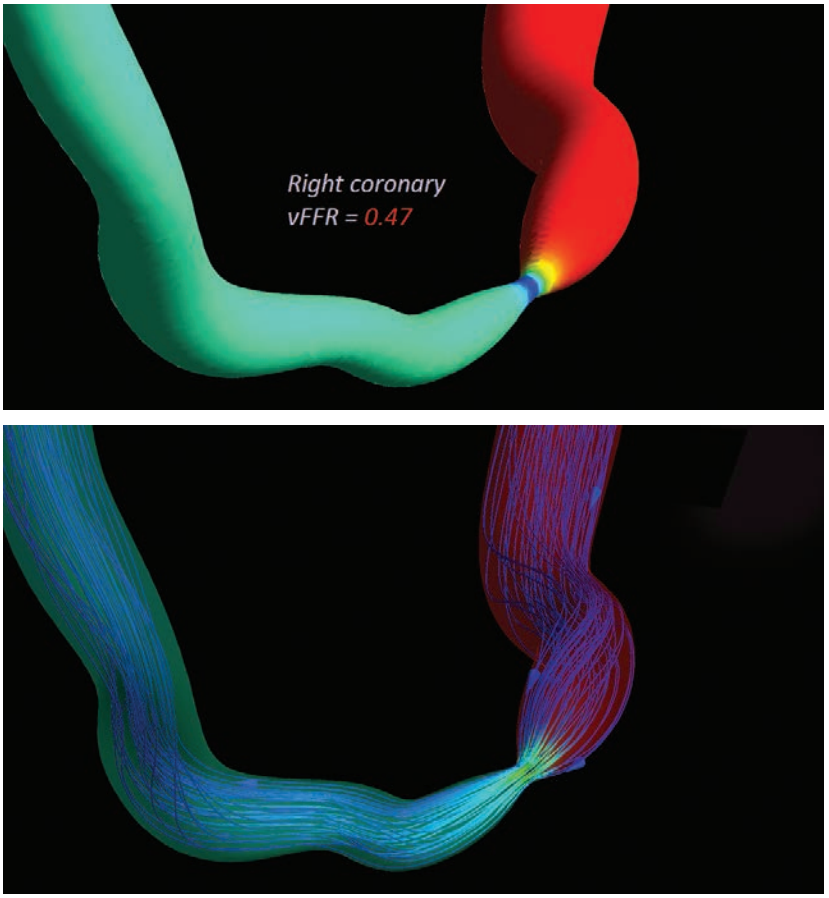
The mFFR and vFFR values obtained for the pilot study were in good agreement. The overall diagnostic accuracy of vFFR was 97 percent. A plot of vFFR versus mFFR had a correlation coefficient of 0.84.

DIAGNOSIS AND PERSONALIZED MEDICINE
ansys.com/91decision

ADVANTAGES OF vFFR

Having the ability to determine FFR virtually should eventually lead to greater adoption of this valuable physiological parameter in determining the appropriate treatment for an arterial lesion. The vFFR method requires only angiogram images and CFD simulation — no invasive insertion of wires with pressure sensors into the patient's arteries. On this basis alone, vFFR could have a huge positive effect on patient outcomes and monetary healthcare savings.

In addition, vFFR provides a pressure profile of the complete arterial system being modeled, which is a great improvement over the single-point pressure drop value that the mFFR method yields. The distribution of pressures in the CFD simulation clearly shows the regions where pressures change the



▲ Coronary lesion with vFFR = 0.47 is clearly a candidate for treatment (top): flow through coronary lesion (bottom)



▲ Angiogram in progress

© 2015 ANSYS, INC.

most. Such comprehensive detail is not available from mFFR, which provides only a single-point pressure measurement.

In cases in which two or three lesions are present in a single coronary artery, the lesions function as part of a system. Cardiologists might be inclined to stent every lesion. However, stenting the first upstream lesion may improve the pressure conditions such that the downstream lesions do not require a stent.

The researchers at VIRTUheart are currently working on extensions to the model that will make virtual stenting possible. For example, the cardiologist might eventually be able to insert stents virtually into the model and see what effect this stent might have on the other lesions and arteries in the system. Being able to preview a treatment in silico before trying it in a human patient should lead to better outcomes and save money by reducing the number of implanted stents.

Even though the pilot study showed that vFFR had a very high diagnostic accuracy of 97 percent, there is room for improvement. The initial study was performed with generic boundary conditions for blood pressure and flow. VIRTUheart researchers are investigating how the use of patient-specific boundary conditions could improve diagnostic accuracy of vFFR even more.

Ultimately, the decision about how to treat a cardiovascular lesion can be a determining factor in patient lifespan and quality of life. The decision to opt for cardiac bypass surgery, perform stent insertion or simply treat with medication can make a big difference. If you are the person lying on the table in the cardiac catheterization laboratory, you want your doctor to have all the best data possible to make his or her treatment decision. VIRTUheart and ANSYS are working together to help your doctor make the best decision for you.

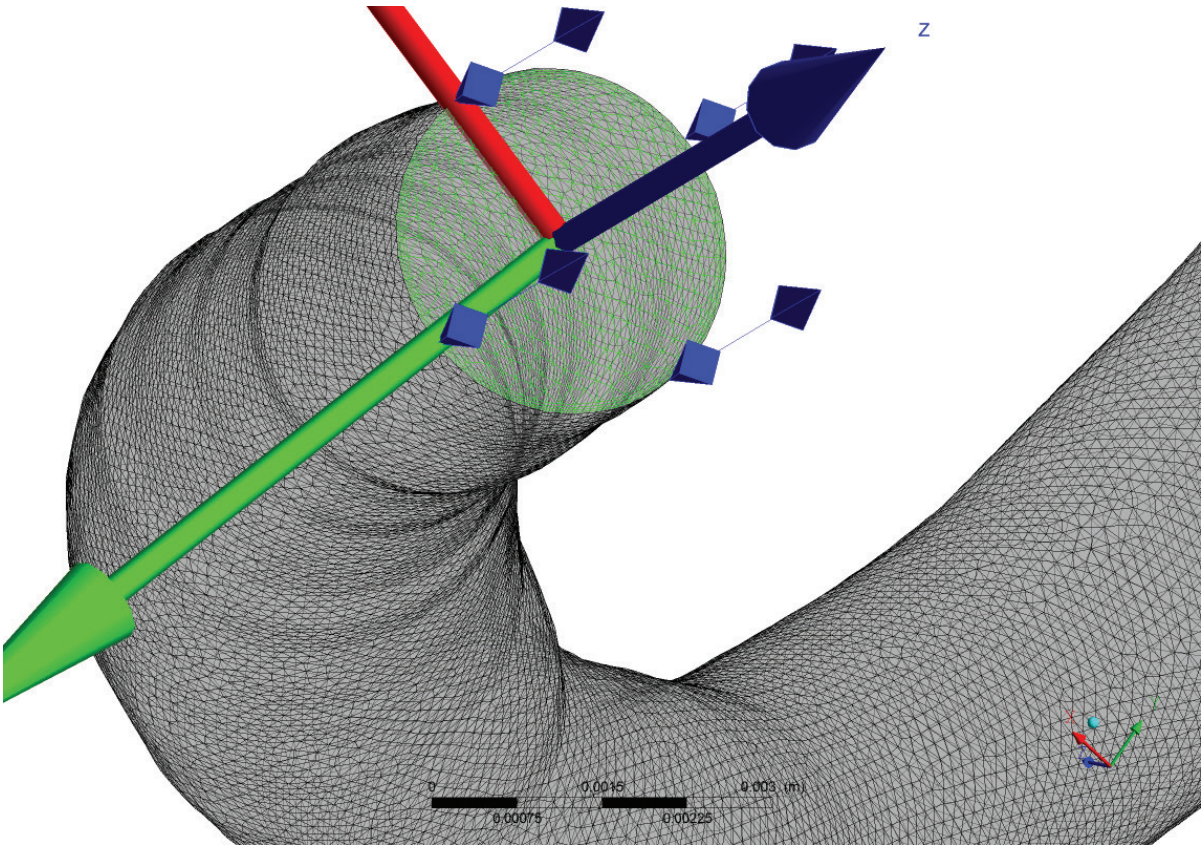
VIRTUheart and ANSYS are working together to help your doctor make the best decision for you.

References

[1] World Health Organization website: *www.who.int/mediacentre/factsheets/fs317/en/*

[2] Morris, P.D.; Ryan, D.; Morton, A.C. et al. Virtual Fractional Flow Reserve From Coronary Angiography: Modeling the Significance of Coronary Lesions: Results From the VIRTU-1 (VIRTUal Fractional Flow Reserve From Coronary Angiography) Study. *J Am Coll Cardiol Intv.* **2013**. Volume 6, Issue 2, pages 149–157.

interventions.onlinejacc.org/article.aspx?articleid=1653763



▲ Meshing applied to coronary artery prior to simulation

© 2015 ANSYS, INC.

THE RIGHT MIX

CFD simulation saves time and money by validating the ability of a single-use mixer design to scale to 5,000 liters.

By Rudolf Pavlik, Director, Product Development, ASI, Millersburg, U.S.A.
Szymon Buhajczuk, Principal CFD Engineer (Canada), and
Mark Goodin, CFD Consulting Engineer (U.S.A.), SimuTech Group Inc., Toronto, Canada



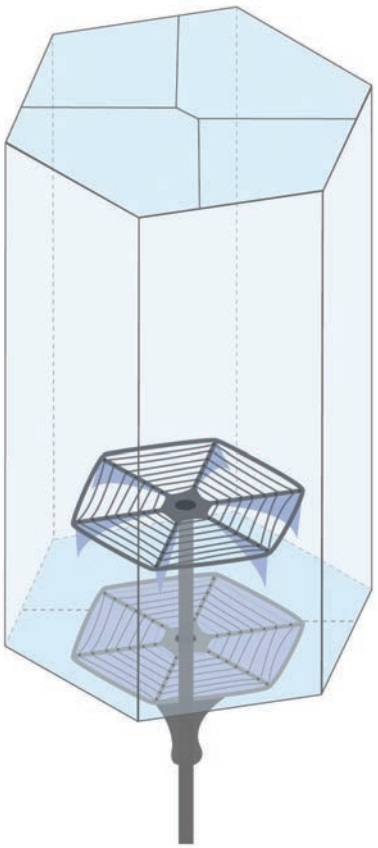
▲ imPULSE single-use mixer

ASI faces the challenge of providing different sizes of its products for various stages of the therapeutic development process.

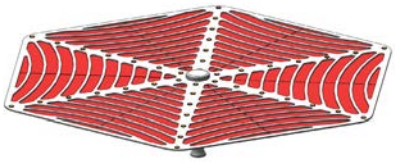
Biopharmaceutical manufacturers continually need to scale up production as they move from small pilot studies to progressively larger clinical trials, then finally into large-scale production as the drug reaches the market. As a provider of single-use systems and bioprocess equipment utilized in biopharmaceutical manufacturing, ASI regularly faces the challenge of providing different sizes of its products for these various stages of the therapeutic development process. Until recently, biopharmaceutical manufacturing facilities relied solely on hard-piped systems, such as stainless steel bioreactors, tanks and piping. ASI has pioneered development of single-use equipment, designed to be employed once and then disposed of. These systems drastically reduce the need for harsh and lengthy cleaning requirements while improving production speed due to quick changeover between batches.

ASI is a leading global provider of advanced single-use systems for the healthcare and life sciences industries. The company's imPULSE single-use mixing series is a unique system that consists of a stainless steel hexagonal mixing vessel and a matching single-use mixing bag. Together, the system can be

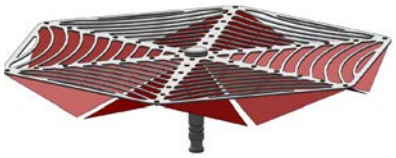
© 2015 ANSYS, INC.



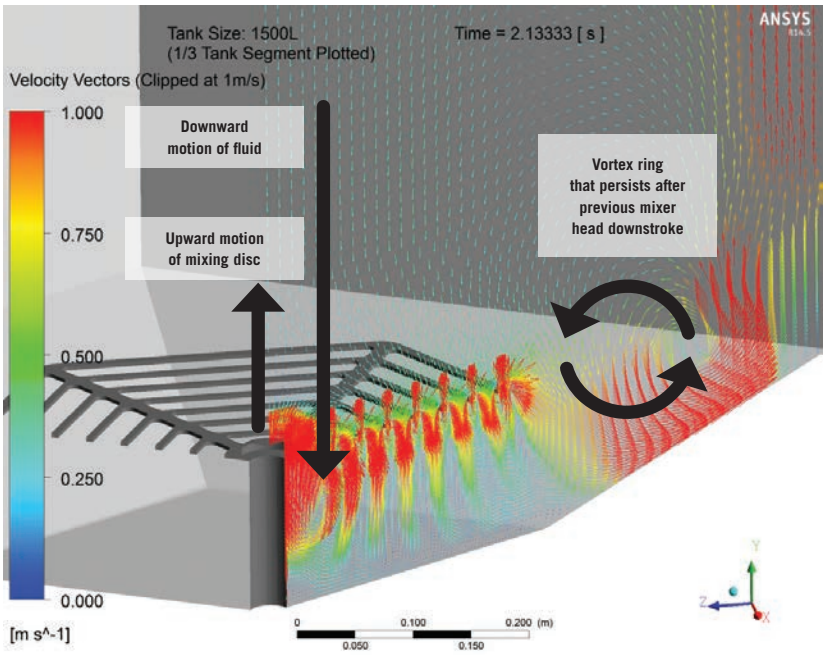
▲ Reusable mixing bag



▲ Flow during mixing downstroke



▲ Flow during mixing upstroke



▲ Velocity vectors show mixing motion.

configured for a variety of end-user mixing applications. The disposable polymer mixing bag is engineered with an integrated mixing disc that consists of multiple slots and film flaps. The flaps open and close as the mixing disc moves up and down within the mixing bag. On the downstroke, the flaps close, and energy is directed to the bottom of the mixing bag and up the sidewalls. On the upstroke, the flaps open, allowing the fluid to flow through the slots, thus producing one-way flow and very effective mixing. Simulation with ANSYS Fluent helped ASI to eliminate the cost and lead time of prototyping, demonstrating that ASI's design could be scaled up to an industry-leading 5,000-liter size while providing the same mixing performance as smaller mixers.

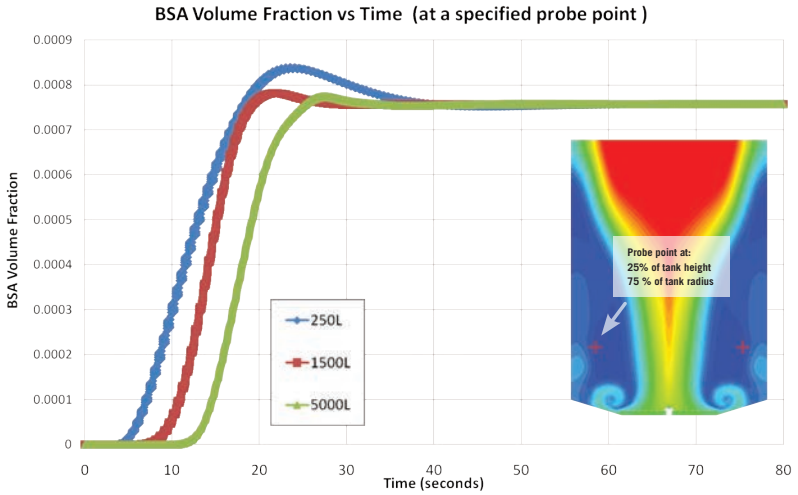
ASI first developed the imPULSE design in a 250-liter (L) size and expanded the portfolio to include sizes from 30 L to 1,500 L. As customers further scaled up their batch sizes, they demanded larger mixers. Although it was not difficult to scale up the mixer, it was a challenge to maintain mixing efficiencies and patterns. The time required to achieve a certain level of homogeneity is critical to the efficiency of biopharmaceutical manufacturing. To sell the larger mixers, ASI needed to prove that mixing time would be consistent in both larger and smaller mixers. The lead time and cost required to build a prototype of the new 5,000-liter mixer was quite high. So ASI investigated the potential for using computational fluid dynamics (CFD) simulation to validate the design of the larger mixer. Besides being faster and less expensive than building and testing a prototype mixer, CFD provides more diagnostic information, such as flow velocities throughout the tank along with shear rate, all of which are useful in diagnosing and improving a mixer design.

Besides being faster and less expensive than building and testing a prototype mixer, CFD provides more diagnostic information.

ASI contracted with consultants from ANSYS channel partner SimuTech Group, a supplier of engineering simulation software, support, training, consulting and testing services. The team used ANSYS Fluent to simulate the motion of the mixer discs. Fluent's dynamic layering method

CUTTING DESIGN COSTS: HOW INDUSTRY LEADERS BENEFIT FROM FAST AND RELIABLE CFD
ansys.com/91right

© 2015 ANSYS, INC.



▲ BSA volume fraction versus time for different tank size



250 L



1,500 L



5,000 L

▲ Flow circulation patterns for different tank sizes

CFD simulation saved hundreds of thousands of dollars, providing characterizations that apply to the overall scalability of ASI's products and significantly reducing the need for building and testing prototypes.

adds or removes layers of cells adjacent to a moving boundary based on the height of the layer bordering the moving surface, which enables simulation of devices with complex moving parts. The dynamic layering method allows users to specify an ideal layer height on each moving boundary. The layer of cells neighboring the moving boundary is split or merged with the layer of cells next to it based on the height of cells in the adjacent layer. This unique approach to simulating a moving boundary eliminates accuracy problems, which are caused by cell shape deformation.

SimuTech engineers simulated performance of the bag in mixing two different particles: salt and bovine serum albumin (BSA). The software enabled engineers to customize material properties to model the properties of each particle type. The simulation showed that the flow traveled up along the outer walls, crossed over at the top of the tank, and returned in a downward moving column. This was expected since the mixing disc, located in the center of the bag, was designed to push the fluid on the downstroke, but not on the upstroke due to the opening of the membrane film flaps. The result is that during the downstroke bulk flow is accelerated, but on the upstroke a more complicated local mixing flow pattern is formed around the mixing disc. A complicated local mixing flow pattern is evidence of the random and aggressive mixing patterns this disc creates. The aggressive behavior creates a turbulence that generates random patterns, which provide additional paths for the solutions and bulk flow to conjoin.

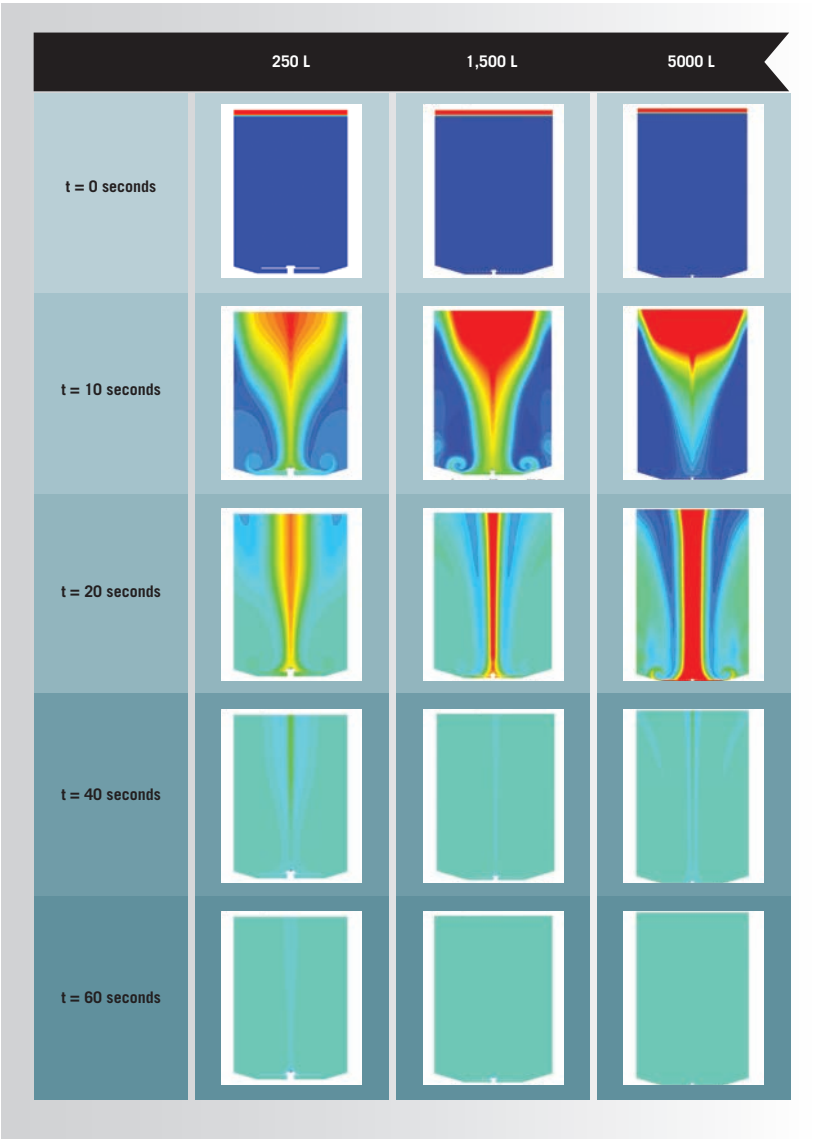
The simulation showed that localized flow near the mixing disc changes significantly depending on its position in the stroke cycle. On the downstroke, with the membranes closed, the flow is pushed outward toward the tank walls at a high velocity. A vortex ring forms around the periphery of the mixing disc, which is beneficial to mixing and persists even after the mixing disc starts to move up again. The vortex generally follows

© 2015 ANSYS, INC.

the bulk flow, so the circulation pattern migrates toward the walls. When the mixing disc is moving up, the bulk of the fluid in the center column continues to move down, but now the mixing disc opposes this motion. With the membranes/holes open, the flow is free to bypass the mixing disc by moving through these holes, which further agitates flow. The localized vortices illustrated in the CFD results generate turbulence with the ability to mix even difficult powder/liquid solutions at a rate that will enhance conjoining the bulk fluid and powder/liquid product solution. The localized vortices near the disc show that air is not being entrained or pulled in; only unmixed product is pulled in through the submerged disc.

To compare and predict scalability across various sizes, SimuTech engineers compared flow patterns of three different-sized mixers — 250 liters, 1,500 liters and 5,000 liters — to determine whether or not the tanks behave similarly. The results showed that flow patterns were largely unchanged in the larger devices as compared to the 250-liter baseline. Within a few seconds, all the tanks establish the pattern of flow moving up along the outer walls and down through the center column.

CFD simulations saved ASI hundreds of thousands of dollars.



▲ BSA particle mixing patterns for different tank sizes

The mixing patterns were observed directly through multiphase simulations with salt and BSA particles present in the tank. These results showed that at 6 seconds all three mixers had significantly suspended salt into the fluid. For the smallest equipment size, significant concentrations of salt were present at the top of the tank; even for the largest sizes, significant concentrations were present two-thirds of the way up the height of the tank. The near-neutrally buoyant BSA particles, which started in a thin layer at the top of the bag, were drawn down in the center column of descending fluid, then agitated by the mixing disc and eventually dispersed throughout the entire tank. The simulations showed that within 60 seconds, the concentrations throughout the tank were relatively uniform.

To quantify the mixing of BSA particles over a longer period of time, researchers created a monitor point in the three tanks. This point was placed 25 percent of the way up the height of the tank at a radial position of 75 percent. The results showed that the smaller tanks mixed faster than the larger tanks, but within practical limits, all tanks mixed very quickly. Within 60 seconds, the volume fractions in all of the tanks stabilized at about the same level. Overall, while slight differences were present in time scales in the different tanks, the tanks all scaled well, since they all mixed in less than a minute and displayed similar mixing patterns for the specific CFD testing conditions.

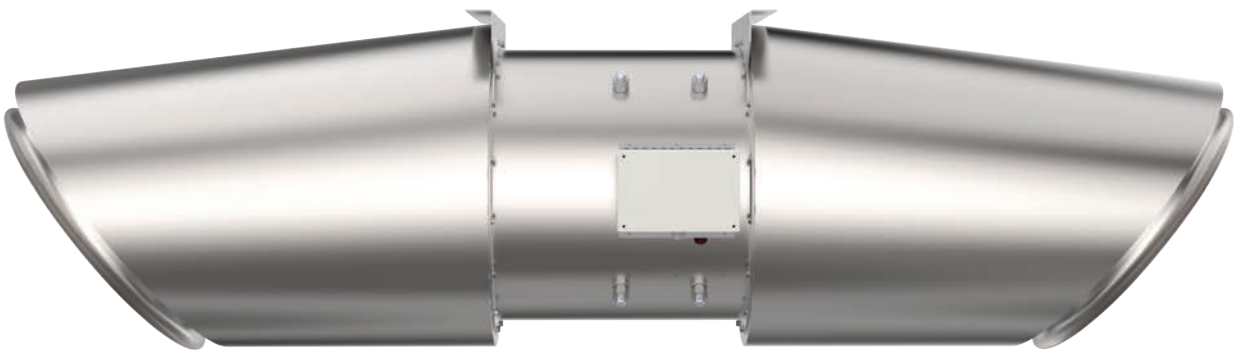
Because ASI engineers confirmed the simulation predictions with actual data in three sizes, they can draw a correlation between the actual and simulated data for application across the company's entire portfolio of mixing products. Overall, CFD simulation saved hundreds of thousands of dollars, providing characterizations that apply to the overall scalability of ASI's products and significantly reducing the need for building and testing prototypes.

© 2015 ANSYS, INC.

ENERGY-EFFICIENT TUNNEL VENTILATION

A leading ventilation company develops jet fans that are smaller and up to 30 percent more energy efficient.

By Fathi Tarada, Managing Director, Mosen Ltd., U.K.



Improving the efficiency of turbomachinery, including jet fans used in tunnel ventilation systems, is essential in combating the volatile cost of fuel and reducing emissions of greenhouse gases. Energy-efficient equipment is also more attractive to worldwide transportation authorities.

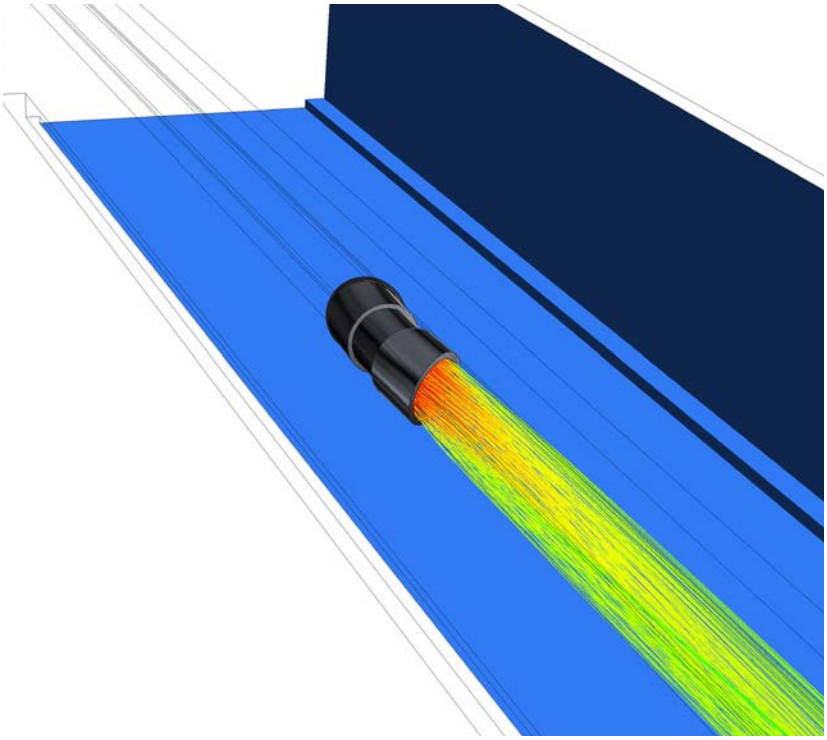
Longitudinal tunnel ventilation systems provide airflow along the length of a tunnel; they are needed to satisfy air quality requirements or to control smoke movement in case of a fire. While there are many different types of ventilation systems available, the most widely used type employs jet fans. In addition to being energy efficient, jet fan ventilation systems should be compact so as not to protrude into traffic space. System designs must also allow for closely spaced fan installation, to reduce cabling costs, and quiet operation that conforms to guidelines.

DESIGNING THE SOLUTION WITH ANSYS

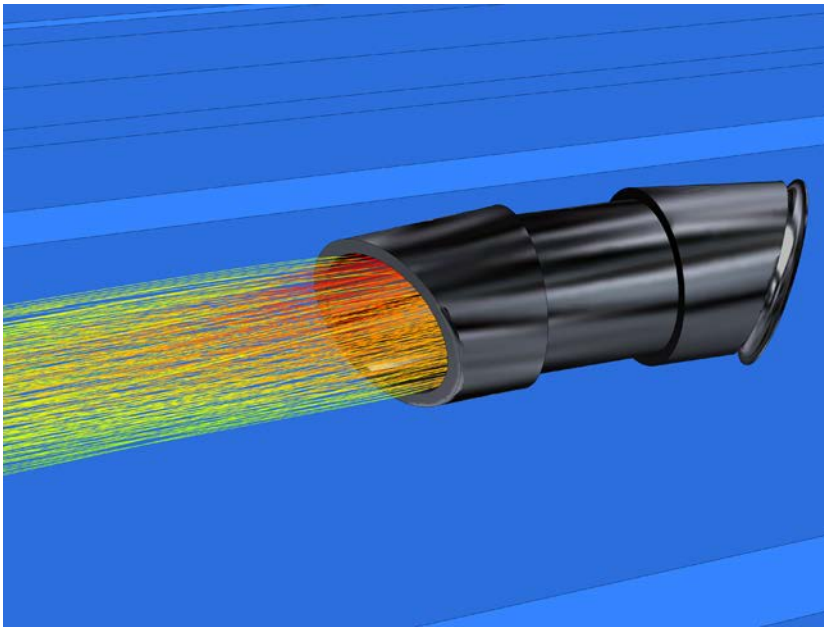
Mosen Ltd., a firm of consulting engineers that develops energy-efficient tunnel ventilation systems, uses ANSYS software to optimize systems design and meet all the other requirements.

One of the most important factors in improving efficiency of tunnel ventilation systems is minimizing the Coanda effect.

© 2015 ANSYS, INC.



▲ Particle tracks colored by air velocity for airport tunnel project. The jet of air is turned away from tunnel walls and soffit by the MoJet.



▲ Particle tracks colored by air velocity; airport tunnel project

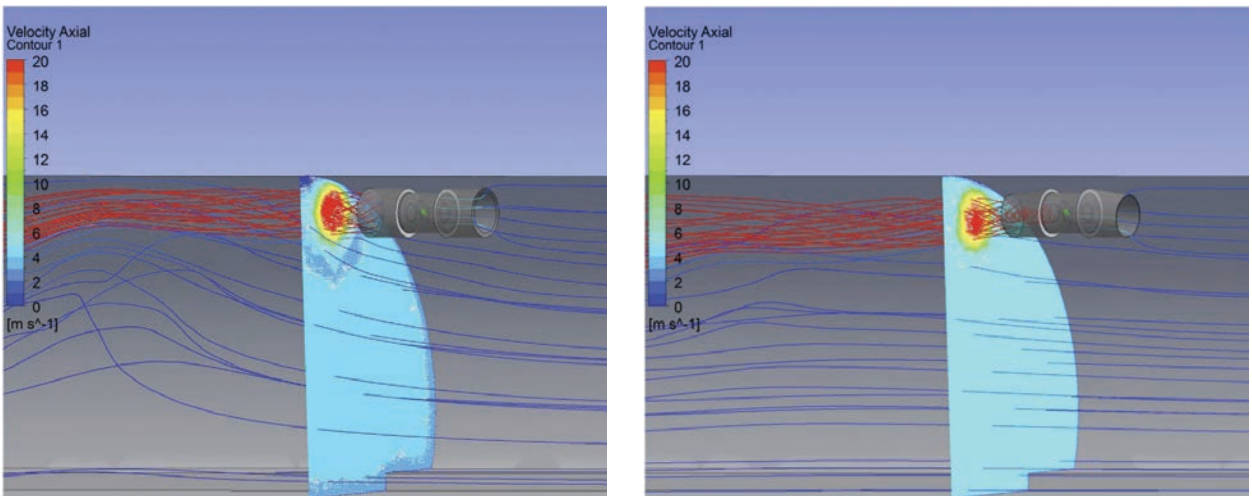
One of the most important factors in improving efficiency of tunnel ventilation systems is minimizing the Coanda effect. "Coanda effect" refers to the tendency of the air steam produced by each jet fan to be drawn to nearby walls, leading to increased friction and significantly increasing the energy required to move the air. This effect occurs because the jet flow close to the wall decelerates, creating a pressure difference across the jet that reinforces its attachment to the wall, even at high velocities. Between 15 percent and 27 percent of the energy used by jet fan ventilation systems is wasted due to this increased aerodynamic friction. Since tunnel ventilation systems consume large amounts of energy, with some installations demanding several megawatts at full load, reducing the Coanda effect goes a long way toward improving energy efficiency.

Engineers at Mosen used ANSYS CFX fluid dynamics software to simulate the airflow, both inside their innovative jet fan — the MoJet® — and within tunnels. To combat the Coanda effect, engineers used ANSYS Workbench to parametrically experiment with different inlet and outlet nozzle designs, all geared toward reducing losses, accelerating flow and diverting it away from tunnel walls to reduce aerodynamic friction.

The top edge of the discharge nozzle on the MoJet is angled to direct flow down from the tunnel ceiling, while the lower edge is undercut to increase the inlet flow area, reducing entry losses. The overall result is that the jet fan has a shape that has been described as a sliced salami. Inlet losses are minimized with a bell-mouth–shaped air intake design. ANSYS Workbench parametric design studies enabled engineers to analyze different inlet bell-mouth shapes and determine the best curvature. This helped to reduce the amount

With the ever-changing costs of fuel and environmental concerns about emissions, improving energy efficiency of turbomachinery has never been more important.

© 2015 ANSYS, INC.



▲ Particle tracking comparison of a conventional jet fan (left) and MoJet (right) reveals reduced downstream Coanda effect with the MoJet.

of flow separation in the intake by 50 percent; it reduced power consumption by approximately 2.5 percent. The final design of the nozzles' sheet metal profiles was complex in comparison with the simple rectangular geometries that conventional jet fans employ, but the design was well within the capabilities of modern machining.

In addition to improving the efficiency of the MoJet, Mosen's engineers reduced its size in comparison with competing designs, which in many applications provided additional clearance to the tunnel ceiling and reduced noise emissions. Once the MoJet was designed, the engineers used ANSYS CFD to model an entire MoJet installation. The resulting model included the tunnel and fan assembly and used a wide range of mesh element sizes to capture flow dynamics.

REAL-WORLD TESTS

Systemair GmbH tested the MoJet in the Galleria Buttoli tunnel near Florence, Italy, in September 2012. Engineers conducted a number of careful tests and reported their results at BHR's International Symposium on Aerodynamics, Ventilation and Fire in Tunnels. The installation factor, which is a measure of the ratio of jet fan thrust to axial thrust imparted to tunnel air, was improved by 11 percent compared to conventional jet fans — a significant enhancement to aerodynamic thrust, and a result that confirmed Mosen's CFD predictions. The test tunnel was horseshoe-shaped. CFD calculations indicated that installation of MoJets in other types of tunnels (rectangular or within niches) would have even better performance; depending upon the shape of the tunnel and the location of the MoJets, the installation factor increase could be as high as 25 percent.

In another case, original specifications for a road tunnel in a major international airport called for 710 mm internal diameter traditional jet fans with two-pole motors. It was anticipated that this installation would be noisy, take up a lot of space, and possibly require increased safety precautions due to high jet velocities required for effective operation. A redesign of the proposed ventilation system used 800 mm internal diameter MoJets with four-pole motors that run at lower speeds. The result was a 30 percent reduction in power consumption and a 7 dB reduction in sound pressure level in the tunnel, along with significantly reduced jet air velocity. In addition, MoJets can be installed much closer to tunnel walls and at reduced distances along the tunnel, reducing construction and cabling costs.

COMPUTATIONAL FLUID DYNAMICS FROM ANSYS
ansys.com/91tunnel

MoJets have been installed in the Grimstad Port Tunnel in Norway. The fans will ventilate the tunnel to dilute CO and NO_2 emissions as well as control the direction of smoke in case of fire. Some of the pre-existing jet fans had been damaged by vehicle strikes; however, replacement MoJets will provide additional clearance for traffic and reduce future damage. Tests on the new system have indicated reduced sound pressure level as well as increased aerodynamic thrust in the tunnel.

With the ever-changing costs of fuel and environmental concerns about greenhouse gas emissions, improving energy efficiency of turbomachinery has never been more important. The ANSYS software suite enabled Mosen's engineers to test, design and enhance tunnel ventilation systems to the benefit of everyone involved.

The ANSYS software suite enabled Mosen's engineers to test, design and enhance tunnel ventilation systems to the benefit of everyone involved.

© 2015 ANSYS, INC.

SHAKING ALL OVER

Multiphysics simulation solves a vibration issue in a Francis turbine.

By Björn Hübner, Development Engineer, Voith Hydro Holding GmbH & Co. KG, Heidenheim, Germany



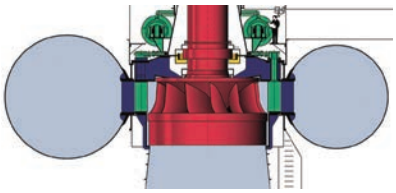
Courtesy Voith.

Strong vibration and pressure pulsation in hydraulic turbomachinery may be quite harmful to machine performance, longevity and safety. It can cause noise, cracks or even machine failure.

Voith Hydro — one of the world's leading suppliers of hydroelectric equipment, technology and services — observed strong vibrations that had the potential to cause fatigue cracking in the guide vanes of a Francis-type water turbine. In a vertical-shaft Francis turbine, water enters horizontally into a spiral-shaped pipe (spiral casing), which wraps around the circumference of a rotating runner. Stationary guide vanes regulate and direct the water to the periphery of the runner. Inside the runner channels, the

Voith Hydro observed strong vibrations that can cause fatigue cracking in the guide vanes of a Francis water turbine.

© 2015 ANSYS, INC.



▲ Runner and shaft (red), guide vanes with servo motor (green). *Courtesy Voith*.

potential energy of the water pressure is transformed into torque, which causes the runner and attached shaft and generator to rotate. Water exits the runner vertically downward into the draft tube where remaining kinetic energy is transformed into additional pressure head.

Using structural simulation, the Voith engineering team ruled out self-excitation and resonance of the guide vanes as the cause of vibration. Employing computational fluid dynamics (CFD), they determined that there was vortex shedding on the runner blades, but not on the guide vanes, that could cause the vibration. This particular machine consists of 24 guide vanes and 13 runner blades; it has an operating speed of 75 rpm. Vibration measurements revealed that all guide vanes vibrated at exactly the same frequencies within the range of 290 Hz to 305 Hz, but it was not possible to perform vibration measurements on the runner blades during operation.

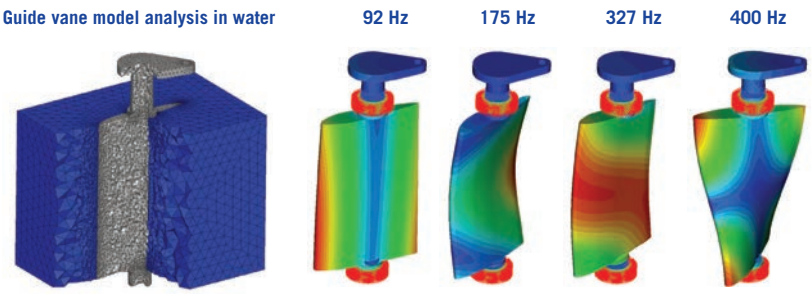
To establish how vortex shedding on the runner was affecting the guide vanes, the team used acoustic fluid–structure interaction with a finite element model of the runner in a water domain. The model used fluid finite elements to couple the dynamic behavior of the runner and water passage. The results proved that excitations at the runner blades' trailing edge were causing the vibration. The simulation matched the measured vibration frequency of approximately 300 Hz. After changing the trailing-edge shape of the prototype runner blades to minimize vortex shedding, observed vibrations were substantially reduced.

SELF-EXCITED VIBRATIONS AND RESONANCE

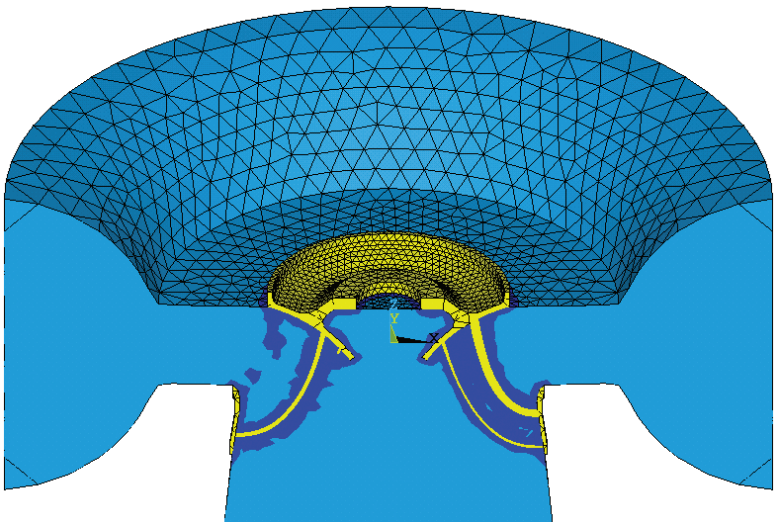
To determine the cause of vibration, Voith engineers began by examining the possibility of resonance effects or self-excited vibrations of a guide vane that would occur at a natural frequency. They used ANSYS Mechanical to create a finite element model of the guide vane in water and calculated the first four mode shapes



▲ Physical measurements of guide vane vibration. *Courtesy Voith.*



▲ Modal analysis showed that guide vane natural frequencies are far from measured frequencies of vibration. *Courtesy Voith*.



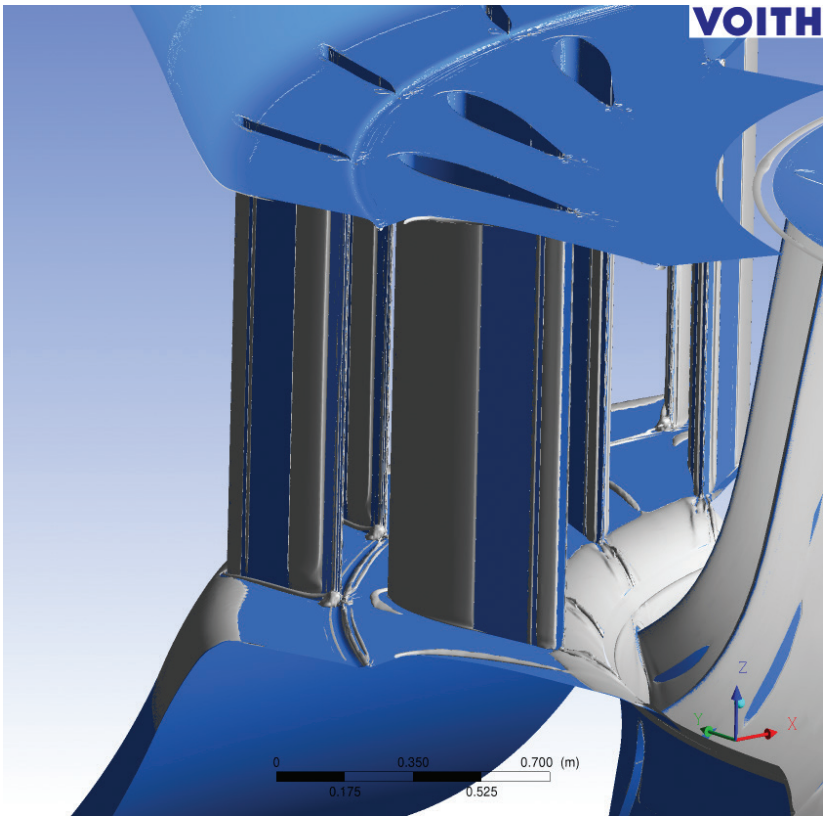
▲ Vibro-acoustic finite element model of runner in a simplified water domain. *Courtesy Voith*.

ANALYZING ACOUSTICS USING ANSYS MECHANICAL
ansys.com/91shaking

© 2015 ANSYS, INC.

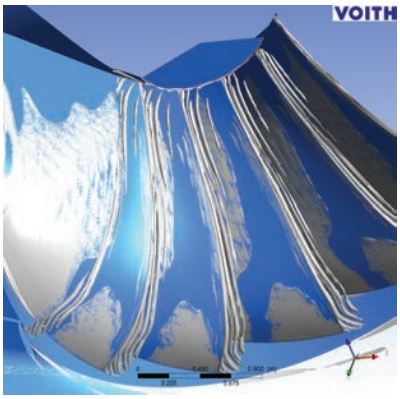
using undamped modal analysis. Engineers found that there were no natural frequencies close to the observed vibrational frequencies, indicating that guide vane resonance or self-excitation was not present. This finding was confirmed by physical measurements that showed all guide vanes vibrated within the same narrow frequency range, even though small differences in geometry and bearing conditions caused each of the guide vanes to have somewhat different natural frequencies.

To determine how vortex shedding on the runner affected the guide vanes, the team used acoustic fluid–structure interaction.



▲ Vortex shedding was not seen around guide vanes by looking at the Q-criterion that visualizes iso-surfaces of the second invariant of the strain-rate tensor, enclosing spatial regions with minimum pressure to identify vortices in a flow field. *Courtesy Voith.*





▲ CFD simulation of runner blades showed vortex shedding, visualized by two different Q-criteria. *Courtesy Voith.*

VORTEX SHEDDING

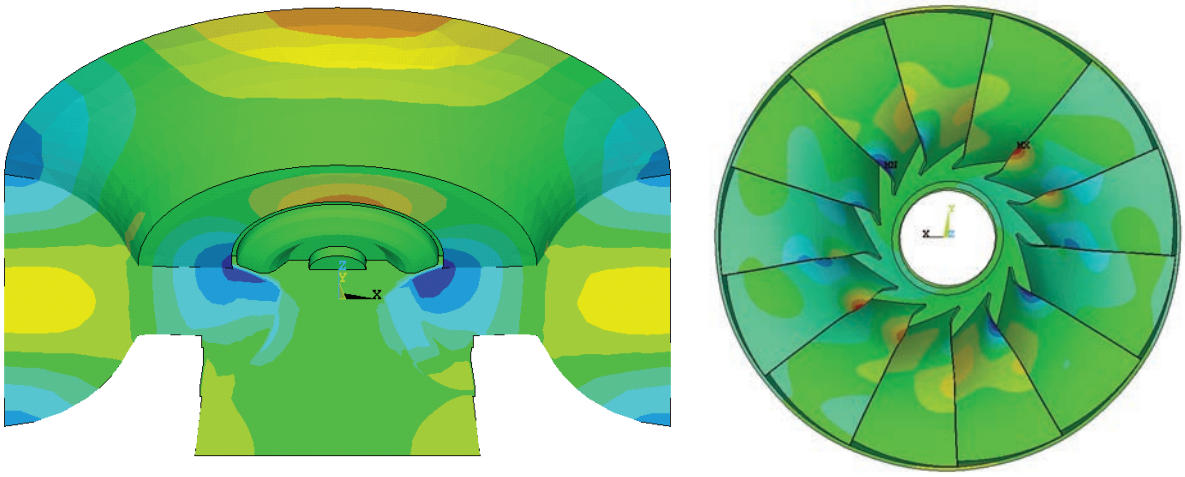
Voith performed unsteady CFD analyses with ANSYS CFX to investigate the possibility of vortex shedding at the guide vanes. The trailing edge used on the guide vanes was designed to prevent vortex shedding, and the analysis showed no sign of shedding. Therefore, the engineers concluded that the problem was not caused by vortex shedding at the guide vanes.

Next, the team performed unsteady CFD analyses at the runner blades. Because the manufactured trailing-edge shape may deviate slightly from the as-designed shape, engineers analyzed both the as-designed chamfered edge as well as a blunt trailing edge. Vortex shedding was clearly observed around 220 Hz for the blunt edge and 370 Hz for the chamfered trailing edge. For a rigid runner, vortex shedding frequencies at different blades and along the trailing edge of a single blade typically differ despite the fact that all of the guide vanes vibrate at the same frequency. The reason is that if some natural frequencies of the mounted runner in water are located in the frequency range of the vortex shedding, and if the corresponding mode shapes include trailing-edge bending, then the vortex shedding frequency may lock in and resonate at this natural frequency. The lock-in effect can cause large amplitude vibrations.

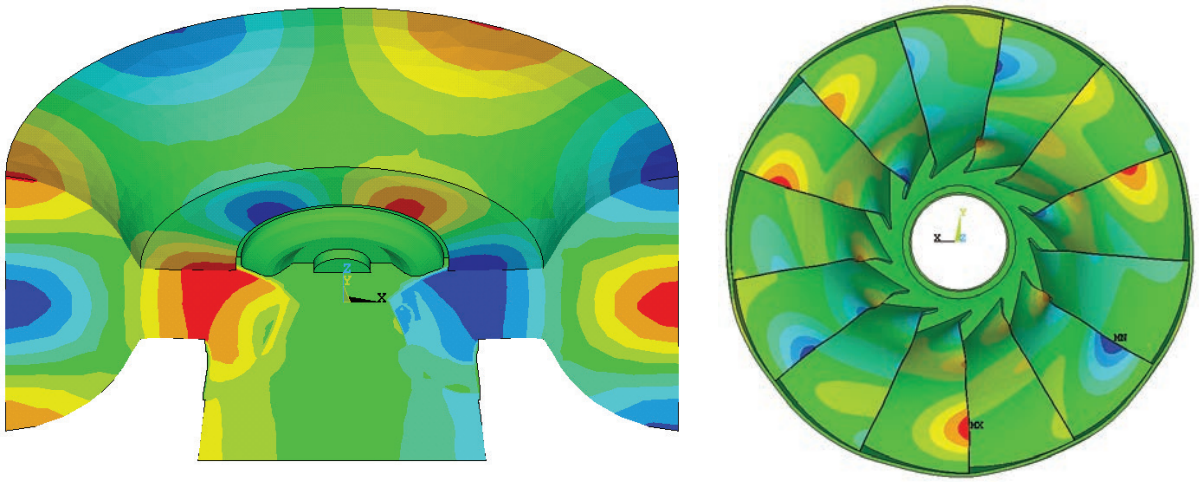
COUPLED DYNAMIC BEHAVIOR

However, vortices that separate from the runner blades move downstream into the draft tube and do not affect guide vanes directly. Thus, even with amplified vortex shedding due to lock-in effects, there must be an additional explanation for the propagation of the pressure pulse in the upstream direction to the guide vanes. Both modal and harmonic response analyses were performed with ANSYS Mechanical to investigate the coupled dynamic behavior of the entire runner and water passage using a vibro-acoustic model of the runner in a simplified water domain created using fluid elements. The finite element model included a rotating frame of reference of the runner and a simplified model of the stationary parts with full rotational symmetry. The runner structure was fixed in the axial and circumferential direction at the connection to the shaft,

© 2015 ANSYS, INC.



▲ Pressure field (left) and axial runner displacement (right) of vibro-acoustic mode shape with two diametrical node lines at a natural frequency of 301 Hz. *Courtesy Voith*.



▲ Pressure field (left) and axial runner displacement (right) of vibro-acoustic mode shape with three diametrical node lines at a natural frequency of 325 Hz. *Courtesy Voith*.

ANALYZING VIBRATION WITH ACOUSTIC–STRUCTURAL COUPLING
ansys.com/91shaking2

and a fluid–structure interface was coupled to the runner structure and acoustic fluid domain. This simplified modal analysis of the undamped vibro-acoustic model provided mode shapes and corresponding natural frequencies. Multiple natural frequencies were detected close to the measured frequency range of the guide vane vibrations. Most of the associated vibro-acoustic mode shapes exhibited large bending displacements at runner-blade trailing edges as well as strong pressure fluctuations in the guide vane area.

Harmonic response analysis was performed to get a clearer picture of the vibro-acoustic effects in the area of the runner and distributor. The runner was excited by rotating force patterns with distinct numbers of diametrical node lines. Each natural frequency has a particular mode shape defined by the number of diametrical node lines. At each runner blade, a single force acts on the trailing edge perpendicular to the blade surface. The results revealed vibro-acoustic resonances with large bending displacements and high pressure pulsations. Both pressure and displacement criteria exhibited clear resonance peaks at 295 MHz for a mode shape with three diametrical node lines and 306 Hz for a mode shape with seven diametrical node lines, which is close to the measured vibration.

The results of the harmonic response analysis together with modal analysis indicate that lock-in effects based on coupled vibro-acoustic resonance conditions synchronize and amplify vortex shedding. The corresponding vibro-acoustic mode shapes propagate and amplify pressure pulsations within the rotating and stationary components of the turbine. The pressure pulsations cause synchronized guide vane vibrations at the natural frequencies of vibro-acoustic mode shapes. The problem was solved by a modified trailing-edge shape that minimized and de-tuned vortex shedding at the runner blades, substantially reducing the guide vane vibrations.

Determining and solving this vibration issue may not have been possible using a single physics. It required understanding all physics involved and applying them appropriately to the problem at hand.

Reference

Hübner, B.; Seidel, U.; D'Agostini Neto, A. Synchronization and Propagation of Vortex-Induced Vibrations in Francis Turbines due to Lock-In Effects Based on Coupled Vibro-Acoustic Mode Shapes. Proceedings of the 4th International Meeting on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Belgrade, **2011.**

© 2015 ANSYS, INC.

KNOWING THE SCORE

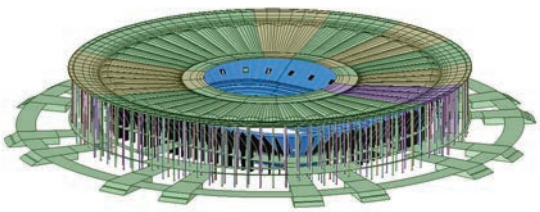
Analyzing a World Cup stadium with ANSYS multiphysics tools takes one-tenth the time of wind-tunnel testing.

By Paulo de Mattos Pimenta, Professor, Polytechnic School at University of São Paulo, Brazil



Photo courtesy Tomas Faquini.

Estádio Nacional Mané Garrincha, a 70,000-seat football stadium in Brasília, Brazil, was rebuilt in 2013 and hosted seven games of the 2014 FIFA World Cup Brazil, including a quarter-final. Validating the design of a stadium of this size for wind loads normally requires wind-tunnel testing, which is time-consuming and costly, and runs the risk of scaling errors — a scale model that fits in a wind tunnel does not exhibit the exact same behavior as an extremely large building. With assistance from CFD simulation specialists at ESSS, the ANSYS channel



▲ Geometrical model for CFD analysis

© 2015 ANSYS, INC.

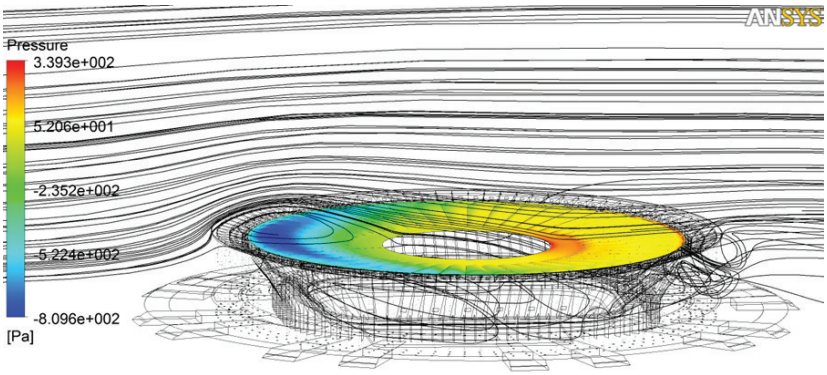
In this case, validation had to be done in only 15 days, far less time than is required to build a scale model and test it in a wind tunnel.

partner in Latin America, the engineering consultant for the project used ANSYS multiphysics capabilities to verify the safety of the stadium. The ESSS team used CFD tools to predict airflow around the stadium and pressure on the stadium cover. Then the engineer (the author) performed a structural investigation to study the combined effects of wind, stadium infrastructure and the cheering crowd. Analysts recommended several changes, such as increasing the number of cables and cable tension. This is believed to be the first time that CFD analysis has been used to replace wind-tunnel testing in the design of a major stadium in Brazil. The analysis was completed in only two weeks. Simulation reduced costs by one-third and took one-tenth the time of wind tunnel testing.

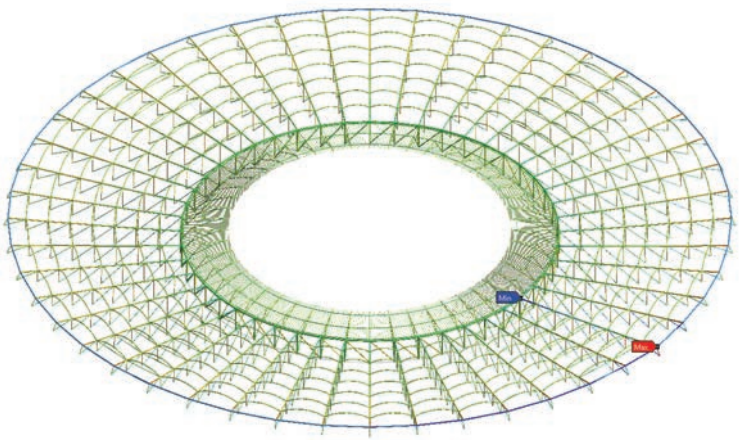
MAJOR STADIUM RENOVATION PROJECT

The stadium originally was built in 1974 and named after the famous Brazilian soccer player Mané Garrincha. Estádio Nacional was nearly demolished through an implosion in 2011 to make way for the current stadium, which includes a new facade, metal roof and stands — as well as a lowered pitch (playing field) that enables unobstructed views from every seat. The reconstruction involved dismantling the lower tier of seats and incorporating the upper tier into a new rectangular bowl. The size of the playing field was reduced to make the stadium into a single-use facility for football. The renovation cost approximately $500 million (U.S.).

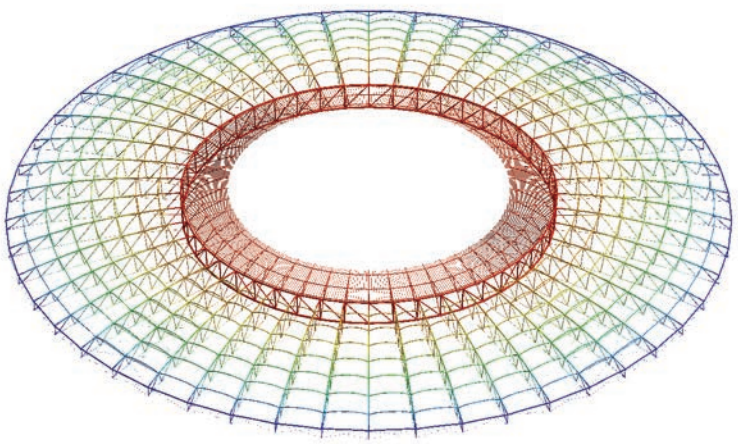
NOVACAP, a Brazilian state company involved in construction in Brasília, contacted the engineer to validate the safety of the stadium design from a wind-loading perspective. Traditionally, this is done by building a scale model and testing it in a wind tunnel while measuring loads on the model. More recently, projects have been completed by using CFD to predict the loads on the structure, then using a wind tunnel to validate CFD simulation. However, stadium CFD simulation has progressed to the point that wind-tunnel validation is no longer mandatory, saving substantial time and money. In this case, the validation had to be done in only 15 days, far less time than is required to build a scale model and perform wind-tunnel testing.



▲ Streamlines and air pressure on roof as predicted by CFD



▲ Structural model



▲ Total deformation at 0.432 Hz

© 2015 ANSYS, INC.

BEST PRACTICE FOR FLUID–STRUCTURE INTERACTION (FSI) AND SYSTEM COUPLING SIMULATIONS
ansys.com/91score

Simulation reduced costs by one-third and took one-tenth the time of wind-tunnel testing.

CFD SIMULATION

NOVACAP provided an architectural model of the design. The stadium was designed as two independent structures. The roof is supported by columns and is independent of the stadium itself, which consists of seating, stairs and ramps. The roof is 309 meters in diameter, the largest circular roof in the world. The CFD design space was 6 km in both horizontal and vertical directions, which is about 20 times the size of the stadium. The model used quadrilateral, tetrahedral and pyramidal elements. The completed model has 20 million computational cells and 120 million degrees of freedom. The team iterated to remove details of the geometry that did not impact the flow, hence speeding up simulation without any loss in accuracy. Wind speeds were taken from Brazilian building code, which specifies a velocity of 35 meters per second. The team applied wind from two orthogonal directions as a boundary condition at the edge of the solution domain and employed the k-epsilon turbulence model.

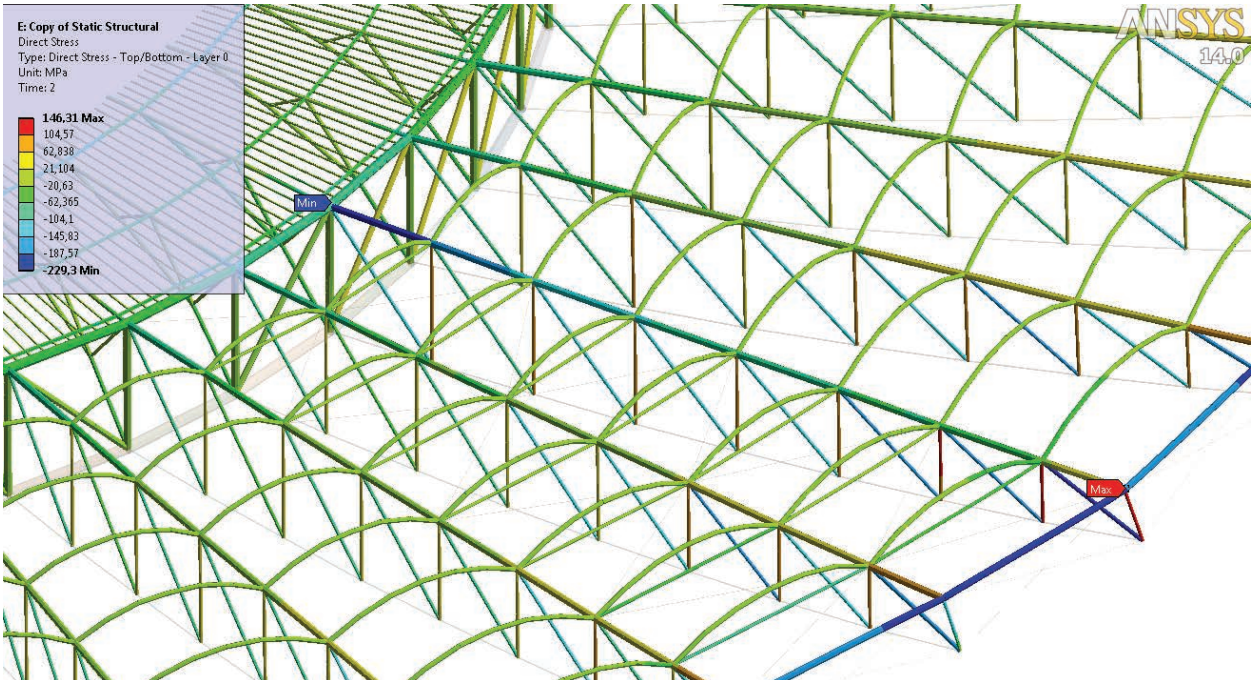
ANSYS CFD simulation took about four hours to complete on a high-performance computing cluster with 12 nodes, 24 processors and 96 gigabytes of RAM. The results of the analysis provided the pressures, both positive and negative, exerted by the wind on the structure's various elements.

STRUCTURAL ANALYSIS

The engineer converted the design into a finite element model with 100,000 beam and shell elements for structural analysis. The pressures predicted by the CFD model were transferred to ANSYS Mechanical using the ANSYS Workbench environment. The gravitational loads provided by spectators in the stands, lights and audiovisual systems were also incorporated into the model. The engineer first performed a static linear analysis with all loads applied. A dynamic analysis calculated the structure's natural mode shapes and frequencies of vibration. Modal analysis was performed on the pre-stressed structure. The lowest frequency mode was a rotating mode at below 0.5 Hz. This mode was a problem because the original design did not have a lot of stiffness in the rotating direction. The lowest bending mode was at 0.8 Hz, which was acceptable. The engineer used hand calculations to determine the amplification factor of the structure.

The CFD model allowed evaluation of a number of different design changes to address the rotating mode problem. Simulation showed that by adding additional cables and increasing tensile force on some of the existing cables, the rotational stiffness of the structure increased and the frequency of the rotating mode was raised to above 0.8 Hz. The architectural design incorporated these changes, and the structure was completed in 2013. The stadium was used for the first time for the opening match of the Confederations Cup, in which Brazil defeated Japan. The stadium hosted seven matches of the 2014 FIFA World Cup Brazil; it will host some football games in the 2016 Summer Olympics to be held in Rio de Janeiro.

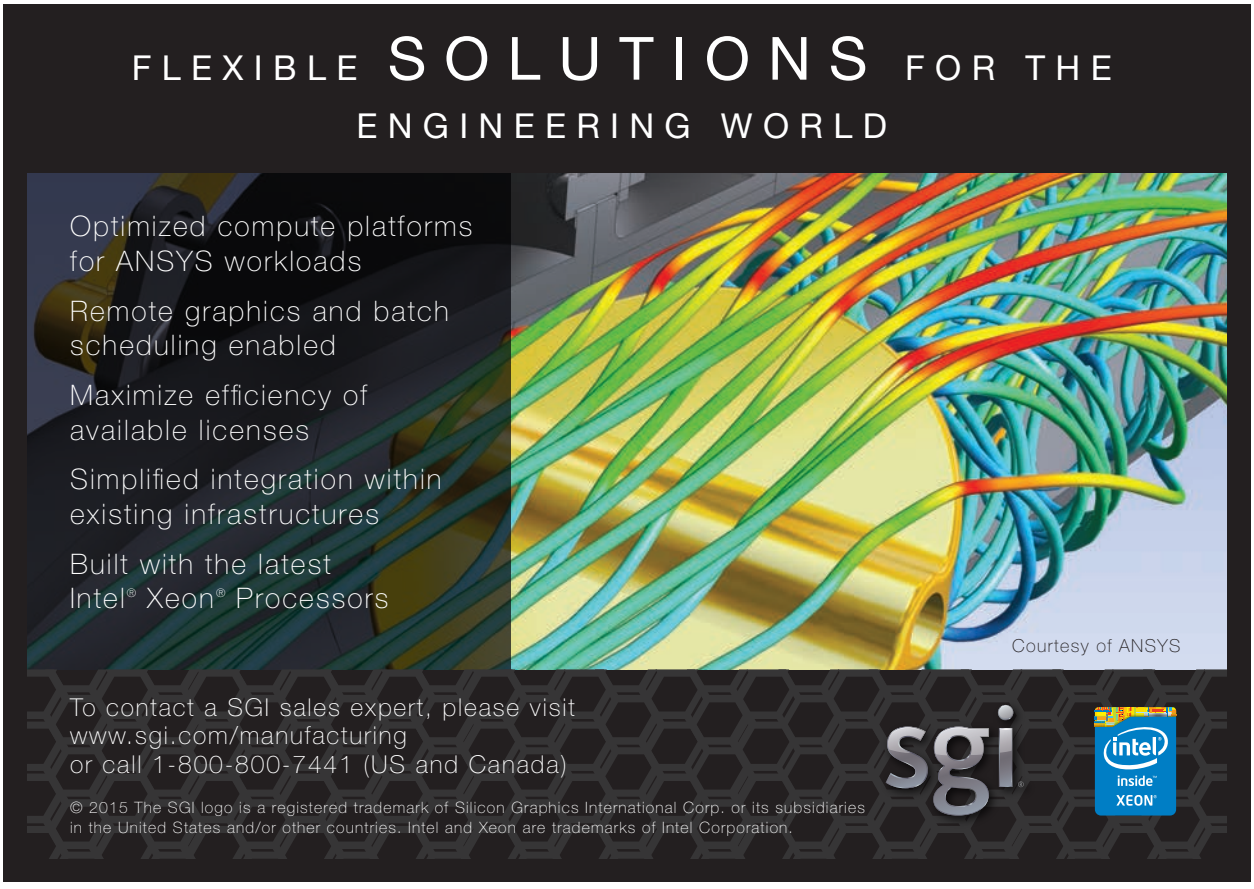
The work was performed under the auspices of Maruska Holanda (NOVACAP) and Pedro Almeida.



Stresses in cables and trusses as predicted by structural analysis

© 2015 ANSYS, INC.

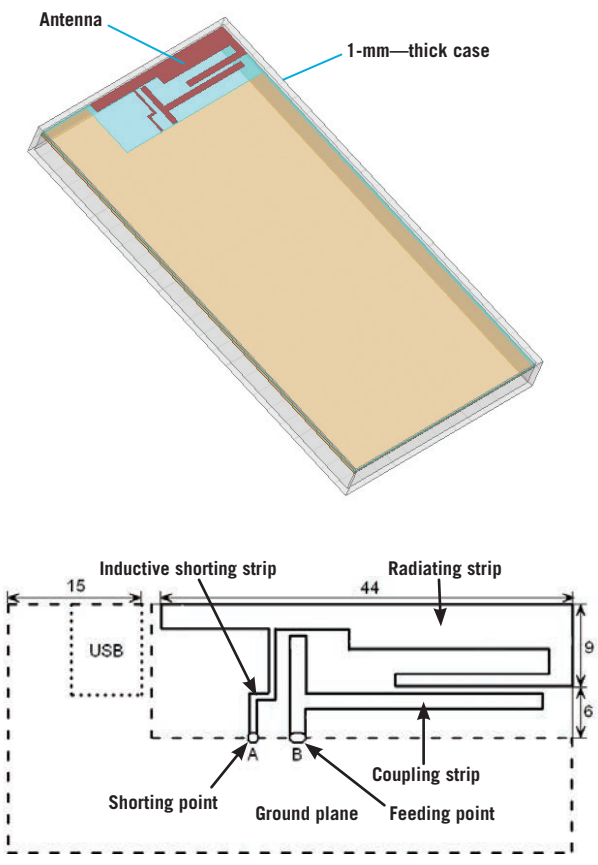

Robust Simulations Demand Robust Performance
Simulation-based design is changing the way companies prototype—Intel and ANSYS are driving those innovations.
intel
intel inside XEON PHI
Now supporting Intel® Xeon Phi™ coprocessor on Windows* with Mechanical 16.0.
http://bit.ly/Intel-AnsysPartnerPage
ANSYS
©2015 Intel Corporation. All rights reserved. Intel, the Intel logo, Intel Inside, and Xeon Phi are trademarks of Intel Corporation in the U.S. and/or other countries. *Other names and brands may be claimed as the property of others.


FLEXIBLE SOLUTIONS FOR THE ENGINEERING WORLD
Optimized compute platforms for ANSYS workloads
Remote graphics and batch scheduling enabled
Maximize efficiency of available licenses
Simplified integration within existing infrastructures
Built with the latest Intel® Xeon® Processors
Courtesy of ANSYS
To contact a SGI sales expert, please visit www.sgi.com/manufacturing or call 1-800-800-7441 (US and Canada)
© 2015 The SGI logo is a registered trademark of Silicon Graphics International Corp. or its subsidiaries in the United States and/or other countries. Intel and Xeon are trademarks of Intel Corporation.
sgi
intel inside XEON

DEVELOPING RELIABLE ANTENNAS FOR SMARTPHONES

A team of researchers used ANSYS technology to design a small multiband internal antenna to successfully operate across eight different frequency bands.

By Jiangsheng Zhou, Principal Engineer, Wireless Terminal Design Team
Jiang Nan Electronic and Communication Research Institute, Jiaxing, China



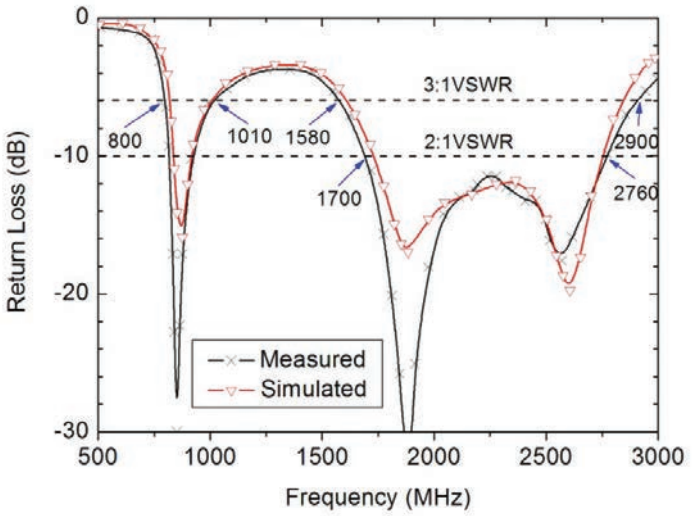
▲ (Top): Geometry showing location of proposed coupled-fed printed antenna design on the device PCB. (Bottom): Dimensions in mm of the antenna's metal pattern. The radiating strip is along the top edge. The thin inductive shorting strip extends off the left side of the radiating strip while the T-shaped coupling capacitive strip is below the radiating strip on the right.

At its introduction in 1983, the first commercially available cell phone was seen as more of an expensive novelty than a necessary tool. Costing the equivalent of more than $9,000 U.S. today, it weighed almost 2 pounds and was notoriously bulky with its rubber whip antenna extending more than 5 inches above its 10-inch body. Despite such hefty dimensions, its battery provided only 30 minutes of talk time before requiring a recharge.

Over the past three decades, mobile phone technology has evolved through many generations, with a rapidly expanding set of features contained within a smaller, lighter and far less expensive package. Modern mobile devices are more like highly portable personal computers than phones. However, a large factor in a device's ability to provide the freedom to communicate from almost anywhere is still its antenna. Designing small-profile, multiband and wideband internal antennas with a simple structure has become a necessary challenge for the mobile phone industry. Phone manufacturers need to keep production costs low while continuing to produce devices with more options. All of these varied capabilities affect one another, increasing the inherent challenges in antenna design.

Designing small-profile, multiband and wideband internal antennas with a simple structure has become a necessary challenge for the mobile phone industry.

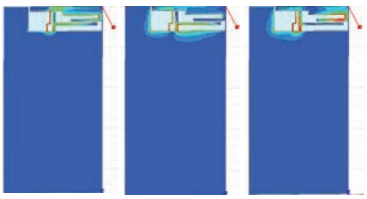
© 2015 ANSYS, INC.



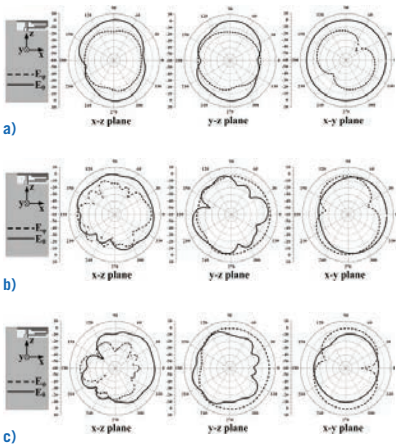
▲ Comparison of measured values of return loss (in dB) versus those predicted by ANSYS HFSS at the range of frequencies for multiband and wideband signals. Dotted lines show the equivalent values of voltage standing wave ratios (VSWR), which indicate the degree of impedance matching at the indicated frequencies.



▲ Front (left) and back (right) views of the manufactured internal planar printed antenna



▲ Contours of surface current density on metal portion of proposed antenna design at 900 MHz (left), 1,900 MHz (center) and 2,600 MHz (right). The colors indicate current density, with red indicating the most dense.



▲ Measured radiation patterns at (a) 900 MHz, (b) 1,900 MHz and (c) 2,600 MHz for the proposed antenna

A team of engineers used ANSYS HFSS to design a small internal multiband antenna that can operate across eight different frequency bands.

At the Jiang Nan Electronic and Communication Research Institute in China, a team of engineers used ANSYS HFSS to design a small internal multiband antenna that can operate across eight different frequency bands. For a wireless device to function properly as both phone and computer, it needs to send and receive wideband (824 MHz to 2,690 MHz) and multiband (GSM 850 MHz, 900 MHz, 1,800 MHz and 1,900 MHz; UMTS 1,920 MHz to 2,170 MHz; WLAN 2,400 MHz; and WiMAX 2,300 MHz, 2,500 MHz) signals. WLAN, WWAN and WiMAX are the most prevalent types of Wi-Fi networks, while WLAN 2,400 MHz is the frequency required for Bluetooth® connectivity. GSM and UMTS bands are used for global cell phone communications, including 2G, 3G, 3.5G and 4G LTE.

The team investigated various antenna designs using simulation. They varied the length and width of the radiating, coupling and inductive shorting strips, as well as the shorting and feeding pin positions. Changing these dimensions for the HFSS simulation led to significant variation in the scattering parameters (S-parameters), specifically the return loss (S11). Return loss can be used to judge antenna performance at different frequencies. The team optimized the dimensions for return loss values at frequencies between 824 MHz and 2,500 MHz.

Additional methods for determining antenna performance include current distributions, far-field patterns, gain and antenna efficiency. The team simulated the current distributions and far-field patterns at 900 MHz, 1,900 MHz and 2,600 MHz. Strong surface current distributions on the radiating and shorting strips indicate that these strips are the main contributors for the lower-frequency band operation. The long coupling strip controls the resonant mode at 1,900 MHz, while the short coupling strip controls the resonant mode at 2,600 MHz. Smooth variations in vertical polarization in all directions indicate good antenna coverage. The radiation efficiency varied between 50 and 64 percent at lower frequencies to a high of 62 to 77 percent at higher frequencies. Antenna efficiencies greater than 45 percent are sufficient for practical mobile phone applications.

The final optimized design required a modest 15-mm by 45-mm area on a small PCB that did not include a ground space at either the top or the bottom. The

© 2015 ANSYS, INC.

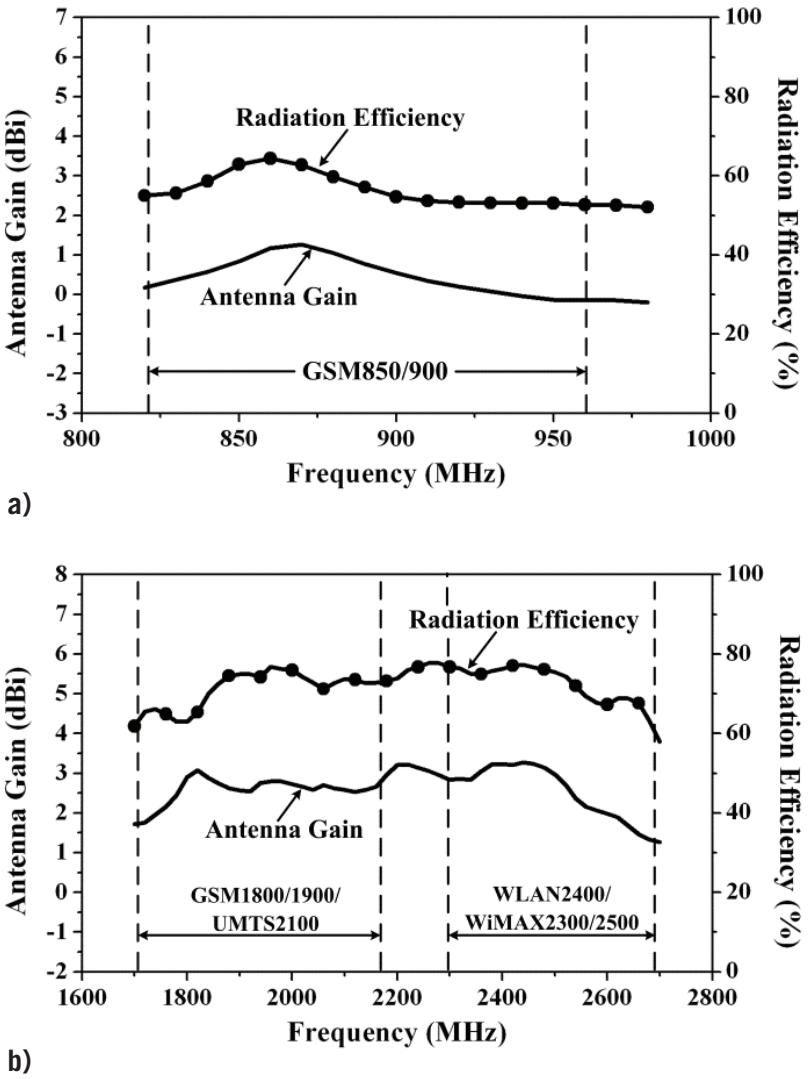
Based on both simulated and experimental results, the design is suitable to be directly printed on the system PCB of the device.

THE ROLE OF SIMULATION IN INNOVATIVE ELECTRONIC DESIGN
ansys.com/91antennas

antenna design consists of a radiating strip, an inductive shorting strip and a coupling strip. The radiating strip is the longest, located on the top edge of the PCB's no-ground space, while the inductive strip is only 0.5 mm wide and is short-circuited to the system ground via a hole in the PCB. The coupling strip is located between radiating and inductive strips. The coupling strip provides an excitation voltage to both the radiating strip and the inductive shorting strip. Together, the radiating and inductive shorting strips produce low-band GSM resonance. The coupling strip has two branches that can generate a wide upper-band range of 1,710 MHz to 2,690 MHz.

To confirm the HFSS results, the team tested the antenna using a vector network analyzer (VNA) and an anechoic chamber. With the antenna printed on the PCB, the team excited the antenna using a 50-ohm coaxial feed at the antenna feed point. To simulate realistic conditions, there was also a 1-mm–thick plastic housing with a 1-mm gap between the housing and the PCB, representing the phone's case. The simulated return loss compared well with the actual return loss determined through physical experiments.

Before engineers started using simulation tools, they had to build prototypes and then test each design variation individually using the same setup required for confirming the simulation results. This was a highly inefficient method that required weeks for each evaluated design, allowing little time for optimization and testing of innovative changes. By incorporating ANSYS HFSS into the design process, the team explored several antenna configurations and optimized the design before building any prototypes, which saved 20 percent in material costs. Engineers were able to run 20 different simulations to analyze various antenna configurations and optimize the final design in just three days.



▲ Measured radiation efficiency and antenna gain for proposed antenna: (a) GSM 850/900 MHz band; (b) GSM 1,800/1,900 MHz/UMTS 2,100 MHz and WLAN 2,400 MHz/WiMAX 2,300/2,500 MHz band

The antenna performance results predicted by ANSYS HFSS for impedance matching showed close agreement with the experimental data obtained through the engineering team's experiments using a vector network analyzer. Based on both simulated and experimental results, the design is suitable to be directly printed on the system PCB of the device. Combined with its low fabrication costs, this makes the antenna a very attractive option for mobile phone applications.

© 2015 ANSYS, INC.

MAGNETIC NANOPARTICLE SIMULATION

Research using simulation will help to develop biomagnetic beads for disease treatment.

By John R. Brauer, Adjunct Professor, Electrical Engineering and Computer Science
Milwaukee School of Engineering, U.S.A.

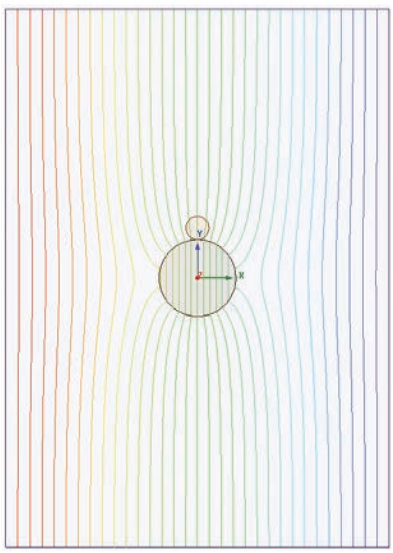


Courtesy Dexter Magnetic Technologies.

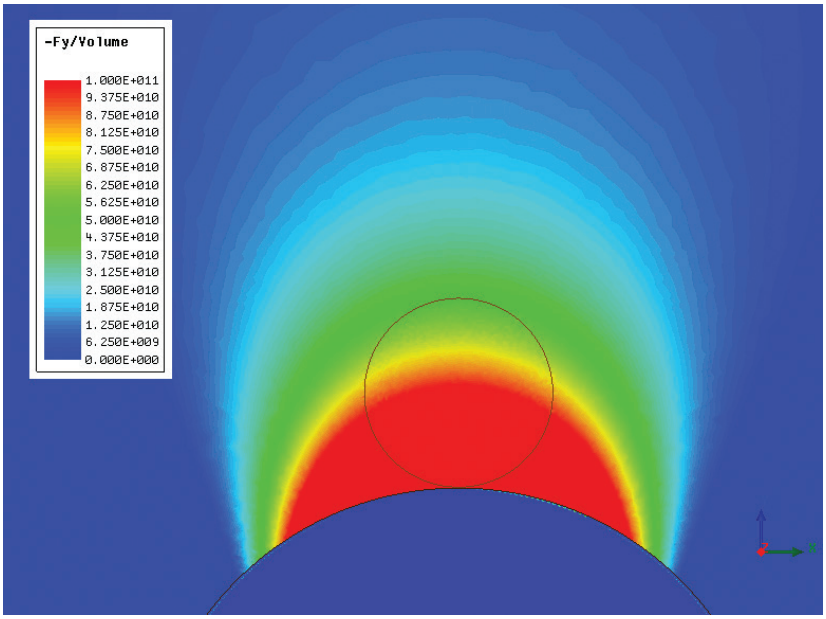
Magnetic separators are used to extract magnetic parts and particles from nonmagnetic materials. For example, recycling centers use magnetic separators to remove steel objects from other metals and nonmetals. The magnetic permeability of steel and iron is much higher than that of other materials and air, and this high permeability interacts with the magnetic field of the separator to produce a magnetic force attracting the iron or steel parts or particles. Magnetic nanoparticles are particles containing iron or iron compounds that range in size from a micrometer to nanometers.

Organizations perform extensive research applying biomagnetic beads and magnetic separators to a variety of healthcare challenges.

© 2015 ANSYS, INC.



▲ Magnetic separator analyzed by Oberteuffer containing a steel wire. The applied vertical magnetic field has flux lines that are distorted near the wire and particle.



▲ Display of force density distribution on particle above the wire in separator

HEALTHCARE APPLICATIONS

Biomagnetic beads are special magnetic nanoparticles that have been developed recently for use in biomedical screening and other healthcare applications. They are spheres made of iron compounds that have a strong magnetic permeability surrounded by a thin coating made of a polymer and biological molecules. Each variety of coating will bind to a specific biomolecule so each type of bead can be used to magnetically separate a specific type of biomolecule. Biomagnetic beads can be used to screen DNA, RNA, various proteins or genes, and possibly even detect cancer cells. Several large healthcare companies and other high-technology organizations are performing extensive research applying biomagnetic beads and magnetic separators to a variety of healthcare challenges.

ELECTROMECHANICAL SYSTEM SIMULATION — BUILDING THE FUTURE
ansys.com/91nano

FORCES ON A HIGH-PERMEABILITY PARTICLE IN A SEPARATOR

A classic magnetic separator consists of cylindrical steel wires placed in a uniform direct-current magnetic field. A classic formula by Oberteuffer has been historically used to compute the forces for both cylindrical and spherical particles. [1] To investigate the accuracy of the Oberteuffer formula, the researcher used ANSYS Maxwell electromagnetic field simulation software to compute the magnetic flux lines and magnetic force for magnetic particles with diameters of 0.1, 1.0, 2.0 and 3.0 µm (micrometers).

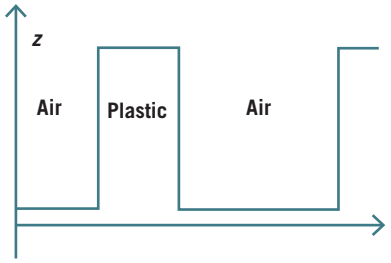
The permeability of both wire and particle is assumed to be 1,000 times that of air, which is typical for steel or iron. The ANSYS Maxwell analysis is a 2-D planar analysis, in which the particle — like the wire — is assumed to be a cylinder normal to the plane. As expected, the flux pattern is not symmetric above and below the wire because the particle alters the field. Unfortunately, Oberteuffer's formula does not account for such alteration by the particle; it includes only the alteration of the field by the wire itself. Additionally, the formula assumes that the magnetic flux density at the center of the particle acts on the entire particle.

To compare Oberteuffer's formula for force per unit volume with ANSYS Maxwell computations, the researcher examined the wire separator with an applied vertical magnetic flux density of 1 tesla. Various particles were considered, but all particles were assumed to be located at a radius 13.3 µm from the center of the wire. The approximate formula yields a force per unit volume in the vertical direction of -105.75×10^9 N/m^3. The negative value indicates a downward force on the particle above the wire.

The researcher carried out finite element force computations using ANSYS Maxwell. He computed the magnetic flux density without modeling the high-permeability particle and developed a fields calculator expression for ANSYS Maxwell to compute and display the force density in color.

ANSYS Maxwell's field calculator can be used to carry out volume integrals to determine the total magnetic force acting on any particular magnetic particle.

© 2015 ANSYS, INC.



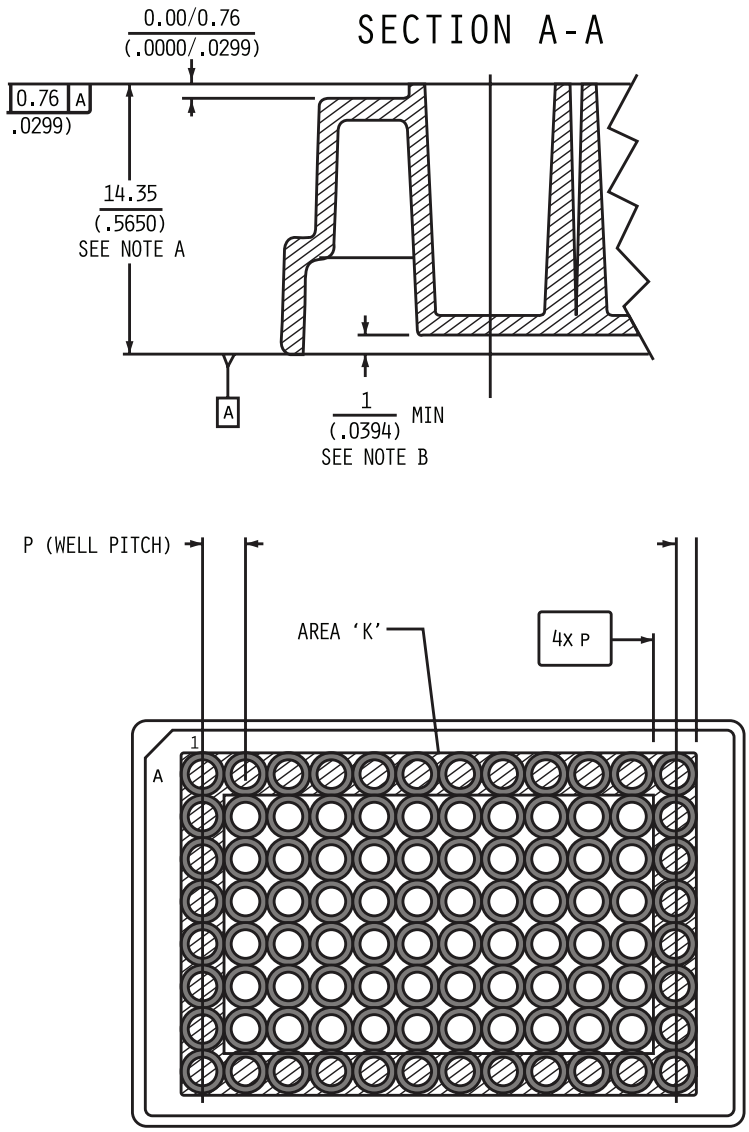
▲ Simulation model with nearby well

ANSYS Maxwell's fields calculator can be used to carry out volume integrals to determine the total magnetic force acting on any particular magnetic particle. The total magnetic forces for various particles with radii in the micrometer and nanometer range are easily calculated for the wire separator. By dividing the forces by the volumes of the cylindrical and spherical particles, the researcher obtained computed average force densities and plotted them versus particle radii. The computations using ANSYS Maxwell agreed reasonably well with Oberteuffer's formula, but the theoretical densities were all slightly smaller, within −8.46 percent to −0.12 percent, which can be explained in part by the more accurate field representation from the finite element solution. [2]

FORCE DENSITY OF A PERMANENT MAGNET SEPARATOR ON A MICROPLATE

In addition to iron separation using magnetic field gradients, a rapidly growing area of magnetic separation involves medical devices and biomolecular screening such as DNA testing. For example, high-field gradient separation is used to separate red blood cells from whole blood and to separate cancer cells from bone marrow. Other biomedical magnetic separators are being investigated to target drug delivery and to remove toxins. Permanent magnets (as opposed to electromagnets) are often used as a field source for magnetic separators; these are being investigated for biomolecular screening using standard biomolecular testing apparatus.

A microplate, which typically contains 96 wells, is regularly used in drug discovery, DNA/RNA testing, detection of proteins/antigens/genes and other



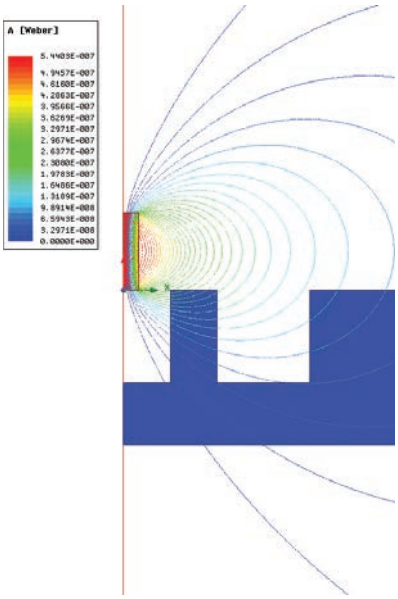
▲ Diagram of microplate for separation of biomagnetic beads (96 wells with well pitch p = 9 mm) (top), cross section with dimensions in mm (bottom)

types of biomolecular screening. A solution with suspended biomagnetic beads is placed in each well of the microplate. Biomolecular screening is possible by coating the beads with different proteins or other biological molecules so that the nanoparticles can serve as magnetic sensors. Bead diameters are often in the range of 1 µm to 4 µm, but may be in the nanometer range.

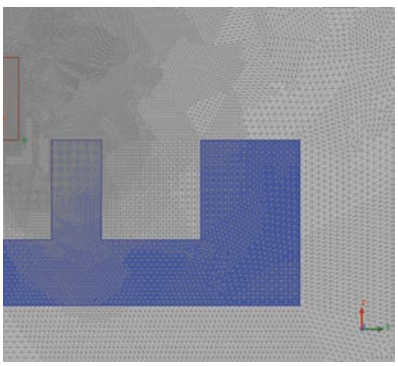
The researcher used ANSYS Maxwell to analyze magnetic force density acting on such nanoparticles in one of the wells with a permanent magnet separator immediately above it. Since permanent magnets can be made of different materials, the analysis first included a permanent magnet made of ceramic 5 ferrite and next a permanent magnet made of neodymium iron boron (NdFeB). [2]

The specific gravity of magnetic beads placed in the solution in microplate wells is less than that of iron because such particles are made of iron oxide mixed with polymers. This reduces the permeability of the nanoparticles compared to iron. Biomagnetic nanoparticle specific gravity and permeability are either unknown or proprietary and, thus, cannot be reported here. For comparison purposes, both

© 2015 ANSYS, INC.



▲ Cylindrical permanent magnet added to axisymmetric ANSYS Maxwell microplate model



▲ Adaptively refined mesh

forces were computed using the properties of iron.

For a permanent magnet separator to function properly, the magnetic force density in the primary well must exceed downward force density. The downward force density is the vector sum of gravitational, buoyancy, viscous and surface tension force densities. Here the gravitational force is assumed to dominate. Since gravitational force density is of magnitude 7,644 N/m^3, magnetic force density is displayed in the range from 0 to 10,000 N/m^3.

CERAMIC 5 FERRITE VERSUS NEODYMIUM IRON BORON

First, the investigator assumed that the permanent magnet is made of ceramic 5 ferrite with a coercive field intensity H_c = 1.91 × 10^5 A/m and a permeability 1.08 times that of air. The simulation of the vertical magnetic force density shows that the 10,000 N/m^3 zone is much smaller than the primary well. Thus, over most of the well, the magnetic force density is insufficient to overcome the gravitational force density.

Next, the researcher assumed that the permanent magnet is made of neodymium iron boron with H_c = 8.9 × 10^5 A/m and a permeability 1.1 times that of air. The vertical magnetic force density simulation now has a 10,000 N/m^3 zone that is large enough to encompass most of the primary well top without affecting the adjacent well. Thus the magnetic force density overcomes gravitational force density in most of the primary well while not interfering with adjacent wells, resulting in proper magnetic separation of the microplate [2].

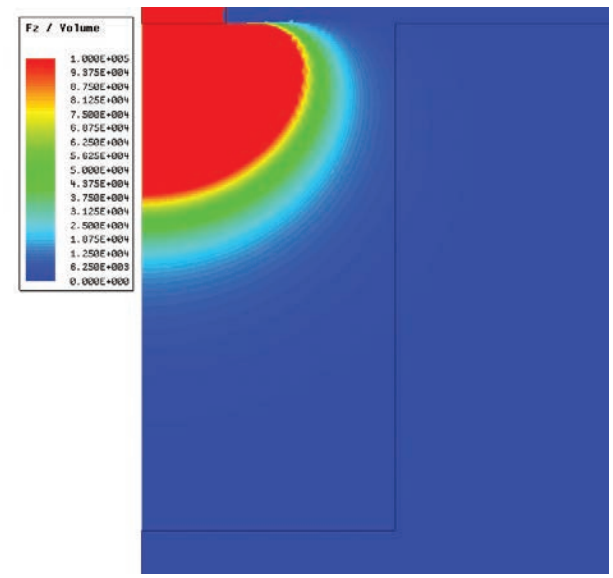
In actual microplate magnetic separation, it is accepted that neodymium iron boron magnets are needed so the force density displays agree qualitatively with observed behavior. ANSYS Maxwell computations can readily analyze many other designs of magnetic bead separators made of permanent magnets, steel and other materials.

The applications of biomagnetic beads are widely varied and rapidly growing. In addition to separation of various cell types, research into DNA extraction and genetic exploration is underway. Exosome analysis, bioassays for determining the efficacy of drugs, and regeneration of tissues such as heart muscle with the help of cell separation are among many exciting areas of research using magnetic nanoparticles.

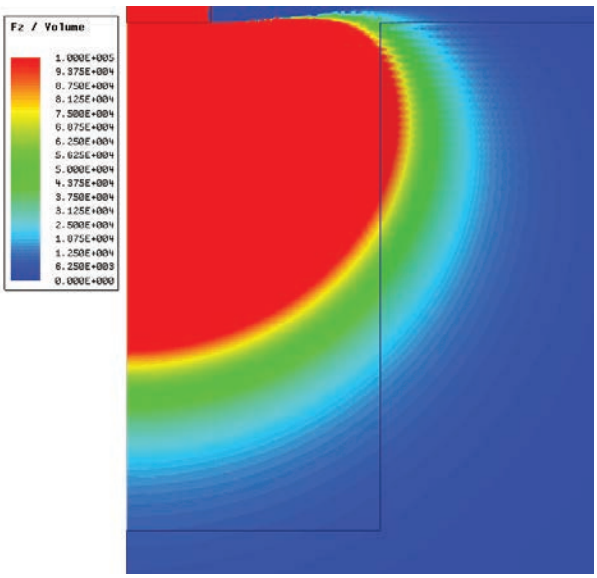
References

[1] Oberteuffer, J.A. Magnetic Separation: A Review of Principles, Devices and Applications. *IEEE Trans. Magn.* **1974**. Vol. MAG-10, No. 2, pp. 223–238.

[2] Brauer, J. R. *Magnetic Actuators and Sensors*, 2nd Ed., Wiley IEEE Press, **2014**.



▲ ANSYS Maxwell-computed force density for ferrite magnet (H_c=1.91 × 10^5 A/m)



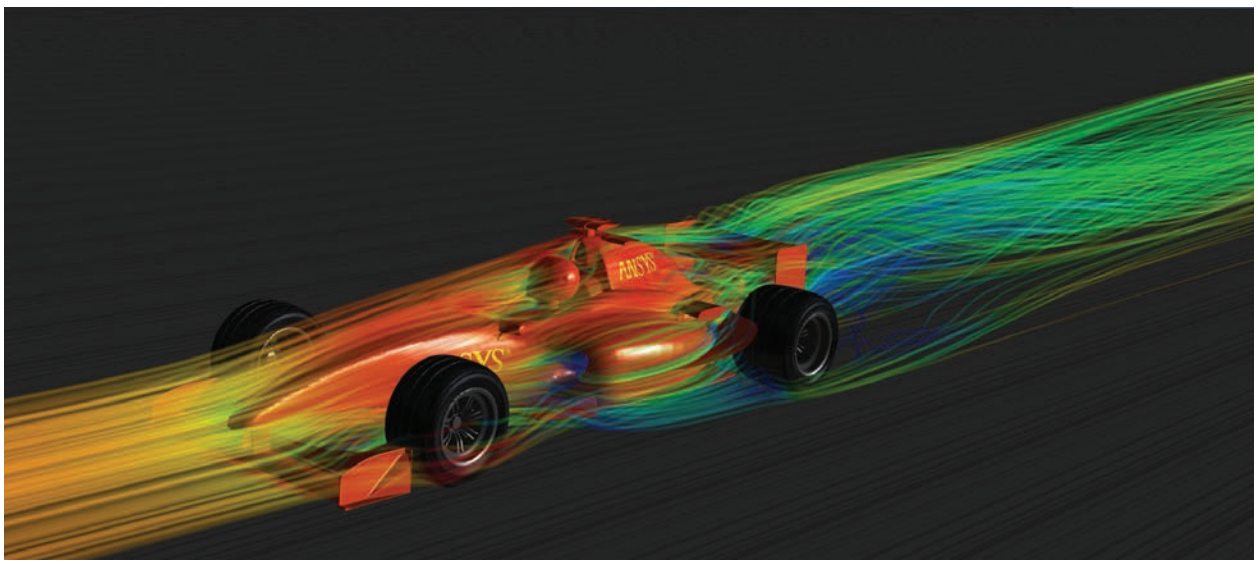
▲ Computed force density for NdFeB magnet (H_c=8.9 × 10^5 A/m)

© 2015 ANSYS, INC.

ACCELERATING ANSYS FLUENT SIMULATIONS *WITH* NVIDIA GPUs

ANSYS Fluent supports GPUs so engineers can meet project schedules and get robust products to market faster.

By Vijay Sellappan, Applied Engineer, and **Bhushan Desam**, Senior Alliances and Marketing Manager, NVIDIA Corporation, Santa Clara, U.S.A.



▲ External aerodynamic flow over the body of a race car is a solution that can be accelerated by using GPUs.

ANSYS Fluent software supports solver computation on NVIDIA® graphics processing units (GPUs) to help engineers reduce the time required to explore many design variables to optimize product performance and meet design deadlines. Integration of AmgX, a library of GPU-accelerated solvers developed by NVIDIA, within Fluent makes this possible. By adding GPUs to existing clusters and workstations, engineers can reduce time to solution by up to half. In addition to speeding up simulation, GPUs

By adding GPUs to existing clusters and workstations, engineers can reduce time to solution by up to half.

© 2015 ANSYS, INC.

GPUs and ANSYS HPC

GPUs are supported in the most recent release of all ANSYS HPC license products including ANSYS HPC, ANSYS HPC Pack and ANSYS HPC Workgroup. Further, each GPU is treated as a CPU core in terms of licensing, so users can gain higher productivity through GPU simulations.

consume less energy when compared with a CPU-only solution.

Activating the GPU feature is straighforward, but will all Fluent simulations benefit from employing GPUs? Read on.

MODEL SUITABILITY FOR GPU ACCELERATION

The algebraic multigrid (AMG) solver for Fluent simulations can be computationally intense, and computing requirements grow as the number of cells in the domain increase. Problems that contain less than a few million cells do not gain speed from GPUs because of communication overheads incurred in transferring matrices from or to CPUs. However, speedup is significant for meshes that contain tens and hundreds of millions of cells because the overhead is relatively small compared to the computing time in the AMG solver.

Would a coupled solver or a segregated solver benefit most from GPUs? In flow-only problems, typically the coupled solver spends about 60 percent to 70 percent of its time solving the linear system using AMG, making GPUs a good choice. Since the segregated solver spends only 30 percent to 40 percent of its time in AMG, GPUs may not be advantageous because of memory transfer overhead costs.

You can determine the AMG portion in a Fluent calculation (and therefore whether it is a good candidate for GPU employment) by adding the following command to the journal file for a CPU run:

```
/parallel/timer/usage
```

The information is reported near the end of the output file after successful completion of calculations. In the sample shown, the AMG portion is nearly 75 percent, so it is a good candidate for GPU implementation.

```
LE wall-clock time per
iteration: 12.299 sec (74.8%)
```

In addition, stiff matrices are difficult to solve and, thus, require more iterations in the AMG solver, making them ideal for GPUs.

An application that incorporates all these factors is an external aerodynamic calculation over automobiles and airplanes that can significantly benefit from using GPUs with ANSYS Fluent.

HOW TO SPEED UP ANSYS WITH GPUS
ansys.com/91accelerate1

BOOSTING SIMULATION PRODUCTIVITY

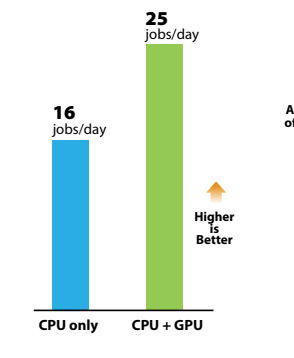
A critical performance metric to consider when evaluating GPUs is job throughput per day or speedup factor based on wall-clock time.

$$\text{Fluent speedup factor} = \frac{\text{total wall-clock time for CPU run}}{\text{total wall-clock time for GPU + CPU run}}$$

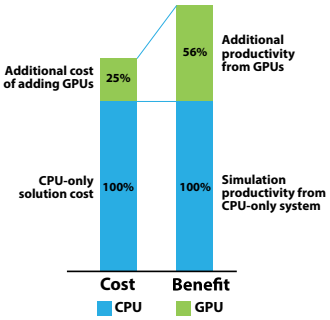
These metrics depend on the AMG portion of the total solution and associated speedup of that portion on GPUs.

GPU performance on external aerodynamics problems along with its value proposition is explored in the graphics.

To accurately account for the value proposition of GPUs, you must consider the system cost of both hardware and software, as well as the overall productivity

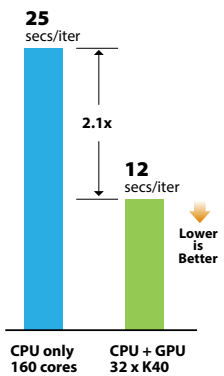


Iterations to reach convergence: CPU – 531, CPU + GPU – 566. Hardware: Intel Xeon E5-2680 (64 CPU cores on 8 sockets) 8 Tesla K40 GPUs. License: ANSYS Fluent and ANSYS HPC Workgroup 64

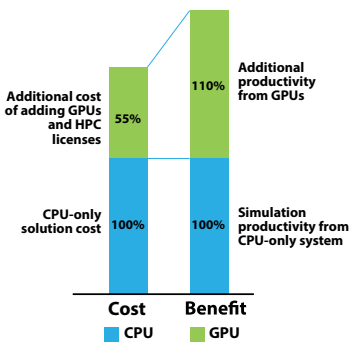


CPU-only solution cost is approximate and includes both hardware and software license costs. The benefit/productivity is based on the number of ANSYS Fluent completed jobs per day in a multi-user cluster

▲ A Fluent truck benchmark model consisting of 14 million cells was used but reconfigured as a steady-state pressure-based coupled solver problem. When running on 64 Intel® Xeon® E5-2680 CPU cores on a four-node cluster, the number of jobs completed to full convergence was about 16 per day. The number of jobs increased to 25 per day when eight NVIDIA Tesla® K40 GPUs were added to the system.



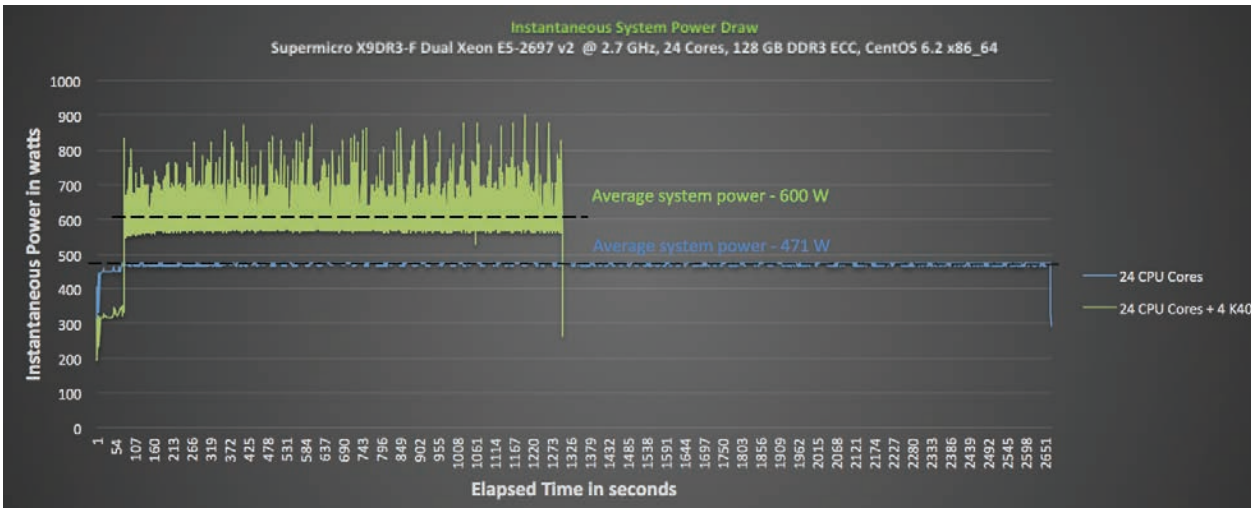
CPU: 8 Ivy Bridge nodes with 20 cores each. F-cycle size: 8. GPU: 32 Tesla K40, V cycle, size 2.



CPU-only solution cost is approximate and includes both hardware and paid-up software license costs. The benefit/productivity is based on the number ANSYS Fluent jobs completed per day.

▲ To examine the GPU performance and value for a large-scale CFD simulation, a generic Formula 1 car model with 140 million cells was run in a steady-state mode with the pressure-based coupled solver. Performance was evaluated based on the time taken per iteration over a period of 1,000 iterations. Adding GPUs decreased the time to solution by a factor of 2.1, while delivering 110 percent additional productivity at 55 percent additional system cost.

© 2015 ANSYS, INC.



▲ The instantaneous power drawn by a 24-core CPU-only system was compared with a CPU plus GPU system to do the same job. The CPU system drew 471 watts on average over a period of 2,651 seconds, which totals 350 watt-hours. Though the CPU plus GPU system drew an average of 600 watts, the job was completed in 1,302 seconds due to acceleration; therefore it consumed only 217 watt-hours. Compared with the CPU-only system, the GPU system resulted in 38 percent energy savings, which can be valuable in assisting organizations to achieve energy-efficiency objectives.

improvements. A CPU-only system (including memory, high-speed interconnect and the associated license cost of 100 percent) delivers 16 jobs per day in the truck benchmark, which is considered 100 percent benefit. Adding eight GPUs increases total system cost by 25 percent while the GPUs deliver 56 percent additional throughput per day. This demonstrates the value of GPUs in Fluent for aerodynamic calculations.

GPU acceleration of single-phase coupled flow problems is not just limited to aerodynamics simulations; it also includes internal flows. However, the Fluent 15.0 GPU capability is not yet offered for modeling other physical phenomena such as detailed chemical kinetics, radiation modeling with discrete ordinates and multiphase flows. Some of these features will be available in future versions along with performance improvements for the AmgX library through ongoing collaboration between ANSYS and NVIDIA.

REDUCE ENERGY CONSUMPTION

Big enterprises running simulations in large CAE clusters want to drive down energy consumption to reduce costs and/or meet broader corporate sustainability initiatives. At the same time, researchers and engineers demand high levels of computing power to model complex simulations and explore large design spaces. GPUs can fill this gap as they are optimized for higher throughput and performance per watt. In fact, large installations of GPUs are typically included in supercomputers to manage energy costs. The same benefits also apply to ANSYS Fluent simulations.

HARDWARE REQUIREMENTS

- Use NVIDIA Tesla GPUs for servers and workstations; use Quadro® GPUs for workstations.
- Configure Tesla GPUs like Tesla K40 or K80 or a high-end Quadro K6000.
- Cards with 12 GB to 24 GB of memory per GPU and high double-precision capacity are recommended.
- GeForce® GPUs, gaming class cards, are not recommended.

ACCELERATING ANSYS FLUENT USING NVIDIA GPUS
ansys.com/91accelerate2

GPUS PROVIDE A COMPETITIVE ADVANTAGE

GPU acceleration of ANSYS Fluent — the result of innovative GPU-based AMG solver developed by NVIDIA in collaboration with ANSYS — supports multiple GPUs to benefit those who are performing coupled flow problems that demand high solver computing power. GPUs demonstrated considerable acceleration in external aerodynamic benchmarks. This capability allows engineers to complete more simulations in the same time, and simulate larger and more complex models without project schedules slipping. Furthermore, GPUs consume less energy to do the same job in ANSYS Fluent when compared with a CPU-only system, which saves energy for large enterprises. GPU-accelerated simulations can reduce product development times — so companies can deliver higher-quality products and decrease time to market — resulting in a competitive advantage for these businesses.

GPU-accelerated simulations can reduce product development times, resulting in a competitive advantage for these businesses.

© 2015 ANSYS, INC.

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

Send address corrections to
AdvantageAddressChange@ansys.com

the electrification of vehicles

stricter fuel economy standards

driverless cars



The automotive industry is in the midst of a complete transformation.
Learn how simulation is leading that revolution.

2015 Automotive Simulation World Congress

June 2-3 • Detroit, Michigan USA

ansys.com/aswc