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

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
275 Technology 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, ALiAxe, Ansoft Designer, Arena, Ansys, Autodyn, BladeModeler, CFD, CFX, Chip Power Module (CPM), Chip Thermal Model (CTM), DesignXRF, DesignxSi, DesignXModeler, DesignSpaces, DesignXplorer, Engineering Knowledge Manager (EKM), Explicit STR, Fatigue, Fluent, Full-Wave SPIKE, HFSS, IC3D, CFD, Inpulx, Genx, Maxwell, Mesh Morph, Multiphysics, Nextem, OptiStruct, FlexPA, PathFinder, FlexPoint, Polyworks, PowerArtist, PowerArtist Calibrator and Estimator (PACE), Professional, Q3D Extractor, QuickEye, Realize Your Product Promise, RedHawk, Rigid Dynamics, RMModel, RTI, Power Model (RPM), SCADIE Display, SCADIE Lifecycle, SCADIE Suite, SCADIE System, Sentinel, Simwave, Simplorer, Simulation Drives Product Development, Solve on Demand, Structural, Super-Compact, TGrid, Tolem, TPA, TurboGrid, Vista TF, VeriFire, WinQ3D, 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. IC3D 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 IBM nCode. All other brand, product, service, and feature names or trademarks are the property of their respective owners.

Tackle your toughest ANSYS simulation tasks with the pure performance of Intel® Xeon® processors.

Discover the difference of Intel Xeon processors at www.ansys.com/DiscoverIntelXeon

Put your best on the line. Every time.
W e’re all familiar with the academic adage “publish or perish.” While the need to carry out groundbreaking research worthy of publication remains unchanged, today’s professors face new challenges. Engineering is becoming more multidisciplinary, as research teams collaborate across schools and departments — sometimes partnering with industry — to develop innovations such as artificial hearts and hybrid vehicles. This requires new simulation tools that capture the full range of physics.

Because they recognize the multiphysics breadth, capability, accuracy and scalability of ANSYS software, faculty today incorporate ANSYS tools in their research to a greater degree than ever. Each year, more than 8,000 academic papers are published based on studies conducted using ANSYS software. This special issue of ANSYS Advantage is filled with examples of exciting, leading-edge research enabled by ANSYS.

According to Dean of Engineering Gerald Holder at the University of Pittsburgh (see article on page 14), dramatic improvements in high-tech tools have supported enormous advancements in engineering research and education.

“Simulation software has been one of the key tools for an engineer, but today its speed and power have improved markedly,” Holder told us. “Not only have these technology improvements greatly accelerated what our researchers can accomplish, they have also democratized specialized tools like ANSYS, making it possible to put them into the hands of more students and research teams.” This theme was echoed again and again as we spoke to faculty worldwide for this academic-themed issue.

In university research labs, funding and physical space are at a premium, so it’s more vital than ever to minimize the use of large, costly physical testing equipment (such as wind tunnels) with a reliable, accurate, virtual alternative. With pressures to innovate on the rise, conducting advanced research via ANSYS software enables teams to publish their findings and contribute to the engineering knowledge base on an ongoing basis.

In the classroom, faculty members train future engineers to use simulation software from ANSYS for one simple reason: It’s part of the toolkit they need to master as future engineering professionals. Our industry partners repeatedly confirm the need to possess simulation skills, even for entry-level job candidates.

At ANSYS, we’re committed to collaborating with leading universities to increase and amplify their use of virtual solutions. To help accomplish this goal, we develop curriculum materials that help faculty teach simulation effectively. For example, our online student training programs help young engineers to leverage more ANSYS features and functionality as they use simulation for papers, theses or student competitions. Through our academic partnership program, ANSYS is a long-standing supporter of cutting-edge research and teaching — and that commitment remains unchanged.

In addition, in 2014, ANSYS is introducing a flexible and easy-to-deploy campus-wide licensing model that will further streamline faculty and student access to ANSYS software. For universities that use only a few ANSYS solutions, this model introduces them to multiphysics capabilities that represent the interdisciplinary product development practices of professional engineering teams.

We’re honored that ANSYS software is used every day by the world’s leading corporate engineering teams. We’re also proud to say that most engineers first encounter ANSYS solutions in the classroom or lab. Making this experience a positive one — while also supporting the advanced research that happens in universities — will always be a high priority for ANSYS. ☞

ANSYS is a long-standing supporter of cutting-edge research and teaching. Putting the power of ANSYS engineering simulation into students’ and researchers’ hands results in a win for everyone involved.
FEATURES

6 BEST PRACTICES
Making the Grade
Engineers need advanced skills to tackle today’s complex, multidisciplinary problems. By partnering closely with engineering schools, ANSYS helps ensure that the next generation of engineers is competent to tackle real-world problems in the highest-impact manner. In addition, at labs around the world, ANSYS fuels groundbreaking academic research that has the potential to shape the future.

14 THOUGHT LEADER
Top of the Class
The University of Pittsburgh’s Swanson School of Engineering was an early adopter of engineering simulation, using ANSYS software for decades to train students in industry-standard practices — while also accelerating faculty research efforts. Swanson’s Dean Gerald Holder discusses the evolving role of technology in meeting future education and research challenges.

19 ACADEMIC
Learning Experience
Collaboration between Cornell University and ANSYS provides access to simulation software for researchers, instructors and students.

24 ACADEMIC LICENSING
Making the Leap to Campus-Wide Licensing
Longtime ANSYS partner LEAP Australia is helping Australasia’s leading universities to embrace license bundling to expand research and teaching objectives.

28 ACADEMIC RESEARCH
In Your Blood
Patient-specific simulation helps improve endovascular aneurysm repair.

32 FLUID–THERMAL SYSTEMS RESEARCH
Run Like the Wind
Computational fluid dynamics is used as a virtual wind tunnel to optimize the design of scramjet engines at up to Mach 6.5.

37 ROBUST ELECTRONIC SYSTEMS RESEARCH
Live Wire
Researchers designed an unusually compact wearable antenna that covers the entire instrument, scientific and measurement band.

41 ACADEMIC STUDENT COMPETITION
Competitions in Education: Learning and Making Engineering Fun
Every year, students around the world use simulation in engineering competitions that are not only fun but prepare them for future careers.

ABOUT THE COVER
Student teams, like Blue Sky Solar Racing from the University of Toronto, participate in competitions around the world.
DEPARTMENTS

46 AEROSPACE AND DEFENSE

Breaking the Code
Turbomeca reduces development time by using ANSYS SCADE Suite for helicopter engine control software development.

50 BEST PRACTICES

Size Matters
The ANSYS Application Customization Toolkit is put to use to more accurately size welds by taking existing standards into account.

WEB EXCLUSIVES

These additional articles are available at www.ansys.com/magazine

Breakthrough for Brake Design
TRW has validated a new method that uses ANSYS Mechanical software to establish the initial contact and compute sliding contact between pads and disc. This approach accounts for system contact conditions, enabling brake noise to be simulated and reducing the need for physical testing to tune the models.

Losing Contact
Instead of using standard gears, engineers are turning to harmonic drives to deliver the robust performance required in space applications, such as power transmission systems for lunar vehicles. Multiphysics simulation helps to improve design of a cryogenic non-contact harmonic drive.

Cooling Out
The next step in understanding the laws of nature: Thermal simulation helps to design a new generation of magnets for the Large Hadron Collider.

Free Passage
Researchers at Linköping University address diagnosis and treatment planning for aortic coarctation by using ANSYS CFX computational fluid dynamics (CFD) software to simulate turbulent blood flow both before and after surgical intervention.

Green Energy Design at the University of Leeds
The Energy Technology and Innovation Initiative focuses on engineering innovations that help achieve more efficient, greener energy production.

Extra: Making the Leap to Campus-Wide Licensing
The Monash Motorsport team and RMIT use ANSYS software extensively for research and competition cars. Additional details.

Extra: Competitions in Education: Learning and Making Engineering Fun
Much more information is available on the student competition teams than could be covered in the print issue of ANSYS Advantage. Learn more about how these teams achieve their goals.

DEPARTMENTS

54 ANALYSIS TOOLS

Drive For Excellence
The key to developing reliable electric drives is creating a best-practice workflow that incorporates model-based systems engineering and embedded software simulation.
Simulation in the News

ELECTROMAGNETIC SIMULATION SUITE FOR PCB DESIGN
Interference Technology
interferencetechnology.com, January 2014

New technology from ANSYS delivers end-to-end signal integrity analysis in a single user interface. ANSYS SIwave-DC targets the DC analysis of low-voltage, high-current PCB and IC packages, enabling the assessment of critical end-to-end voltage margins for reliable power delivery. SIwave-PI adds AC analysis to accurately model power delivery networks and noise propagation on PCBs.

RIDING THE RAILS AT 200 MPH
Peninsula Publishing
penpubinc.com, November 2013

With European high-speed rail incidents making headlines, Eric Bantégnie from ANSYS discussed how systems designed to protect train passengers are being greatly scrutinized. Today’s complex software systems control speed, help avoid other trains on the track, and make sure that the train travels as intended. These systems work very well if properly implemented. Systems and software engineers design elaborate automated systems, and errors can result from a misunderstanding of the requirements and specifications for the system in development. Engineers continue to refine these electronic systems, moving toward more-automated systems that greatly improve passenger and train safety.

ANSYS 15.0 RELEASED
Engineering.com
engineering.com, December 2013

The new ANSYS 15.0 focuses on pre-processing, structural, fluid and electromagnetic simulation capabilities. Improvements, including pre-processing and meshing capabilities, allow for better use in many different physics simulations regardless of range, size or complexity of the model. Structural analysis enhancements enable easy design of composite materials; fluid dynamics includes upgraded reliability for turbomachinery flow; HPC is improved by a factor of five. Electronics analysis offers an improved electric motor design process. In addition, specialized meshing for silicon substrates, printed boards and redistribution layers is available. For systems-level analysis, ANSYS 15.0 can embed mechanical control code using the SCADE suite.

The new release of ANSYS improves on core technology employed by many users, including pre-processing, structural, fluids and electronics.

USING SIMULATION TO DEVELOP OFFSHORE WIND TURBINES
Konstruktion.de
konstruktion.de, November 2013

When developing offshore wind farms, engineers face considerable challenges in designing turbines and towers that will withstand specific forces — high wind speed, water currents and waves — as well as realizing the least expensive option. REpower Systems turned to ANSYS simulation software to design, test and optimize virtual models before building expensive prototypes.

OVERCOMING DESIGN CHALLENGES OF NEXT-GENERATION UASs
SAE International
sae.org, October 2013

Successful unmanned air system (UAS) design plans must incorporate advanced power and thermal management strategies in the earliest stages of the design process. The driving technologies are low-power design and 3-D integration, according to Rob Harwood of ANSYS.
In developing its high-bandwidth real-time oscilloscope (the first to reach the 60-GHz barrier), Agilent Technologies utilized ANSYS software to model challenging electrical, mechanical and thermal requirements. The major obstacle in designing the calibration head was cooling; by using simulation, the team delivered a successful first-pass model.

To optimize the design of an electrical calibration source as a new standard for measurement accuracy, Agilent Technologies engineers turned to simulation to exceed challenging requirements.

A leading developer of chemistry simulation software, Reaction Design signed a merger agreement with ANSYS. The flagship product, CHEMKIN®-PRO, is the gold standard for modeling and simulating gas-phase and surface chemistry, offering engineers the ability to predict the effects of chemistry in a combustion system. This is critical to developing competitive products in transportation, energy and materials processing applications.

A survey of the chip industry indicated that no segment is exempt from reducing its product’s power profile. Pressure from sectors such as mobile is forcing more companies to explore techniques such as dynamic voltage and frequency scaling, and power gating to reduce leakage power. In this round-up article featuring industry thought leaders, William Ruby of ANSYS suggested that widespread clock gating — baseline entry into power optimization — has reduced dynamic power usage up to 40 percent, while back-end optimization techniques are responsible for another 10 to 20 percent savings. Companies want to take the existing optimization further. Virtual prototyping is one way that hardware and software can be executed together to solve this kind of problem.

Infiniti Red Bull Racing wrapped up its fourth consecutive championship in 2013 F1 competition. Its most recent car, finishing with 13 victories, was optimized using ANSYS CFD and HPC. The team’s chief technical officer said the improvements were “all in the details. We’ve tidied up some bits we thought could be improved. Our simulation results tell us we’ve taken a step forward.”

Hyperloop Transportation Technologies plans to produce a working Hyperloop prototype by 2015 — a concept developed by Elon Musk to transport humans between cities in pods that are accelerated to near-supersonic speeds. Key partnerships have been developed to help with the design process, including ANSYS, which created some of the earliest simulations of the rail. “Cost is paramount,” said Marco Villa, coleader of the project.

A major barrier to improving simulation speed is how detailed simulations become, resulting in slow runs and days or weeks to get final simulation analysis. However, high-performance computing is an option that some organizations have been hesitant to adopt. In a survey conducted by ANSYS and IBM, 59 percent of the respondents said they need evidence that HPC has technical benefits before adopting HPC capabilities. In a webinar hosted by ANSYS and IBM, the benefits of HPC were presented so that organizations don’t have to fret over the hardware, expertise or support tied to upgrading to an HPC system. In engineering disciplines, there are tides in the affairs of technology, and HPC is the next wave.
Making the Grade

Engineers need advanced skills to tackle today’s complex, multidisciplinary problems. By partnering closely with engineering schools, ANSYS helps ensure that the next generation of engineers is competent to tackle real-world problems in the highest-impact manner. In addition, at labs around the world, ANSYS fuels groundbreaking academic research that has the potential to shape the future.

By Murali Kadiramangalam, Director, Academic Program, ANSYS, Inc.
Since ANSYS was founded in 1970, a company hallmark has been its close connection to the academic world. Just as multiphysics engineering simulation software has revolutionized the way professional engineers design and test products, it has changed the work of academic researchers — allowing them to produce groundbreaking technical research reliably, faster and more cost-effectively than ever. In classrooms around the world, ANSYS solutions have helped generations of students prepare to tackle real-world engineering challenges.

Each day and in every corner of the globe, thousands of engineering faculty, researchers and students leverage the power of ANSYS. Annually, at least 8,000 academic papers and 12,000 dissertations rely on simulations using ANSYS solutions. More than 2,400 teachers have embedded the tools into their engineering curricula, with over 86,000 students enrolled in these classes. Another 10,000 faculty employ the software in academic research.

In the world’s leading engineering schools, ANSYS solutions have proven their value not only in delivering the highest-quality, most-relevant education to students — but in supporting the kind of outstanding research that attracts funding, while helping to increase national and global rankings.

REAL TOOLS FOR REAL ENGINEERING PROBLEMS

In this special academic issue of ANSYS Advantage, university professors from around the world describe why they use ANSYS software in their classrooms. They all give basically the same answer: because simulation is a requisite part of the skill set that today’s entry-level engineers are expected to have.

In the professional workplace, the majority of engineering teams have reduced cost- and time-intensive physical experimentation and testing, replacing them with virtual product design and verification. The sooner that entry-level engineers can play active roles in simulation-driven product development, the greater their value to an engineering organization. According to Professor Marius Geller of Dortmund University of Applied Sciences, graduate students with proven simulation skills are often recruited before they even finish their master’s degrees.

ANSYS software allows academic researchers to produce groundbreaking technical research reliably, faster and more cost-effectively than ever.
Hands-On Learning at the University of Iowa

At the University of Iowa, Professor Frederick Stern has developed a customized educational approach using the ANSYS Workbench interface and tools that help students quickly learn to use ANSYS Fluent for advanced computational fluid dynamics (CFD) simulations, while reinforcing introductory and intermediate fluid mechanics concepts. This work was done in partnership with research scientist Dr. Maysam Mousavirad, graduate student Timur Dogan and undergraduate student Michael Conger. This approach, and a series of teaching modules created by Stern and his team, form the foundation of CFD laboratories for two undergraduate courses at the university: Mechanics of Fluids and Transport Processes, and Intermediate Mechanics of Fluids.

“I wanted my students to be able to perform complex CFD simulations without a steep learning curve,” says Stern, who is the George D. Ashton Professor of Hydroscience and Engineering at Iowa. “So I worked with academic experts at ANSYS to create a unique approach that teaches students systematic CFD modeling, numerical methods and procedures in a hands-on, user-friendly, interactive manner.”

Stern’s educational approach for ANSYS Fluent automates the CFD simulation process, leading students step by step through setup and solution of a range of realistic engineering problems. This approach has proven popular with students at the University of Iowa and has also been adopted by other universities.

A number of recent ANSYS initiatives are making it easier than ever for students to access and apply simulation software. One great benefit is the flexibility built into the student version of ANSYS software, which can be accessed in the dorm room, classroom or lab. Students can take advantage of a range of live and recorded training sessions as well as web-based learning portals to improve simulation skills in a self-guided manner. The company sponsors a student-focused Facebook page, supported by a growing user community, where students can exchange best practices, ask questions and offer advice.

Around the world, student competition teams rely on ANSYS software to design cars, aircraft, robots and other products for competition. By leveraging the power of engineering simulation to make design candidates sturdier, lighter-weight, faster and more energy-efficient, they attack the same complex problems they will face as working engineers. ANSYS technology has helped dozens of teams win awards, scholarships and other accolades.

Universities add significant value for the world of industry by incorporating ANSYS software into curricula and research programs.
Robust Fluid-Mechanical Design

At Dortmund University of Applied Sciences in Germany, Professor Marius Geller challenges his students to conduct extremely sophisticated engineering simulation exercises as part of the master's in Mechanical Engineering program. Each year, only 15 to 20 students are admitted to this highly competitive program, which is designed to provide young engineers with the advanced skills — such as simulation — that they need to succeed in the workplace.

Geller teaches an 18-week course called Computer Simulation in Mechanical Engineering, which begins by introducing students to the mathematical and physical problems underlying performance of a variety of products — ranging from Formula 1 cars and robotic hands to industrial compressors and ship propellers. Next, students begin to optimize product performance by modeling these problems using ANSYS software.

The course culminates in a real-world engineering project, in which each student applies engineering simulation and optiSLang to robustly design a product optimized for characteristics such as strength, energy efficiency, light weight or aerodynamics. While they work individually, all students in the class focus on designing the same product. In recent years, Geller’s design projects have included a boat, a wind turbine and a bicycle.

“Whether they come into the course with any knowledge of ANSYS or engineering simulation, by the end, my students are working at a highly sophisticated level,” notes Geller. “They create robust products by using the same parametric design principles that the world’s leading engineering teams apply. For example, if they are optimizing a ship’s hull, they set up ANSYS software to test more than 200 design variations in an automated fashion. They apply multiple physics, simply because that is the way engineering teams work in the real world today.”

Why is ANSYS the right tool on which to build this challenging class? “I have used ANSYS for almost 20 years, and I find it to be the best software for multiphysics studies,” says Geller. “Its ability to produce native results for both finite element analysis and fluid dynamics is unmatched. The user-friendly ANSYS Workbench platform — combined with the software’s parametric capabilities and compatibility with high-performance computing environments — place even the most advanced simulations within the reach of my students.”

Yet another reason is that, in Geller’s opinion, ANSYS is the industry-standard simulation software for German engineering teams. “Recently, one of my students received 10 job offers when he showed his ANSYS simulation project to potential employers,” adds Geller. “In fact, it has become a challenge to keep students in the program after they learn ANSYS, simply because their expertise in engineering simulation makes them so attractive to recruiters.”
As the editor-in-chief of the journal Building and Environment, Professor Qingyan Chen of Purdue University is an internationally recognized expert in designing comfortable, healthy and environmentally responsible environments.

In both his teaching and research at Purdue — where he is the Vincent P. Reilly Professor of Mechanical Engineering — Chen relies on ANSYS Fluent to visualize and address air quality issues in interior and exterior spaces. His work is aimed at designing innovations such as new building facades or heating, ventilation and air conditioning (HVAC) systems that optimize occupants’ health and comfort, while also meeting environmental goals.

In Chen’s popular class Indoor Environment Analysis and Design, seniors and graduate students apply Fluent to model interior spaces and map air flow patterns. Students test the effects of various architectural changes — such as operable windows or new HVAC vent locations — on air quality, temperature, humidity and energy efficiency. Chen’s students also work to ensure healthier spaces by creating airflow patterns that minimize the spread of viruses and other airborne contaminants.

“ANSYS Fluent is an ideal tool for students, because it has an intuitive interface that is easy to learn,” notes Chen. “In addition, Fluent is very stable, even when managing the numerically large simulations that are typical in environmental design. Students can become easily frustrated by slow processing times or system crashes, but these are not an issue with ANSYS software.”

A research team at Syracuse University leveraged ANSYS Mechanical to develop a finite element technique to merge 3-D scans, MRI images and PET scans to improve detection of cancerous breast tumors. Experimental studies with actual cancer patients confirmed the accuracy of this technique.

ADDRESSING THE GLOBAL TALENT SHORTAGE

By bringing advanced simulation technologies into classrooms and lectures halls, universities do far more than help students. They fill a pressing industry need for qualified entry-level engineering employees.

Today, companies worldwide face a shortage of engineering staff with the simulation skills needed to drive time and costs out of the product development cycle. The reasons for this are varied. In North America, where simulation is a mature discipline, many simulation experts are reaching retirement age. In Asia, as more and more organizations realize the benefits of simulation, industry demand is simply outstripping the current supply of qualified candidates.

ANSYS has partnered with universities around the globe to develop curriculum materials that prepare students to enter the workforce with simulation knowledge. And because leading professional engineering teams use the software, ANSYS is uniquely qualified to develop these materials, ensuring that they are relevant and practical, reflecting actual workplace demands.

Academic partners can easily integrate the technology into their undergraduate and graduate programs via a full range of tutorials, videos, books and web-based portals developed by education experts. Many colleges have collaborated with ANSYS to create their own customized teaching materials.

Universities that incorporate simulation into their engineering programs can expect to realize increased placement rates as their graduates enter today’s job market — an environment driven by advanced technologies that reduce costs, support
In addition to using ANSYS Fluent to introduce graduate and undergrad students to concepts in fluid dynamics, Professor John Cimbala has relied on that software in his own research for over two decades. Cimbala leads the Hydropower Research Program at Pennsylvania State University, which applies funding from the U.S. Department of Energy to solve real-world industry problems via advanced simulation and analysis.

Cimbala's current research efforts focus on understanding turbulent flow patterns as water flows through the hydroturbine at a hydropower plant. Certain characteristics of the flow — such as cavitation — can have a negative impact on equipment, shortening its life and increasing maintenance requirements.

“When water exits a hydroturbine, it expands and makes a 90-degree turn, which causes an unsteady vortex rope to form inside the flow. This is a real problem, causing vibration and material erosion inside the exit shaft — while also limiting the operating range and efficiency of the turbine,” notes Cimbala.

The professor and his research team use the detached eddy simulation (DES) capabilities of Fluent to simulate flow conditions that combine high turbine performance with minimal vibration and unsteadiness in the exit shaft, called the draft tube. They also are exploring the effects of new equipment and process innovations, such as injecting the vortex rope with a jet of water to minimize its effects.

“Obviously, turbulence is a complex problem, and our models routinely include tens of millions of cells,” says Cimbala. “We are running massively parallel simulations across hundreds of high-performance computing processors. We have been very pleased with the fidelity, accuracy and speed of ANSYS Fluent as it runs these simulations. The software is easy to work with and well suited for this kind of unsteady-state work.”
Professor Ever Barbero has literally written the book on using ANSYS software to perform finite element modeling (FEM) for advanced composites materials. This faculty member at West Virginia University recently published the second edition of his popular book, *Finite Element Analysis of Composite Materials Using ANSYS*.

“In many industries, composites are now the material of choice for combining light weight with high strength,” says Barbero. “But it can be difficult to mimic the behaviors of the various materials, including fibers and layers, via engineering simulation. I have found ANSYS capabilities to be ideal for accomplishing this task — providing fast, accurate answers about how to best configure composites in designing new products.”

Campus-wide licensing assists different departments, schools and research groups in coming together for interdisciplinary studies. The multidisciplinary nature of research is probably the one trend that has most impacted engineering schools in recent years — and ANSYS has responded by creating seamless, unrestricted access and tool sharing via the license-bundling program. Academic teams can venture into new areas and study systems-level problems without incurring additional costs or navigating a cumbersome procurement process.

As academic customers worldwide see the advantages, campus-wide licensing is growing dramatically. For example, in Australia, ANSYS partner LEAP has helped virtually every leading university on that continent access the software via campus-wide licenses. More and more customers are expected to embrace license bundling as a solution to academic budget cuts, as well as in recognition of the software’s ability to support collaborative, interdisciplinary study.
Researchers at National Taiwan University of Science and Technology used ANSYS Workbench to investigate different methods for lumbar fusion. Simulation helped to optimize implant procedures for outcomes such as range of motion, spinal stress and implant stress. Nonlinear contact conditions and tension-only springs were applied to simulate bone-implant interfaces and spinal ligaments.
The Swanson School of Engineering at the University of Pittsburgh combines the traditions of the city’s industrial past with a forward-looking perspective that focuses on innovations in areas such as bioengineering, alternative energy and sustainability. More than 2,600 students major in one of six traditional engineering topics — or focus in a multidisciplinary area such as civil engineering and chemical engineering or manufacturing systems engineering.

With proximity to the world headquarters of ANSYS — and an engineering school named for ANSYS founder John Swanson — the University of Pittsburgh was an early adopter of engineering simulation software, in both the classroom and research lab. Under the guidance of Gerald D. Holder, who has been the U.S. Steel Dean of Engineering since 1996, Pitt’s Swanson School has grown in its adoption of advanced technologies that make the work of engineers faster and more cost-effective — without sacrificing accuracy.

Dean Holder recently spoke with ANSYS Advantage about the challenges and opportunities he envisions for the future of engineering education. Based on his unique perspective as leader of a top U.S. engineering school, Holder had much to say about the way technology has shaped, and will continue to influence, engineering classrooms and academic research laboratories.
During your tenure as dean, what are some of the biggest changes you’ve witnessed in engineering education?

More than any other single factor, technology has changed our entire world over the past two decades — and we certainly feel its enormous impact in engineering education. It has dramatically altered both our labs and our classrooms. For example, simulation software has always been one of the key tools for an engineer, but today its speed and power have improved markedly. Not only have these technology improvements greatly accelerated what our researchers can accomplish, they have also democratized specialized tools like ANSYS, making it possible to put them into the hands of more students and research teams.

Today, more than at any other time in Swanson School’s history, technology is an essential part of our curriculum and physical environment. We have more than 750 workstations loaded with ANSYS software across our classrooms, student labs and research facilities. Close to 100 percent of our students work with ANSYS in the classroom as part of their hands-on instruction. All our buildings are wired for high-speed, high-volume data transmission so we can support numerically large simulations and other computing tasks. We have created some flipped classrooms in which students watch lectures remotely then spend classroom time working more actively on hands-on problem-solving.

Now we have created a director of technology position to ensure that the Swanson School stays ahead of technology trends, allowing us to give students the skills they will need in the workplace. We have a 95 percent success rate in placing our graduates, which reflects our close connection to industry and partners like ANSYS — and this provides us with a better awareness of the knowledge and experience that employers are looking for. Hands-on technology expertise is increasingly a vital part of the skill set that our graduates need.

Do you see any downside to all the technology in the college classroom today?

I think the only potential danger is that students can rely too heavily on advanced tools like ANSYS software that do all the work for them. At the Swanson School, we make sure that students understand the basic mathematical and physical principles that underlie engineering simulation. Our curriculum covers all the traditional engineering topics, providing that solid foundation, before introducing computer-aided design and verification tools.

There’s no doubt that ANSYS can streamline and accelerate design and experimentation. It can also dramatically amplify the work of engineers by allowing them to consider thousands of design variations, instead of constructing one prototype of a single design and running repetitive physical tests. But engineers also need to set the right parameters and define the right conditions for their simulations — which requires a deep understanding of math and science. Even as technology improves and makes an engineer’s job easier, we can’t lose sight of providing a strong fundamental engineering education.

Both ANSYS and the Swanson School are committed to providing engineers with the skills and toolsets they need.

Technology is an essential part of our curriculum and physical environment.
What is the biggest challenge confronting U.S. engineering programs today — and how is Pitt responding?

Worldwide, we’ve seen an explosion in the growth of engineering graduates. In 1996, there were approximately 50,000 B.S. degrees awarded. Today, that number is more than 90,000. Much of that growth is happening in Asia.

For the Swanson School, that means two things. First, we must attract more international students at the undergraduate level and become more recognized in the global engineering community. About half of our graduate students come from outside the U.S., but we need to recruit more heavily overseas for our undergraduate programs.

Second, we must join forces with other universities to increase enrollment here in the United States in science, technology, engineering and mathematical (STEM) topics. President Obama has challenged American universities to graduate 10,000 more engineering majors per year. Certainly the advanced technologies in our classrooms, our leading-edge research, our close partnership with industry leaders like ANSYS, and our collaborations with corporate partners are helping to make the Swanson School more appealing to students. Our strong recruiting efforts, especially among women and under-represented minorities, are helping to increase our enrollment.

For urban schools like the University of Pittsburgh, however, growth itself can be a challenge. We are landlocked, with no real physical space to expand. To address this obstacle, we’re currently undertaking a $150 million renovation of our 40,000-square-foot engineering building. This facility upgrade will not only help us attract new students, but it will provide space for our increased enrollment as we support President Obama’s STEM education goals. We’re taking this challenge very seriously.

The Swanson School has always had a strong focus on cooperative education. Why is this concept so important?

About 60 percent of our undergraduates complete a cooperative education experience before they graduate, and that’s a statistic we’re very proud of. As much as we try to prepare students for the real world...
To create a hands-on learning experience, Kimber first has his students experimentally measure a machined part with geometric complexities. They manually calculate the areas of stress created by material irregularities, such as notches and holes, using the experimental data.

Next, after learning the basics of numerical modeling, students recreate the part geometry in ANSYS Mechanical. They repeat their experimental measurements and calculations in the virtual world to test the accuracy of their manual work. In addition, the flexibility of ANSYS software enables the students to assess a discrete number of new geometry variations. As they change the size or placement of a hole, they can immediately see how stress loads distributions change across the complex part.

“ANSYS provides a user-friendly, graphic way to understand exactly how different stress loads conditions result from the smallest geometry choices,” Kimber explains. “I could talk about stress equations for hours, but ANSYS provides an immediate way to see and understand the effects of stress distribution on a real-world product specimen. This experience not only increases my students’ familiarity with simulation software — which they will need to use on the job after graduation — but I believe it also makes them more thoughtful designers.”

How would you describe your relationship with ANSYS?

We have always had a terrific relationship with ANSYS. Over the years, the company has been many things to the Swanson School. Obviously, ANSYS is a supplier of industry-standard software tools that our students will need to use in the workplace. Through its flexible licensing agreements developed for academia, ANSYS has allowed Pitt to create hundreds of workstations loaded with its software across our classrooms and labs.

In addition, ANSYS has been a strong supporter of our cooperative education programs. We have a 95 percent success rate in placing our graduates.
program for over two decades. Since 1990, more than 200 Pitt undergrads have worked at ANSYS as co-ops, and today there are at least 30 full-time employees at ANSYS who began their careers as co-ops. ANSYS has also been an important source of philanthropy and support through its founder, John Swanson, who earned his Ph.D. in applied mechanics from the University of Pittsburgh in 1966. His generous financial support led Pitt to name its engineering school for John in 2007. But, equally important, John is generous with his time in visiting the campus and mentoring our young engineering students. He is obviously an inspiration to the next generation of engineering innovators.

Although John Swanson retired from ANSYS in 1999, our relationship with ANSYS remains very strong today. We are both committed to providing engineers with the skills and toolsets they need to do their jobs faster, more cost-effectively and more efficiently. As I look toward the future of engineering education, I am grateful that the Swanson School can continue to rely on our close partnership with ANSYS to accomplish our shared objectives.

Pitt’s Kevin Chen applies ANSYS Mechanical in support of his leading-edge research aimed at understanding the microstructural properties of silica optical fibers used for a range of telecommunications and sensing applications. Chen is an associate professor and Paul E. Lego faculty fellow in the Swanson School’s Department of Electrical and Computer Engineering.

Chen’s innovative work with ANSYS software has been supported by a Faculty Early Career Development (CAREER) award from the National Science Foundation, which provided more than $420,000 in funding. Highly prestigious and highly competitive, CAREER awards honor junior faculty members for their outstanding accomplishments in both research and teaching.

While many engineers use ANSYS Mechanical to assess the structural strength of materials for large civil and mechanical applications, Chen is breaking new ground by applying simulation software at micro and nano scales. In his fiber optics laboratory at the University of Pittsburgh, Chen and his research team conduct micro- and nano-mechanical structural analysis of silica fibers that are only 125 microns in diameter.

“When silica fibers are used for telecommunications and fiber optical sensing applications, their performance is inherently limited by their structural properties, most notably their response to physical stresses,” notes Chen. “To make these fibers mechanically highly robust beyond solid silica (for telecommunication) and sometimes highly susceptible (for sensing applications) to external environments, we are engineering them with microstructured airholes that distribute stress to the core of the fiber in an optimal way, given a particular application.”

Chen uses the finite element analysis (FEA) capabilities of ANSYS Mechanical to simulate stress distribution across these tiny fibers with even tinier holes (10 to 100 nanometers). According to Chen, ANSYS software is his tool of choice because of its ability to build an ultrafine mesh that replicates both the airholes and the fiber core in a time- and cost-effective manner.

“Because optical fibers are produced in batches of thousands of meters at once, trial and error is not a feasible solution for making manufacturing improvements,” explains Chen. “We need to accurately predict the fibers’ performance characteristics before we commit to an expensive production run. ANSYS Mechanical allows us to do that.

“As my research team changes the shape, size and orientation of airholes, we ask ourselves, ‘Will this provide a 10 percent improvement in fiber strength — or a 10-fold improvement?’ ANSYS software enables us to answer these types of questions and to quantify our design modifications with complete confidence,” Chen says. “We could not accomplish our important research without simulation software such as ANSYS Mechanical.”
Learning Experience

Collaboration between Cornell University and ANSYS provides access to simulation software for researchers, instructors and students.

By ANSYS Advantage Staff

ANSYS and Cornell University have developed a unique collaboration that has flourished for well over a decade, helping to extend Cornell’s reputation as one of the world’s leading research institutions. Many engineering classes at the university utilize ANSYS software, giving students the opportunity to learn industry’s leading software while they immerse themselves in engineering study. Furthermore, Cornell has developed an online learning wiki system based on ANSYS software that is used by many universities. Beyond the classroom, student competition teams have established an unparalleled track record in engineering award-winning racing cars, satellites, unmanned aerial vehicles and many other designs using ANSYS tools. In addition, research teams comprising faculty and graduate/undergraduate students have extended the frontiers of knowledge in many fields using ANSYS software.

“ANSYS provides the leading software for designing physical objects and fluid modeling,” says Dr. Lance Collins, dean
Cornell’s research teams have extended the frontiers of knowledge in many fields.

As bio-analytical microsystems and biosensor devices become increasingly miniaturized, it is much more challenging to efficiently mix two or more fluid streams. The Bioanalytical Microsystems and Biosensors Laboratory is embedding electrospun nanofiber mats into microchannels as a means of dispersing solutes between two initially unmixed solutions. The team used ANSYS Fluent for CFD analyses of solution flow and species transport.

Cornell Formula SAE Racing created its first-ever ultra-lightweight carbon fiber monocoque, which incorporates body and frame into one structural element. Cornell undergrads used ANSYS Composite PrepPost to evaluate dozens of possible designs by varying the composite ply material and number, and orientation of plies throughout the monocoque.

of the College of Engineering. “We have a license agreement that gives our students and faculty virtually unlimited access to a full range of powerful computer-aided engineering tools. We have a room outfitted with computers that serves as a lab for the many classes that teach simulation skills along with engineering subject matter. Students are stimulated by gaining the ability to solve difficult physical projects, and the classes that use the lab are nearly always full.”

John Swanson, the founder of ANSYS and a graduate of Cornell, has endowed a faculty position focused on the integration of modern computer-based simulation technology into the engineering curriculum. Rajesh Bhaskaran, Swanson Director of Engineering Simulation in the Department of Mechanical and Aerospace Engineering, has developed web-based tutorials and other curriculum materials that help students learn to use ANSYS Mechanical finite element analysis (FEA) and ANSYS Fluent computational fluid dynamics (CFD) software packages. He also assists in integrating the software into Cornell’s curriculum.

Bhaskaran has overseen the development of SimCafe (simcafe.org), a wiki-based online learning system for simulation technology. SimCafe incorporates a collection of web-based tutorials and homework problems, developed by Bhaskaran and Cornell students. Currently, SimCafe has more than 40 modules that cover learning FEA and CFD with ANSYS software. Each module contains a step-by-step tutorial that shows students how to solve a selected problem using ANSYS software.

Bhaskaran is also developing an online course called Introduction to Practical Engineering Simulations through Twenty Case Studies in ANSYS Software that is planned to be offered starting in the fall semester 2015. The course will be multidisciplinary, introducing students to FEA and CFD applications in solid mechanics, fluid mechanics...
Cornell researchers are developing a high-efficiency, low-emissions cookstove that also produces biochar, another term for charcoal, to improve soil productivity. The team used ANSYS Fluent extensively during the design process.

Researchers in the Department of Biomedical Engineering at Cornell use simulation to study hemodynamics of embryonic hearts and valves. Hemodynamics may be an important stimulator for cardiac development, as slight variations in normal hemodynamics may lead to heart defects.

Practical Engineering Through Case Studies

Cornell plans to offer an online course called Introduction to Practical Engineering Simulations through Twenty Case Studies in ANSYS Software starting fall semester 2016. In this course, students can develop the conceptual understanding and software skills required to effectively use simulation technology for engineering analysis and design. Case studies come from eight mechanical and aerospace engineering courses at Cornell University as well as from engineering practice. The course will draw upon learning modules on the SimCafe wiki at simcafe.org. The simulation tools used are ANSYS Mechanical for FEA and ANSYS Fluent for CFD.

Each case study starts with a problem statement followed by a tutorial showing the solution steps and rationale behind the steps. Before launching into ANSYS software in each case study, students undertake a pre-analysis phase that involves considering the mathematical model, numerical solution approach, and hand calculations to predict expected results and trends. After obtaining the software solution, students proceed to a verification and validation step that involves formally checking results by assessing whether the solution honors the physics contained in the mathematical model, the level of numerical error, and comparison to prior hand calculations.

Each case study ends with exercises that take students through a guided exploration of the underlying principles, software capabilities and limitations.

Mini-lectures are embedded within each case study to explore underlying principles on a just-in-time basis. “We intend for participants to learn how to deploy engineering simulation like an expert by practicing to think like an expert,” Bhaskaran explains. “In the process, they can move beyond a recipe-type approach to simulation that is characteristic of beginners.” The course utilizes the student version of ANSYS software, which is available for $25. Prerequisites include basic calculus, including integrals, differentials and differential equations.

ADDITIONAL RESOURCES
SimCafe Makes it Easier to Learn Principles of Simulation

The SimCafe wiki (simcafe.org) developed by Cornell contains simulation learning modules covering a broad spectrum of fields: solid mechanics, fluid mechanics, heat transfer and dynamics. SimCafe modules integrate industry-standard ANSYS simulation technology with innovative online pedagogy that links multiple courses and disciplines in a novel way. SimCafe is being used in 10 mechanical and aerospace engineering courses at Cornell as well as at other universities in the U.S. and abroad. It is also used by students and others to engage in self-directed study. SimCafe received more than 132,000 unique visitors from 140 countries during the 2012–2013 academic year, an increase of 18 percent from the previous year. This included 59,000 visitors to the section on FEA using ANSYS Mechanical and 73,000 visitors to the section on CFD using ANSYS Fluent. Over 55 percent of visitors return.

The learning goal is to enable students to approach simulation like experts by moving beyond a recipe-type approach common in online tutorials. This is accomplished by employing a common process across modules and disciplines to promote expert thinking. The pre-analysis step in the SimCafe process emphasizes the mathematical model, assumptions, how the model is solved numerically, and hand calculations of expected results/trends. There is a direct link between each online module and traditional course content. The verification and validation step undertakes a systematic process for checking results, including comparison with prior hand calculations. The SimCafe process and pedagogy have been developed via an NSF grant.

SimCafe learning modules are available for free online at simcafe.org and run side by side with ANSYS software. Many of the learning modules juxtapose the numerical approach in ANSYS with the traditional textbook approach. For instance, each FEA learning module leads the user through the steps involved in solving a selected set of problems in solid mechanics or heat transfer. The module provides the solution steps as well as the rationale behind them, so the user learns the underlying concepts and is prepared to correctly apply ANSYS Mechanical to other problems.

The wiki format makes it easy for users to collaborate on developing and maintaining curricular materials using ANSYS technology. Users can request an account to upload related materials, such as homework problems and PowerPoint® presentations, to share with the community. Cornell is developing a template that contributors can use to create new tutorials for the site. The template will ensure that tutorials follow a similar structure and utilize sound pedagogical practices even though they are developed by different authors.

One of the greatest challenges facing the Cornell team involved in the design of the Cerro Chajnantor Atacama Telescope (CCAT) was to understand and minimize the reflecting surface error caused by weight and thermal loading of this large structure, tens of meters in each dimension. Detailed ANSYS Mechanical finite element modeling helped to determine the overall error for the telescope.

Cornell researchers have developed a microscale environmental simulation framework to better understand pollutants and emission control devices. User-defined functions in ANSYS Fluent describe the mixing, chemical reactions and particle dynamics.

Cornell researchers have developed a microscale environmental simulation framework to better understand pollutants and emission control devices. User-defined functions in ANSYS Fluent describe the mixing, chemical reactions and particle dynamics.
The university takes great pride in student teams as they undertake very challenging design projects.

and heat transfer. “It will use ANSYS simulation technology to cut across and adopt a uniform solution approach to multiple traditional disciplines,” Bhaskaran says. “Learning will be through case studies drawn from eight mechanical and aerospace engineering courses at Cornell as well as from engineering practice.”

In 2011, Cornell organized the Integration of Simulation Technology into Engineering Curricula (ISTEC) workshop to bring together major stakeholders in a sustained conversation on scaling up the effective use of advanced simulation in engineering curricula. ISTEC 2011 offered participants the opportunity to step back from individual specialties and courses and think more broadly about enabling a unified approach, one that crosses disparate courses and disciplines in engineering education and the engineering profession. “Participants shared ideas, best practices and learning resources, and they left with a charter to bring about improvement in their organizations,” Bhaskaran says. “The workshop generated palpable excitement about the potential for simulation to improve problem-solving and critical thinking skills of budding engineers.”

Every year, 15 to 20 Cornell student teams engage in competitions that run the gamut from race cars and satellites to autonomous underwater and air vehicles, and many others. The students run the teams themselves and manage the budget, which provides exceptional training not only in engineering skills but also in leadership, management and marketing. Many student teams use ANSYS software to gain a better understanding of their products’ design challenges, evaluate alternative solutions, and iterate to optimized designs.

“We take great pride in our student teams as they undertake very challenging design competitions,” Dean Lance Collins notes. “Our student teams have been blessed with incredible success, and I am continually hearing that ANSYS software gives our students an edge in these projects and design competitions. It’s unusual for a research institution like Cornell to have this level of activity in student project teams because it requires a considerable amount of faculty time, financial resources and space. We commit substantial university resources to these activities because of the enormous value they provide. Most important, the projects inspire our students and enrich their education by giving them the opportunity to learn skills they will use in their careers. A recruiter from Lockheed-Martin told me that hiring a student who has worked on one of these project teams is like hiring someone with five or 10 years of experience.”

Cornell ranked 15th among United States universities and first among New York universities with research expenditures of $671 million in 2009. Cornell researchers are known for breakthroughs such as the invention of multiphoton microscopy, leadership in nanotechnology and nanoscience, creation of the Mars exploration rovers Opportunity and Spirit, and leadership of the world’s largest and highest 25-meter submillimeter wave telescope project. Faculty and student research teams there utilize ANSYS software in a wide range of projects aimed at expanding the frontiers of knowledge in many different disciplines.

The Cornell University College of Engineering was ranked by the U.S. News & World Report as one of the top seven engineering programs in the United States. “ANSYS software makes a major contribution to the engineering program at Cornell by providing tools used by students and teachers in the classroom, project teams and researchers to solve challenging mechanical and fluid flow problems,” Bhaskaran states. “We are in the process of acquiring two new Dell PowerEdge R820 servers to increase parallel computing power available for solving problems with ANSYS software, particularly for 3-D CFD calculations.”

**ADDITIONAL RESOURCES**

READ THE FULL STORY OF ANSYS SOFTWARE USE AT CORNELL
ansys.com/81learning2
For more than 17 years, LEAP Australia — Leading Engineering Application Providers — has helped universities across Australia and New Zealand to leverage the benefits of ANSYS software for advanced research and educational programs. Every year, in classrooms and labs across the continent, thousands of future engineers first encounter the power of simulation software using ANSYS tools.

As they perform academic exercises or support faculty research projects, these students prepare for future careers in which they’ll use ANSYS solutions to verify product and system performance in a virtual testing environment. Given the broad acceptance of simulation tools by Australian industry and government research teams alike, simulation skills using ANSYS software have become a prerequisite for students at graduate and undergraduate levels.

As a long-standing ANSYS channel partner, LEAP has helped every leading university in Australia and New Zealand to specify, install and support their simulation suites. LEAP’s experience shows that universities typically encounter some challenges in leveraging the full scope and power of the software. Often, individual research groups lack the generous budgets of industry customers or university IT departments — as well as the sophisticated procurement processes that enable them to consolidate licenses and take advantage of volume discounts.

In addition, individual university users sometimes are restricted by the narrow research focus of their academic department or research group, which means that they have access to a limited number of single-physics licenses. But increasingly, these researchers need to conduct multidisciplinary studies, for which multiple ANSYS software solutions provide great benefit.
As an example, at the University of New South Wales (UNSW) in Sydney, from an initial rollout of ANSYS single-physics tools within the School of Mechanical and Manufacturing Engineering, the full suite of ANSYS tools is now available for both teaching and research across all other engineering schools within UNSW.

**LICENSE BUNDLING: A PRACTICAL, COST-EFFECTIVE SOLUTION**

LEAP has partnered with many leading universities Down Under to help them overcome such obstacles and create the ideal academic environment for broader adoption of engineering simulation — one in which software is scaled for easy, cost-effective application by a wide range of users, across multiple faculties, students and researchers. Working closely with ANSYS, LEAP helps universities to benefit from academic license-bundling agreements, creating an environment in which more users across all departments can capitalize on the full multiphysics capabilities of the ANSYS suite.

In these bundling agreements — also known as campus-wide licenses — access to ANSYS software is centrally procured by a single group at a university, typically the IT department. By replacing individual department licenses with a broad, multi-user license, most universities are able to increase their license capacity and access to all necessary physics, while leveraging the cost advantages of a volume purchase. At Monash University, Scott Wordley found that “by making a large, multi-year commitment to a campus-wide research license and HPC bundle, we simplified the procurement process and increased the resources available to all our researchers.”

**Increasingly, researchers need to conduct multidisciplinary studies.**
For ANSYS users themselves, the most obvious benefit is the ability to explore and access the full capabilities made available by the bundling agreement. For instance, civil engineering faculty and students who previously used only structural analysis software can apply CFD capabilities as well as other ANSYS tools to study the effects of multiple physical forces on their designs. This worldwide trend for professional engineering teams to become more interdisciplinary means that access to multiphysics solutions from ANSYS helps to better prepare students for the workforce; it has been shown to deliver better graduate employment outcomes. At RMIT University, Professor Jiyuan Tu reports that broad exposure to ANSYS products enabled by campus-wide licensing is viewed as extremely valuable by alumni who work as professional engineers: “The feedback from our graduates who have been working in industry — and are using ANSYS software now in their careers — has been extremely positive.”

From a user perspective, campus-wide licensing makes it easier and faster to access ANSYS software from multiple locations across a university’s network of labs and classrooms. It once resided on individual CPUs or clusters; now ANSYS software is increasingly hosted on a centralized server to leverage advanced high-performance computing (HPC) capabilities.

Taking Full Advantage of Bundling

Making the jump from individual licenses to a campus-wide bundle can represent a major transition for some universities — yet the benefits are well worth it. In Australia, LEAP collaborates with the ANSYS academic team to support customers in easing this transition. Based on this experience, any university considering a campus-wide license should deliberate over these key elements as part of the license-bundling process:

- **Software sizing and specification**: Which ANSYS software solutions are needed by various engineering departments and research teams? What is the right mix of teaching and research licenses based on maximum class sizes and concurrent research needs? For each academic customer, ANSYS software can be configured in the highest-impact manner to meet both research and teaching objectives.

- **Broad user training**: Campus-wide licensing puts the software into the hands of thousands of students, faculty members and researchers — but are they prepared to capitalize on its full capabilities? To maximize results for each university, ANSYS can deliver a training program that meets that institution’s specific needs.

- **Appropriate HPC specification**: Multiphysics and systems-level simulations can have large computational requirements, and university IT departments must be prepared to support the increased number-crunching that goes hand in hand with campus-wide agreements. ANSYS and its channel partners work with each university to provide sufficient ANSYS HPC licensing and to verify that the technology infrastructure is up to speed. Because increasingly complex multiphysics simulations require HPC to run larger models and provide

---

**A Healthy Outlook at RMIT University**

RMIT University in Melbourne has benefited greatly from its campus-wide ANSYS license. Professor Jiyuan Tu, deputy head of research and innovation, says, “Not only can we run a large class, a CFD lab or a tutorial across multiple campuses to support our teaching, but it has also made ANSYS more accessible to all researchers.”

One researcher benefiting is Dr. Kiao Inthavong, who studies how inhaled airborne particles affect long-term health. “When environmental toxicity is involved, it’s not viable to conduct experimentation with human subjects. Our team uses ANSYS to perform integrated CFD simulations accounting for room ventilation combined with facial features and a detailed nasal-trachea airway to better understand links between respiratory health problems and the inhalability, deposition patterns and pathological effects of airborne particles.”

Tu notes that ANSYS is also increasingly used in undergraduate research projects, as well as in post-graduate research. In addition, it has positively impacted the quality of hands-on learning in large undergraduate classes.

According to Professor Tu, RMIT alumni are grateful for their exposure to ANSYS as students, with student feedback confirming that ANSYS makes it easy for them to engage in and become enthusiastic about very complex topics.

---

**At RMIT University, Dr. Kiao Inthavong uses ANSYS Fluent to understand how airborne particles are inhaled — without physical risk to human subjects.**
The Formula SAE (FSAE) team from the Department of Mechanical and Aerospace Engineering at Monash University in Melbourne has relied on ANSYS software for more than a decade to optimize the design of its award-winning cars. Monash Motorsport comprises more than 70 undergraduates with a diverse range of backgrounds and engineering skill levels.

The Monash Motorsport team won their fifth consecutive Australasian FSAE Championship in 2013, with a strong performance on the track and in all static events. They are looking forward to competing with their 2013 car at Formula Student UK and Germany in 2014, and hoping to improve upon their 2012 third- and fourth-place finishes in these events.

To help new members learn to use ANSYS software, lecturer and FSAE team supervisor Scott Wordley has worked with LEAP to develop tutorials tailored for FEA and CFD analyses of common FSAE applications. While Formula SAE is highly competitive, this material is shared with other local teams to allow all students to harness the power of simulation.

Yet another benefit of the Monash ANSYS relationship is the flexibility of campus-wide software licenses. “Our ANSYS license bundle has really benefited researchers at Monash University. First, we save money by bringing users from different departments together to increase our purchasing power and maximize value,” Wordley explains. “Second, today our access is much more consistent and universal, instead of siloed. With our increasing reliance on high-performance computing, it makes sense to share both software and HPC clusters. Finally, we have eliminated needless paperwork and red tape that interfered with our researchers’ already busy schedules. By making a large, multi-year commitment to a campus-wide research license and HPC bundle, we simplified the process and increased the resources available to all our researchers.

Undergraduates benefit from bundling too, as we have enough academic advanced licenses to run ANSYS in all our engineering and science computer labs. We even have students install the software on their laptops and home computers. The tools are available, students know how to use them, and they see the value in the results they provide. And our students actually enjoy using ANSYS!”

Student engineers on the Formula SAE team at Monash University use ANSYS CFD-Post to visualize pressure contours and surface streamlines of their car design. Here, yellow and red areas denote pressures above static pressure, and blue areas indicate pressures below static pressure. Surface streamlines help denote flow separation and re-attachment lines, as well as vortex activity impinging on vehicle surfaces.

Accelerating Results at Monash University

Making the Choice to Bundle

One of the most common oversights university administrators make is overlooking the potential to bundle their ANSYS licenses. Many colleges mistakenly believe that they don’t have enough users, don’t need multiple ANSYS products, or simply cannot afford a campus-wide license.

In truth, campus-wide licensing is surprisingly cost-effective when compared to individual licenses; it is becoming increasingly critical in delivering on the ambitious research and teaching goals of many universities. As educational institutions seek to address huge, complex problems, such as energy efficiency or smart systems, faculty members and researchers are working as multidisciplinary engineers, since they are increasingly being asked to collaborate across departments — and, in the process, embracing the trend to perform multiphysics and systems-level simulations.

Reference

www.leapaust.com.au

Additional Resources

More about the Use of ANSYS Software at RMIT and Monash
ansys.com/81campus2
An aneurysm is a bulge in the wall of a blood vessel caused by a weakened vessel wall. As the aneurysm grows, the risk of rupture increases, and surgical repair may be recommended. The standard surgical method for treating an abdominal aortic aneurysm involves opening the abdominal cavity and removing the aneurysm. In the minimally invasive approach, called endovascular aneurysm repair (EVAR), the surgeon inserts a catheter into an artery in the groin and threads it to the aneurysm. Then using an X-ray imaging device to see the artery, the medical team inserts a guidewire into the artery and uses it to maneuver a stent graft to the aneurysm. The graft is then deployed inside the aorta and fastened in place to reinforce the weakened section, preventing the artery from rupturing. Compared to the traditional surgical method, EVAR results in a higher short-term success rate, less blood loss and faster recovery.

Planning for EVAR is based on pre-operative 3-D computerized tomography (CT) scans that are used to size and position the stent. With the newer capabilities of computer-assisted medical interventions, the 3-D scans could be overlaid on the 2-D images acquired during the operation to help guide the procedure. A key challenge is that introducing the stiff guidewire causes the artery to straighten and deform. Currently, surgeons estimate the amount of deformation when determining whether to recommend EVAR. However, since each patient is unique, the estimated amount of deformation might vary from actuality.
This is believed to be the first time that FEA has been used on individual patients with the goal of improving surgical outcomes.

This can make surgery more difficult, especially in challenging cases in which calcium accumulates in the artery wall. In other cases, if the patient’s arteries are too calcified, it is simply not possible to navigate the EVAR tools in a too-rigid (calcified) cardiovascular system, and the surgeon may not discover this until the patient is in the operating room. “During an operation, time is critical; a surgeon needs to react quickly to any unexpected situation. If we have more information up front, we can better determine alternative strategies that are safer and more expedient for patient treatment. Simulation gives doctors the luxury of knowing what we will experience during surgery while the patient is still at home so that we are better prepared for surgical treatment,” said Dr. Jean-Philippe Verhoye, full professor and cardiac, thoracic and vascular surgeon at the University of Rennes.

SIMULATING THE ARTERIAL SYSTEM
Researchers at the University of Rennes are addressing this challenge by using finite element analysis (FEA) to simulate the individual patient’s arterial system under the influence of the guidewire. While finite element analysis has been greatly used with non-specific patient data to design endovascular devices and to study the behavior of aneurysms, this is believed to be the first time that FEA has been used on individual patients with the goal of improving surgical outcomes. The pre-operative CT data was analyzed using Therenva EndoSize™ software, which extracts vessel center-
lines, contours and surfaces, and generates 3-D geometry. This geometry was imported into ANSYS DesignModeler software to re-create the complete aortic surface and produce a tetrahedral mesh with between 5,000 and 10,000 shell elements, depending on the individual patient.

A linear elastic model was used to describe the deformation properties of the arterial wall. Mechanical properties were based on the amount of calcification, which, in turn, was estimated based on analysis of the artery wall from the patient’s pre-operative CT images. Young’s modulus values defining elasticity were applied based on literature. Material properties of the guidewire were determined by physical testing. The upper extremity of the abdominal aorta is fixed by the aortic hiatus, while the femoral artery is fixed in the femoral triangle. Between the coeliac aorta and the femoral artery, there are no other strong anatomical structures to fix the arterial system. So the superior extremity of the abdominal aorta and the guidewire insertion site on the femoral artery were assumed to be fixed. Elastic supports were used to model the anatomical relation between the posterior side of the aorta and the anterior side of the spine.

**SIMULATION PROVIDES ACCURATE PREDICTIONS**

Simulations were carried out using the ANSYS Mechanical FEA solver on a workstation with a six-core Xeon® processor. Researchers simulated placing the guidewire onto the centerline of the aortoiliac arterial structure using pre-stress to initialize guidewire interactions. They then removed the pre-stress condition to initiate guidewire–artery contact. The end result was a deformed model that showed how the shape of the artery changed under the influence of the guidewire.

The team tuned model parameters by simulating 10 patients and projecting the

---

**The surgeon can more accurately determine the optimal size and position for the stent graft before surgery.**
simulation results onto the intraoperative images that showed the actual deformation of the arteries. The model parameters were then adjusted independently for each patient to minimize errors. The mean simulation error was 0.8 mm with the model parameters tuned specifically for that patient. All simulation error measurements also include the error associated with registering the simulation onto the intraoperative image.

Based on these results, researchers established rules to set general simulation parameters based on patient data. The simulations were then rerun using the general rules to set model parameters. The mean simulation error was 2.3 +/- 0.6 mm, which is well within acceptable limits.

One patient’s intraoperative data was used to match 3-D and 2-D data at two different incidence angles. The simulated guidewire was projected on two intraoperative images with different angles of incidence. The simulation error for this patient was 3.5 +/- 2.5 mm for the first image and 2.0 +/- 1.3 mm for the second image, which again was within acceptable limits.

Simulation was then applied to a test group of 12 patients. Mean simulation error including registration error was acceptable at 2.9 +/- 0.5 mm. Mean simulation calculation time was approximately 300 seconds. The time for the complete process was 10 minutes for data analysis and extraction, 10 minutes for preparation of simulation, five minutes for simulation, and two minutes for registration. Simulation results correlated well with fluoroscopy intraoperative images under several observation angles. Simulation was demonstrated to provide much more accurate predictions of the artery’s deformed shape than could be achieved by the surgeons using their experience and intuition.

**MORE ELABORATE SIMULATION FRAMEWORK**

University of Rennes researchers and partners are currently working on the use of finite element analysis to study an elaborate simulation framework using a more accurate description of mechanical properties of arteries and endovascular devices. Assuming further developments, this method could be used pre-operatively to support decision-making in terms of navigability, access path, endovascular device choice, therapeutic strategy, aneurysm neck behavior, and evaluation of new devices. Additional evaluation and validation is required, especially in cases with complex anatomical configurations.

This new approach has the potential to provide several major improvements for EVAR. In the pre-operative phase, the surgeon might be able to more accurately determine whether EVAR is possible in difficult cases, such as when the patient’s arteries are calcified; the team might more accurately identify the optimal size and position for the stent graft. During the operative phase, simulation can provide more-accurate 3-D images to guide the intervention. This method has been used as a secondary tool during a number of EVAR surgeries, but a clinical study will be required before it can be used as the primary method.

**ADDITIONAL RESOURCES**

| ![FEA model created in ANSYS DesignModeler](image1) | ![Deformed geometry after simulation of guidewire insertion](image2) | ![Surgeon using simulation data for navigation during surgery](image3) | ![Nonlinear Modeling for Healthcare Applications](image4) |

© 2014 ANSYS, INC.

**ANSYS ADVANTAGE** Volume VIII | Issue 1 | 2014 31
Computational fluid dynamics is used as a virtual wind tunnel to optimize the design of scramjet engines at up to Mach 6.5.

By Dr. V. Babu, Professor, Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai, India

Conventional jet engines use a turbine-driven compressor to compress air prior to combustion of fuel. The exhaust from the combustion drives the turbine and creates thrust from the nozzle to propel the plane. Ramjet engines replace the compressor with a specially shaped duct, open at the front, that uses the forward motion of the aircraft to compress air. No moving parts are required in a ramjet. Fuel is sprayed into the airstream, and the mixture is ignited. Combustion in a ramjet takes place at subsonic speeds, but the exhaust gas is accelerated to supersonic speeds. Ramjet engines can function only at above Mach 1 speeds, so the aircraft must reach this velocity through some other method of propulsion. The turboramjet engine uses a turbojet engine for subsonic and low supersonic flight as well as a ramjet engine for sustained cruise at high supersonic Mach numbers. Turboramjet-powered planes, such as the Concorde supersonic transport and Lockheed SR-71 Blackbird strategic reconnaissance aircraft, operate at up to Mach 3–4. A scramjet, or supersonic combustion ramjet, is similar to a ramjet, but combustion takes place at supersonic speeds. This allows the scramjet to achieve theoretical speeds of up to Mach 24, or 18,000 mph.
Researchers have been working on scramjet technology for over 50 years, but scramjets have achieved powered flight only very recently.

**SCRAMJET DESIGN CHALLENGES**

The scramjet comprises three basic components: Air is compressed and decelerated in the inlet, gaseous or liquid fuel is burned with atmospheric oxygen in the combustor to produce heat, and heated air is accelerated to produce thrust in the nozzle. While scramjets are conceptually simple, producing a working one requires

▲ CFD simulation of scaled intake
A scramjet, or supersonic combustion ramjet, is similar to a ramjet, but combustion takes place at supersonic speeds. 

When air enters the combustor, which is about 1 meter long, it travels at approximately 1.2 kilometers per second, so the fuel must be injected, mixed, ignited and burned within approximately 1 millisecond. The air is moving so fast horizontally that it is difficult to get the fuel to spread in the vertical and lateral directions. Some scramjet development teams are considering employing a gaseous fuel, such as hydrogen, because it will quickly mix with the air and burn. Others are pursuing the use of liquid fuels, such as kerosene, that are denser and require smaller fuel tanks. Liquid fuel must vaporize before it can burn, adding another time factor that increases the combustion challenge. The higher the Mach number at the combustor inlet, the more heat that can be added and the more power the engine can generate. However, higher Mach numbers also make it more difficult to maintain stable combustion.

Ground testing full-scale combustors is very difficult because of the challenge of mimicking speeds above Mach 5 at an altitude of up to 32.5 kilometers (20 miles). Scramjets are tested in high-enthalpy wind tunnels, and there are only a few that exist in the world. Testing scramjet engines requires on the order of 10 kilograms of air per second. This is normally accomplished through vitiation (removing the oxygen), compressing the air, injecting fuel, and burning it in the air to achieve the temperature and pressures needed for the experiment. This adds complications because vitiated air has different properties than atmospheric air, and extrapolation to flight conditions is difficult. Another problem is that scramjet combustors present a very hostile environment for instrumentation and measurement.

SCRAMJET SIMULATIONS

Researchers at the Indian Institute of Technology (IIT) Madras are working on the hypersonic technology demonstrator vehicle (HSTDV) for the Defence Research Lab. The HSTDV is an unmanned scramjet demonstration aircraft for hypersonic flight (Mach 6.5). Researchers first evaluated the ability of ANSYS Fluent to provide accurate design predictions for the HSTDV scramjet by simulating a scaled-down intake design for which wind tunnel results have been published in open literature. The simulation results captured overcoming extreme technical challenges. When flight speed exceeds Mach 5, the temperature of the air entering the combustor is so high that, if decelerated to subsonic speeds, any heat generated by combustion will result only in the dissociation of air; it will not produce thrust. Therefore, the air is decelerated to a Mach number typically between 2 and 2.5 prior to combustion.

Scaled intake simulation results matched physical experiments. The intake has two components: a ramp on the bottom and a cowl on the top. For the intake shown in this figure, the cowl has been split into a hinged movable portion (front) and a fixed portion (back). The four cases correspond to different orientations of the front part of the cowl.
Simulation can predict important performance metrics.

Simulation of preliminary design of full-scale combustor

The intricate details shown in the physical test results, including impinging shock-induced separation and re-attachment of the boundary layer. Simulation accurately predicted operating conditions in which unstarted flow occurs, meaning that the pressure rise in the combustor is so high that enough air cannot be pushed through the inlet, extinguishing the flame. CFD also accurately predicted the pressure throughout the intake. A similar exercise was carried out for validating predictions of supersonic combustion of different fuels — both liquid and gaseous in model scramjet combustors. With the confidence gained from the validation study, IIT researchers moved to simulating the full-scale intake of the HSTDV. Simulation results showed the shock reflections as the incoming air hit the intake across the entire range of operating conditions, including different angles of attack.

Next, the IIT team used CFD to evaluate different injection schemes to optimize the design of the HSTDV scramjet combustor. The full-scale combustor calculations utilized a 2-million-cell mesh. These models use full 3-D, compressible, turbulent reacting flow and include very fine meshes with a mesh spacing of less than 0.1 mm with gradient-based adaption to fully resolve shocks. To model supersonic combustion of hydrogen, researchers used an eight-species, 37-reaction mechanism as well as short- and even single-step mechanisms. For ethylene fuel, they employed a nine-species, 20-reaction mechanism. For kerosene fuel, a single-step mechanism was used. The models incorporate one-equation Spalart–Allmaras and two-equation SST k-ω turbulence models. These are some of the first simulation results for a full-scale scramjet combustor to be reported in open literature. Because there are so few wind tunnels in the world capable of accommodating a full-scale scramjet combustor, CFD simulation is crucial to optimizing the design. The calculations, starting from scratch, took about four to five months of run time to converge to the desired level.

DESIGN OPTIMIZATION

The team used CFD to simulate a full combustor with five struts, each containing 22 injectors and using V-gutters for flame stabilization. The struts are staggered to map fuel across the entire cross
section of the combustor. The simulation shows liquid droplets injected from the struts; particle tracking (DPM) is used to track their movement. The tracks disappear once the droplets have completely evaporated. CFD aided in evaluating different injection strategies with the goal of evaporating all of the fuel and mixing it as thoroughly as possible within the combustor. Simulation results showed that the initial designs released too much heat too quickly, so the next design candidate moved the struts downstream in the combustor. The best design achieved with simulation was able to load 95 percent of the combustor with kerosene. After the simulation was completed, a prototype of this design was built and tested, and the experimental results matched the simulation predictions within the measurement margin of error.

Simulations have been shown to predict the flow in model combustors quite well for different fuels and injection schemes. Simulation can predict important performance metrics, such as mixing and combustion efficiencies, degree of mixing and total pressure loss. As a result, IIT researchers are using ANSYS Fluent software as a virtual wind tunnel to evaluate preliminary designs and identify a small subset of designs for fabrication and testing while using much more expensive and time-consuming wind tunnel testing primarily to validate CFD results.

**ADDITIONAL RESOURCES**

- [USER-DEFINED FUNCTIONS FOR DISCRETE PHASE MODEL IN ANSYS FLUENT](ansys.com/81run)

![Validation study of full-scale combustor showed good correlation between simulation and experiments.](image-url)
A wireless body area network (WBAN) consists of a network of computing devices worn on the body that communicate with a server over a wireless network. WBANs can interact with sensors that monitor the wearer’s medical status and provide an early warning of health problems (and this is only one application of the technology). Traditionally in WBAN design, circularly polarized (CP) microstrip patch antennas are used, since they can maintain performance in spite of movement by the wearer. However, CP antennas tend to have relatively narrow bandwidth. It is particularly difficult to obtain broad bandwidth with a compact CP antenna.

Researchers at Hanyang University set out to design a compact CP antenna with broad-enough bandwidth to cover the entire instrument, scientific and measurement (ISM) band, which runs from 2.4 GHz to 2.485 GHz. The antenna was designed as a square slot etched into the bottom plane of an FR-4 substrate with a pair of Y strips connected to ground with an inverted L-shaped microstrip feed line. A square slot is etched on the bottom plane of the 36 x 36 x 1.6 mm substrate with a relative permittivity of 4.4. To reduce the dimensions of the antenna, a portion of the inverted L-shaped feed line is meandered or folded over itself to provide the required resonant length but

Researchers designed an unusually compact wearable antenna that covers the entire instrument, scientific and measurement band.

By Kyeol Kwon, Ph.D. Candidate, and Jaehoon Choi, Professor, Department of Electronics & Computer Engineering, Hanyang University, Seoul, Korea

Researchers set out to design a compact CP antenna with a broad bandwidth.
within a smaller area. Other key design parameters include thickness of the feed line and width of each slotted section of the feed line. The research team considered adding strips connected to ground to broaden the 3 dB axial ratio bandwidth. They also considered I-shaped, T-shaped and Y-shaped strips.

Researchers faced the challenge of configuring geometric parameters to optimize the antenna to achieve ideal return loss and axial ratio bandwidths. The return loss bandwidth consists of the frequency range over which the loss of signal power caused by the reflection at a discontinuity in the transmission line — such as mismatch with the terminating load — is below a certain value expressed in decibels. The axial ratio bandwidth is a measure of the quality of the circular polarization of the antenna. A circularly polarized field is made up of two orthogonal E-field components of equal amplitude that are 90 degrees out of phase. The axial ratio is the ratio of these components. The ideal value is 0 dB, and the axial ratio bandwidth is typically quoted as the bandwidth over which orthogonal components differ by no more than 3 dB.

Optimizing the design of the antenna using physical measurements is very time-consuming and expensive due to the high cost and long time involved in building each prototype iteration. So Hanyang researchers used ANSYS HFSS electromagnetic field simulation software to evaluate performance of the proposed antenna using a wide range of geometric parameters (listed below) prior to building hardware. They created basic parametric geometry by drawing it in HFSS and assigning material properties. By utilizing the integral automatic adaptive meshing capability of HFSS, they generated a mesh conformal to the 3-D structure and appropriate for the electromagnetic problem. Researchers assigned boundary conditions and excitations and then set up the analysis and frequency sweep. The final step was to run the simulation and view the results including antenna parameters.

The key parameters whose geometries were varied in the simulation were \( I_t \) (x-axis dimension of feed line), \( w_2 \) (thickness of feed line), \( w_3 \) (width of each slotted section of feed line), and \( w_s \) (distance between verticals on Y-strip), as shown in the diagram. The results for return loss

<table>
<thead>
<tr>
<th>( I_t ) (mm)</th>
<th>( w_2 )</th>
<th>( w_3 )</th>
<th>10 dB Return Loss (GHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>18</td>
<td>18</td>
<td>0</td>
<td>3.12 – 3.7</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>0.5</td>
<td>2.62 – 3.12</td>
</tr>
<tr>
<td>50</td>
<td>1</td>
<td>0.5</td>
<td>2.87 – 3.5</td>
</tr>
<tr>
<td>66</td>
<td>0.5</td>
<td>0.5</td>
<td>2.13 – 3.12</td>
</tr>
</tbody>
</table>

Researchers used ANSYS HFSS electromagnetic field simulation software to evaluate performance of the proposed antenna using a wide range of geometric parameters (listed below) prior to building hardware. They created basic parametric geometry by drawing it in HFSS and assigning material properties. By utilizing the integral automatic adaptive meshing capability of HFSS, they generated a mesh conformal to the 3-D structure and appropriate for the electromagnetic problem. Researchers assigned boundary conditions and excitations and then set up the analysis and frequency sweep. The final step was to run the simulation and view the results including antenna parameters.

<table>
<thead>
<tr>
<th>Strip</th>
<th>3 dB ARBW</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Strips</td>
<td>0</td>
</tr>
<tr>
<td>I-Shaped Strips</td>
<td>2.43 – 2.63</td>
</tr>
<tr>
<td>T-Shaped Strips</td>
<td>2.43 – 2.56</td>
</tr>
<tr>
<td>Y-Shaped Strips</td>
<td>2.15 – 2.8</td>
</tr>
</tbody>
</table>

Researchers used ANSYS HFSS electromagnetic field simulation software to evaluate performance of the proposed antenna using a wide range of geometric parameters (listed below) prior to building hardware. They created basic parametric geometry by drawing it in HFSS and assigning material properties. By utilizing the integral automatic adaptive meshing capability of HFSS, they generated a mesh conformal to the 3-D structure and appropriate for the electromagnetic problem. Researchers assigned boundary conditions and excitations and then set up the analysis and frequency sweep. The final step was to run the simulation and view the results including antenna parameters.

<table>
<thead>
<tr>
<th>( W_s ) (mm)</th>
<th>3 dB ARBW</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>2.3 – 2.7</td>
</tr>
<tr>
<td>3.5</td>
<td>2.2 – 2.8</td>
</tr>
<tr>
<td>4</td>
<td>2.15 – 2.8</td>
</tr>
<tr>
<td>4.5</td>
<td>2.55 – 2.65</td>
</tr>
</tbody>
</table>
They faced the challenge of configuring geometric parameters to optimize the antenna to achieve ideal return loss and axial ratio bandwidths.

<table>
<thead>
<tr>
<th>SIMULATED BY ANSYS HFSS</th>
<th>MEASURED</th>
</tr>
</thead>
</table>

Simulated and measured return loss of proposed antenna vs. frequency for different values of $I_{th}$ show that the resonant frequency of the antenna is reduced as $I_{th}$ increases. Based on these results, the proposed antenna uses Y-shaped strips, as it has the widest axial ratio bandwidth (ARBW): 3 dB. When $w_s$ increases from 3 mm to 4.5 mm, the 3 dB bandwidth becomes wider. However, the enhanced axial ratio performance deteriorates as $w_s$ becomes larger than 4 mm.

The proposed antenna was fabricated using the optimized design parameters determined through simulation: $L$ (internal side length) = 24 mm, $G$ (external side length) = 36 mm, $l_{tv}$ = 19 mm, $w_1$ = 0.5 mm, $w_2$ = 0.5 mm, and $w_s$ = 4 mm. The measured return loss characteristics of the fabricated antenna closely matched the simulation predictions. The antenna has a 10 dB return loss bandwidth of 1,120 MHz (from 2,170 MHz to 3,290 MHz) and a 3 dB ARBW of about 29 percent with respect to the center frequency of 2,450 MHz, which is wide enough to cover the full ISM band. The overall size of the antenna is small: 36 x 36 x 1.6 mm. The new antenna is more compact than similar antennas presented in the past, yet the bandwidth is broader. These features make this new antenna a good candidate for modern WBAN systems that require high performance, small size, low weight and low production costs. This optimized design may well be the catalyst for a wide variety of safe, comfortable body-worn devices for future medical and consumer product applications.

**ADDITIONAL RESOURCES**

ANTENNA DESIGN AND PLATFORM INTEGRATION ANALYSIS WITH HFSS
ansys.com/8live
40+ World-Class Locations

Learn From Your Peers

Network with Leading CAE Professionals

North America • Asia • Europe • Latin America • Africa & Middle East

Visit ANSYS.COM/Conference for more information
Competitions in Education: LEARNING AND MAKING ENGINEERING FUN

Every year, students around the world use simulation in engineering competitions that are not only fun but prepare them for future careers.

By Helen Renshaw, Strategic Alliances Specialist, ANSYS, Inc., and ANSYS Advantage staff

▲ The Blue Sky Solar Racing Team from the University of Toronto participated in the Bridgestone World Solar Challenge in Australia.
Studying can be tedious. And if that study is engineering, after awhile theory, formulae and equations can all start to look alike. To relieve the routine, learn about practical applications of curriculum topics, work with an enthusiastic team and get a jump on skills that can be useful in future careers, thousands of students throughout the world participate in engineering competitions — and have fun doing it.

These student competitions relate to many industries, including aerospace, automotive, civil engineering, electrical, robotics and marine. Global in nature, activities involve designing and building a workable product — but they’re usually much more than that. The competitions often include planning, budgeting, finding sponsors, marketing, teamwork and all the other tasks that are required in the business world. Student teams vie for the top spot based on rules and criteria.

The most pervasive activities globally are held by SAE International for racing formula, off-road and other vehicles. Other events on the long list of competitions include Formula Hybrid, the World Solar Car Challenge and Design/Build/Fly. Sponsorship on event and team levels is provided by industry-leading corporations that look to these competitions to develop future employees skilled in engineering and decision-making.

As in the business world, simulation can be critical to success in these contests. To design and manufacture a car, aircraft, boat or other product on a tight budget and time frame requires simulation-driven product development, which can be accomplished using ANSYS software. Team members do not have the time or funds to build multiple prototypes, so employing engineering simulation along with optimization tools makes it possible to improve reliability and performance at a low cost, within tight deadlines. Students depend on the same best practices as those used in industry: robust design, system simulation and streamlined design processes. ANSYS enthusiastically supports these student competition teams and organizations, having witnessed students grow, graduate and become industry leaders. By providing software, training and support to teams, ANSYS is helping to nurture the next generation of innovative engineers.

**Robust Fluid–Mechanical and Advanced Material Systems Race Car Design**

The Harbin Institute of Technology racing team from China is very familiar with the use of simulation for automotive design. The team has participated in Formula Student Germany and Formula Student Japan; it received second prize for Formula Student China in 2013. ANSYS software has been instrumental in the design of many aspects of the team’s lightweight vehicle, including the unique carbon-fiber monocoque, aerodynamics and the intake system. Of the 58 members on the team, 30 use ANSYS tools to achieve their individual design goals. The Harbin Institute of Technology team employs ANSYS Mechanical, ANSYS Fluent, ANSYS Composite PrepPost, ANSYS DesignXplorer and ANSYS LS-Dyna.

The team solves design challenges by changing design parameters, simulating and optimizing. The students perform parameterization using ANSYS DesignXplorer to create a faster car that is also powerful, lightweight and reliable. ANSYS Composite PrepPost was used to design the composite monocoque car body, fluid–structure interaction employing ANSYS Fluent and ANSYS Mechanical helped ensure performance of the air intake system. Using engineering simulation allowed the team to explore many designs and manufacture only the best one.
Competing in the Baja SAE

One of the most highly recognized collegiate engineering competitions is the Baja SAE, a worldwide off-road design competition. Engineering students are challenged to design and manufacture all-terrain vehicles, based on a defined set of regulations, then compete with other universities from around the world, overcoming grueling off-road dynamic obstacles and a four-hour endurance race. Universidade Federal de Santa Catarina’s Equipe UFSC Baja SAE team from Brazil is one of the leaders in the circuit, competing for over 18 years in regional, national and international competitions.

Equipe UFSC qualified this year to compete at the international level in the Baja SAE World Challenge. The ultra-competitiveness of this event demands much more engineering design (including simulation) than workshop time to develop improved vehicle components. ANSYS simulation software helped the team improve the chassis, suspension links and wheel mounts to optimize the vehicle’s dynamics with a more reliable, lighter design and to continue to be highly competitive, securing 20th place in this year’s competition.

Much of the Baja work focuses on topics beyond what is normally covered in classes, providing an opportunity for the students to solve engineering challenges and improve engineering and design skills that will better prepare them for their professional lives.

To prevent fracture of the trailing arm suspension link for the Baja vehicle, the team performed static structural, linear buckling and modal simulations. After finding regions of high stress near the steel plates, the students simulated the deformation to ensure that fracture would not occur.

In the Baja SAE, engineering students are challenged to design and manufacture all-terrain vehicles, then compete with other universities from around the world in a grueling off-road event and a four-hour endurance race.
The Kyoto Academic Racing Team (KART) took first place in Student Formula Japan in 2013. The competition challenges student teams to plan, design and produce a formula-type car. KART participants improved their automotive knowledge and enhanced technical skills in a practical application that will be valuable in obtaining future employment. The competition is sponsored by almost all the major automotive manufacturers in Japan. KART used ANSYS fluid dynamics software to improve aerodynamic design of a wing for its car. It will continue, with assistance and training from ANSYS channel partner CYBERNET SYSTEMS, to employ CFD in the future to enhance the vehicle’s body shape and analyze vehicle/component part aerodynamics.

During the 2013 Bridgestone World Solar Challenge in Australia, 38 solar-powered vehicles from 22 countries participated in a grueling 3,000-kilometer timed race. Blue Sky Solar Racing from the University of Toronto, Canada, used ANSYS Mechanical for static structural analysis of their solar vehicle’s double wishbone suspension system, roll cage and rims along with ANSYS CFD to design the aerodynamic car body. ANSYS Workbench, CAD interfaces and ANSYS HPC solutions streamlined the analysis workflow.

Performing systems simulation with ANSYS Mechanical software enabled the students to design a light, strong and robust suspension system to withstand loads that the car would encounter during worst-case scenarios. Reliability and safety are the team’s highest priorities. Using ANSYS ICEM CFD and ANSYS CFX, the team designed a very aerodynamic car body using the best transition turbulence model on the market.

Designing, building and racing a solar car is probably the most challenging and rewarding university experience that students can have. The problems encountered are unique and require students to look beyond what they have learned, supplementing established methods with new insight gained from modern computational techniques.

**MORE INFORMATION ON HOW SIMULATION HELPS STUDENT TEAMS SUCCEED**
ansys.com/Bfun

**FORMULA FOR RACING SUCCESS**
ansys.com/Bfun2
Fast Charge

Simulation helped team Fast Charge from Sapienza University of Rome in Italy design an electric car, and 10 of the 20 team members employ ANSYS engineering simulation. They used ANSYS structural simulation to design and optimize the steel wireframe chassis, hub, uprights, brake discs, steering system, transmission pulley, and battery pack container structure. The team employed ANSYS Fluent to optimize the gap between battery pack cells by simulating both 2-D and 3-D thermal-fluid behavior. In addition, students designed the electric motor and inverter cooling using Fluent simulations of the open car. By understanding the aerodynamics, they optimized the radiator dimensions and position.

The team was able to accomplish many major goals. It developed a very light and powerful car with an overall weight of 238 kg — a very good result for a steel wireframe electric car. Teammates enabled good vehicle dynamics by optimizing the chassis stiffness where needed and by optimizing unsprung mass weight. Fast Charge also reduced the battery pack dimensions as well as the battery pack cooling system power consumption.

Electrifying the Competition

Representing Politecnico di Torino in Formula SAE competitions throughout Europe, Squadra Corse from Italy features vehicles bearing the classic Italian rosso corsa racing colors, and they have steadily shown improvements in competition results since launching in 2005. In 2012, Squadra Corse began competing in the full electric category and achieved a seventh-place worldwide ranking at events in the UK, Hungary and Italy. The present competition vehicle is powered by electric motors provided by team sponsors. However, these motors are not ideally suited for SAE racing events because their maximum power output exceeds competition rules, meaning that the car is carrying unused load.

To address this issue, students are working to enhance the motor design in preparation for future competitions. They have added simulation software from ANSYS into their engineering analysis process. Using ANSYS Maxwell, the team has performed electromagnetic simulations on designs for a new surface permanent magnet (SPM) synchronous motor. The team is planning to do structural and fluid analyses in the coming years as it seeks to further its legacy of excellent Formula SAE results.
The engine control system is a critical part of today’s helicopter engines that controls fuel injection and other engine functions. In developing the embedded software that runs the engine control system, Turbomeca switched from manual coding to a model-based design approach that involves the creation of an executable model in a block diagram design environment. Engineers define the functionality of the control system within ANSYS SCADE Suite using blocks that represent algorithms or subsystems. They validate the model and use it to automatically generate embedded code. Turbomeca engineers have developed control systems that are powering the company’s two latest engines using the new method. The new process has demonstrated the ability to substantially reduce errors and development time.

Turbomeca, part of the Safran group, is the world’s leading producer of helicopter engines and has produced 70,000 engines since its founding in 1938. The company specializes in the design, production, sale and support of low- to medium-power gas turbines to power helicopters. Turbomeca turbines propel civil, parapublic (such as police and fire department) and defense helicopters for all the leading helicopter manufacturers. Including its joint programs with other manufacturers, 18,200 of its turbines are now in operation, and its engines have provided 90 million operating hours.

DIGITAL ENGINE CONTROL UNIT
Turbomeca’s engines are organized into engine families based on the level of power output. Each family of engines...
Arriel 2D engine includes several different variants that meet the specific requirements of different types of helicopters. The engines have a modular architecture with the key modules being the compressor, combustion chamber and turbine. The digital engine control unit, called the full-authority digital engine control (FADEC), regulates the engine's speed by modulating fuel flow based on the environment, torque evolution and use case.

The FADEC includes two identical control channels, each of which is capable of independently controlling the engine. The FADEC can transfer control from one channel to the other if a channel is not functioning correctly. Each FADEC platform consists of hardware and an operating system, and it is specified to be compliant with several engine families. Application software is developed for each engine variant to take into account helicopter characteristics and customer needs. There are many commonalities between engine control units of different variants, and a modular architecture has been defined with re-usable components to encourage re-use between different variants.

SOFTWARE DEVELOPMENT PROCESSES
Since 1985, Turbomeca has progressed through four distinct software development processes for developing application software for its FADECs to address improved technological solutions, evolution in airworthiness requirements, and increased software functionality.

In 2005, Turbomeca developed the G4 software process by implementing a development environment that includes requirements management, model-based design, simulation, automation of tests, and qualified code generation in compliance with DO-178B (the standard used to qualify all avionics software by the FAA, EASA and other certification authorities). SCADE Suite is used in the new software process because model-based development is clearer and more understandable than working with source code for systems teams and promotes co-engineering between systems and software teams.

Model simulation provides an efficient method of detecting functional faults at the earliest possible moment. SCADE Suite delivers an efficient model checker that enables engineers to detect problems early in the design phase rather than later in the integration phase. Test cases can be run in the PC environment rather than in the much more expensive and complicated target hardware environment that is deployed on the aircraft. SCADE Suite incorporates a reusable symbol library that promotes re-use and commonality of design within and across software projects.

SCADE Suite helps reduce development duration and cost by enabling efficient codesign engineering between control law and software teams through the use of formal language and methods that are clearer and more understandable than source code and by reducing the code integration phase. Defects are checked at the design level so that they are detected at the earliest possible stage. Using a qualified code generator, SCADE KCG guarantees compliance between the design model and the code, and strongly reduces formal verification at code level. Consistency of modules integration is verified at the model level before generating the C code, eliminating the need for integration verification at the code level. The code generator is qualified as a DO-178B development tool (cf. section 12.2 of DO-178B), so the conformance of the code to the input model is trusted, eliminating the need for verification activities related to the coding phase.
The development team achieved a decrease of 50 percent in the number of open problems on certified versions.

**CODE SHARING AND RE-USE**

The G4 process incorporates a generic modular software architecture based on configurable functions that can be easily re-used for multiple software development projects. It has enabled teams to focus on activities for which they have specific skills and easily share data with other teams working on other parts of the project. The application software developed by the G4 process is independent from the target hardware platform, which reduces the time and expense associated with changing hardware platforms.

The G4 process was first used in the development of the FADEC for the Arriel 2D engine, which is designed for light, single-engine helicopters in the 5,000-pound weight class and currently powers Eurocopter AS350 B3e and EC130T2 helicopters. The Arriel family of engines is used on more than 60 percent of all helicopters in the world in the 700-to-900 shaft horsepower (shp) power class. The dual-channel FADEC provides engine state monitoring, including fuel and oil filter clogging. It also regulates gas generator speed and power turbine inlet temperature for better power optimization and increases mean time between unscheduled removal. The time between overhaul at initial entry into service is 4,000 hours, but the target for engines in service is 6,000 hours.

Turbomeca engineers first used the G4 process in developing embedded software for the Arriel 2D engine that was certified in 2011 and the subsequent Arriel 2E that was certified in 2012. The development team achieved a decrease of 50 percent in the number of open problems on certified versions by detecting problems earlier in the development process and correcting them prior to certification. Development time also was reduced by 30 percent by taking advantage of G4 process improvements, including SCADE Suite. FADEC software for five more engines is currently under development at Turbomeca using the G4 process. The company is also evaluating the latest version of SCADE Suite for future projects because of its potential to bring improvements in compute time and language capability.

**ADDITIONAL RESOURCES**

REDUCING PRODUCT DEVELOPMENT RISK AND COMPLEXITY WITH MODEL-BASED SYSTEMS ENGINEERING AND EMBEDDED SYSTEMS

ansys.com/81code
SCADE model for control function that determines whether electricity produced by engine is used to power accessories.
The ANSYS Application Customization Toolkit is put to use to more accurately size welds by taking existing standards into account.

By Rod Scholl, Principal Analyst, Epsilon FEA LLC, Minneapolis, U.S.A.

The sizing of welds is often controlled by standards, such as those developed by the American Institute of Steel Construction (AISC) and the American Welding Society (AWS). These standards were written with the expectation that hand calculations would be used to determine generalized loads for structures with relatively simple geometries. While finite element analysis (FEA) more accurately simulates complex geometries, it normally requires a laborious manual process to transform localized stresses (as determined by FEA) into the generalized loads needed to size welds that meet standards. By creating an extension using the ANSYS Application Customization Toolkit (ACT), Epsilon FEA has made it easier for the company’s engineers to automate the process. The improved accuracy this approach offers provides significant cost savings while ensuring full compliance with relevant codes.

STANDARDS BASED ON HAND CALCULATIONS
Welds today are frequently sized based on decades-old standards founded on the use of free-body diagrams to hand-calculate loads based on forces and moments. Hand calculations have diminished accuracy when predicting statically indeterminate and complex structures. Even a fairly simple structure, such as a gusset with a large hole, can’t be assessed reliably with hand calculations. Judgment calls are typically used in these situations — relying on the standards’ large safety margins. The downside is that for some geometries and loadings, welds are sized considerably larger than necessary, which tends to drive up the manufacturing cost of structures. While structural analysis tools and methods such as FEA have rapidly advanced, the standards have failed to keep pace, in part because the weight and cost of welding are only minor factors in many structures. But for some industries, the labor-intensive nature of the welding process and part volume has sufficiently driven up costs to warrant a more thorough analysis. In many cases, companies have already created an FEA model of the structure as part of the development process, which can be leveraged to minimize additional effort.
Most welds are sized considerably larger than necessary, driving up the manufacturing cost of structures.

FEA WELD SIZING
Around 1950, engineers began using FEA to size welds. Many approaches were developed using different specifications and methods involving meshing the weld bead and relying on localized stresses. Using existing commercial tools for evaluating welds with FEA may require substantial amounts of pre-processing, large computing resources and correlation with extensive test data for prediction of performance. The more recently developed Verity method (Batelle, 2006) accommodates virtually any geometry and provides impressive accuracy in predictive results that enable welds to be sized more accurately than ever before. The cost of this approach can be considerable, given the need to geometrically model each weld bead and continually update the model as the required weld sizes are determined iteratively.

But because these methods approach the problem from completely different directions, the resulting weld sizes will usually not comply to a standard such as those of the AISC.

For cases in which a standard based on hand calculations must be met and when an FEA model is either cost-justified or already available, the analyst must convert the very localized FEA results to the load-based approach of the standard.

EXTENSION AUTOMATES STRESS AVERAGING
Converting the localized FEA results to a form applicable to a given standard is a manual process that becomes costly and error-prone (even tedious) on an assembly with many welds. ANSYS ACT offers a powerful object-oriented programming environment that provides the ability to fully access and manipulate the model, scope to selections, run custom calculations and plot contour results. ACT also can encapsulate legacy ANSYS Parametric Design Language (APDL) scripts to leverage the investment in legacy code.

Epsilon developed an extension that leverages ACT to automate the process of sizing welds using ANSYS Mechanical in accordance with existing standards. The first step is that the engineer selects the surfaces to be welded and invokes the ACT extension. The extension prompts the user to enter a few items into the details window, such as weld type and filler material strength. Then the engineer plots the...
required weld size per the specified standard. Different calculations are performed for each specified weld type, such as double fillet or partial joint penetration (PJP).

For further investigation, the user examines the weld stresses in local coordinates to plot tensile and shear loads. Behind the scenes, the extension calls a 150-line APDL script that determines the normal thickness of parent materials at each shared node. The extension’s flexibility accommodates curved welds and varying material thicknesses along the joint. By querying normal and shear stresses on both parent materials on the front and back of solids, engineers calculate average stresses and input loads. These results can be plotted or used in calculating required weld size using a known standard, such as that from AISC or a custom criterion per internal design practices. Finally, the node-by-node results are listed in an ASCII text for reference or use in an external weld evaluation tool.

To meet a particular customer methodology, another APDL macro script with 100 lines converted the localized stresses to generalized ones by averaging the localized stresses along a welded length four times the thickness of the parent material to account for the ductility. These generalized stresses were then used to determine the minimum weld size in another APDL script with 25 lines based on the AISC method (Chapter J, Table J2.5). The object-oriented scripting within ANSYS Workbench combines with legacy APDL code into a seamless tool that is operated through the user interface and transferable as a single add-on extension. This successful automation and codification of a design process demonstrates the future path for advanced customization and automation within the Workbench environment.

**POTENTIAL COST SAVINGS**

Epsilon FEA’s weld ACT extension can provide significant welding cost reductions while still delivering complete compliance with relevant standards, by identifying regions where smaller fillets or PJP welds are acceptable. In the example, a wind turbine tower was analyzed using structural simulation for modal, vibration and seismic conditions. The geometry consisted of tapered cylindrical sections. The wind load varied in a nonlinear manner from the top to bottom of the structure; wind load
This application is a good example of how using ACT extensions can save money through more accurate weld sizing.

ACT Improves Productivity Through Customization

Also varied circumferentially due to drag. It would have been very difficult to determine the varying weld size requirement using hand calculations. Relying on conventional hand calculations for the 2-inch-thick material likely would have resulted in complete joint penetration (CJP) welds being used throughout the structure. The FEA results, however, showed that the top third of the structure can be manufactured with PJP welds while still complying with the standard, saving significant assembly cost for each tower. This application is a good example of how the ACT extension approach can save money through more-accurate weld sizing.

Many product development organizations improve productivity by customizing engineering simulation tools to local workflow requirements. Customizing can take the form of introducing unique capabilities to enhance what’s already available within the base product — capabilities that are uniquely different from a simple automation of existing features and functionality. Through customization, engineering simulation activity becomes integral to a firm’s product development process, resulting in improved overall productivity for the engineering staff. At the same time, customization provides the opportunity to involve a broad cross section of the product creation team in engineering simulation activities.

ANSYS launched the Application Customization Toolkit (ACT) as part of the new ANSYS Customization Suite. ACT introduces new customization capabilities for the ANSYS Mechanical environment, allowing users to:

- Encapsulate APDL scripts cleanly through GUI buttons and menus
- Introduce new loads and boundary conditions
- Create custom results
- Integrate third-party tools

An introductory course, Introduction to Application Customization Toolkit, is available for download by ANSYS customers in the Knowledge Resources area of the ANSYS Customer Portal. In the Extensions Library at the Customer Portal, ACT extensions can be downloaded to facilitate training and improve the performance of ANSYS Mechanical for various tasks, including vibro-acoustics simulation.

— Shane Moeykens, Democratization of Simulation Program Manager, ANSYS, Inc.
The key to developing reliable electric drives is creating a best-practice workflow that incorporates model-based systems engineering and embedded software development.

By ANSYS Advantage Staff

Any best practice must manage the complexities of the product development process and verify designs long before system integration and physical testing. Products that incorporate embedded software add to that complexity with their higher level of functionality and requirements — such as interactive displays and complex controls applications. The interdependent subsystems that result can benefit from model-based systems engineering solutions. Furthermore, incorporating hardware behavior (the simulated model of hardware components within the system, called the plant model) during software simulation (for fine-tuning embedded code) requires integrated multiphysics/software virtual analysis.

In designing control systems, one best practice is model-based systems and embedded software engineering workflow that relies on the ANSYS suite. The specific collection of tools — ANSYS SCADE products, ANSYS Simplorer and appropriate physics simulation software — is designed to address the needs of various engineering groups, including system architects, design engineers and validation engineers. For example, electric drive design can benefit from this model-based systems engineering and embedded software development workflow.

ELECTRIC DRIVE BEST PRACTICES

Electric drives are a key component in the power conversion chain between power sources (the grid or battery, for example) and loads (such as industrial equipment or traction motors for cars or trains). As the price of energy increases, developing reliable, efficient drives becomes more critical. A systems approach is mandatory in creating new generations of drives, as it gives designers a global view of their designs early in the development cycle.

In the traditional development lifecycle, each discipline, system, physical component, and software control code has an independent flow. Power electronics designers focus only on semiconductors, electrical performance, EMC/EMI effects and efficiency. Software developers concentrate solely on code validation with elementary and non-realistic test cases, while mechanical and thermal engineers use approximate values as load cases and boundary conditions. Because of the silo-like workflow, no system validation is even possible until a prototype is built.

Using an ANSYS-based process for development, each design team working on the electric drive maintains standard workflows and design tools. Accuracy and investment in existing tools are not sacrificed to work within a common simulation environment. The output of system simulation benefits each discipline: Power electronics designers have realistic temperature predictions, system architects determine optimal system topology, and design engineers optimize performance.
operating points, software developers can test codes with realistic hardware models early in the design cycle, and the same control models can be implemented in the electric drive. As a result, early validation is possible as systems engineers incorporate simulation models up front in the design process.

**REQUIREMENTS MANAGEMENT AND FUNCTIONAL/ARCHITECTURE DESIGN**

Two ANSYS product families form the foundation of this best practice: SCADE and Simplorer. A number of SCADE tools play specific roles in accurately modeling and simulating the behavior of embedded software code; Simplorer’s powerful technology enables analysis of all aspects of large-scale systems — in this electric drive case, electrical–electronics systems and cosimulating embedded code.

ANSYS SCADE System is a SysML-based systems design and modeling tool that has been developed specifically for use on critical systems with high dependability requirements. It provides full support of industrial systems engineering processes, such as ARP 4754A for aerospace, ISO 26262 for automotive and EN 50126 for rail.

To create the system and product requirements of the electric drive, systems engineers use SCADE System to model the functional and architectural design processes. The software uses block diagrams to represent system components and connect them through ports and connectors. Components can be either physical, such as transistors, or software, like controls for a motor.

With SCADE System, systems engineers support and structure requirements management, as well as functional and architecture design processes, to model the system requirements of the electric drive. Systems engineers start with product requirements, creating functional and architectural descriptions of the system. SCADE System software uses block diagrams to represent system components and connect them through ports and connectors.

Once the system functional decomposition is available, the next step is to produce a system architectural design of the electric drive, implementing the functions in terms of physical and software blocks. This must include an explanation of how the functions and data of the initial system have been allocated to the architecture.

While the electric drive system is being designed, the systems engineer must establish full traceability between initial system requirements and the functional and architectural designs. This can be done with the SCADE LifeCycle Requirements Management Gateway. This tool allows the user to graphically manage links between SCADE System and SCADE Suite models and other structured documents — in particular, high-level requirements and test plans.

To manage legacy data, systems engineers can use data dictionaries within SCADE System. Data can be imported from and exported to external databases using Microsoft® Excel®.csv as an intermediate format. Any piece of data can be associated to a block, port or connector in a SCADE System design.

**ELECTROMECHANICAL SYSTEMS DESIGN WITH ANSYS SIMPLORER**

ANSYS Simplorer assists in developing detailed 3-D and simplified 0-D simulation of the physical components. Simplorer is an intuitive, multi-domain, multi-technology tool that enables engineers to simulate complex power electronic and electrically controlled systems. Most systems are too complex for complete 3-D simulation, which requires enormous simulation time; using simplified
reduced-order models (ROM) and cosimulation provides a viable solution. Zero-D simulation is achieved using standard simulation languages that implement the fundamental laws of physics, such as VHDL-AMS for electrical–electronics systems and Modelica® for fluid–mechanical systems. In this way, Simplorer’s powerful technology allows engineers to analyze all aspects of large-scale systems, from detailed component analysis to system performance, in a single virtual design environment. With Simplorer, engineers working in the early stages of the design cycle can identify problems that other simulation or build-and-test methods cannot detect, allowing them to maximize product performance and reduce time to market.

MODELING CONTROL
SOFTWARE WITH SCADE SUITE
SCADE Suite is a model-based development environment dedicated to critical embedded software. It integrates critical applications spanning model-based design, simulation, verification, qualifiable/certified code generation, and interoperability with other development tools and platforms, including requirements traceability. By using SCADE System in conjunction with SCADE Suite, system and software engineers can work within the same framework.

With SCADE Suite, embedded software engineers model the embedded software designed to control electrical components of the electric drive. The team uses state machines and data flows to model the logic and control laws. Requirements of the embedded software can be traced through the development process to ensure that they are met. Developers can work together with the systems engineers that leveraged SCADE System to develop the system architecture. This is an important and unique feature of the ANSYS process.

Once software engineers have modeled the software design with SCADE Suite, they automatically generate the C code that will be used within the electric drive. SCADE Suite’s automatic code generator is 100 percent accurate, greatly reducing the time needed for code verification and validation.

COSIMULATING PHYSICAL
COMPONENTS AND
CONTROL SOFTWARE
ANSYS Simplorer is used for the more electrical–electronics systems cosimulating with SCADE Suite. To perform cosimulation of the physical system model designed in Simplorer and control laws designed in SCADE Suite, a .dll of the SCADE generated code that is specifically wrapped for Simplorer can be imported into Simplorer.

INTEGRATION AND VALIDATION
Integration and validation consists of assembling the final system and

The model-based method addresses the challenges associated with developing products that incorporate increasing functionality and complexity.
validating that it obeys the final requirements. Virtual integration based on models is favorable compared to laboratory testing based on prototypes. By using a virtual prototype of the system, changes can be made at any stage of the design process, reducing development effort and cost. This agile approach allows for perfecting the design before developing a physical prototype.

**DESIGN OPTIMIZATION**

Electric drive engineers can use the capabilities within ANSYS Optimetrics and ANSYS DesignXplorer to seamlessly optimize the overall design — for example, to design optimal controls or passenger comfort. Optimization improves the model to optimize performance.

**FULL-SYSTEM VIRTUAL PROTOTYPING**

Companies looking to improve their product lifecycle development processes can benefit from best practices that embrace simulation-driven product development, such as the one described above. Integrated simulation of a number of models — high-fidelity 3-D physics, reduced-order, 0-D and accurate control software — provides an extremely effective solution.

The ANSYS systems strategy combines embedded software development with power electronics hardware design. The SCADE Suite model that is used at the simulation level is the same one that will be used to develop embedded code that controls the electric drive. The approach is extensible with more-detailed models of hardware components that are typically designed using 3-D simulations.

The model-based method addresses the challenges associated with developing products that incorporate increasing functionality and complexity. Managing design complexity is achieved through model-based system engineering, which reduces overall development costs, improves the optimization process and eliminates system integration failure.

**ADDITIONAL RESOURCES**

![Cosimulation between ANSYS Simploter and SCADE Suite](image1)

![Design optimization with ANSYS Simploter](image2)
Winners demand the best. Ferrari would know. It has one of the best racing records the world over.

Using ANSYS simulation technology, Ferrari is realizing its product promise by optimizing critical aspects of its race cars, like brake cooling systems and full-body aerodynamics, to better handle the twists and turns of the racing world.

Sounds like Ferrari is in the driver’s seat.

Visit ANSYS.COM/Ferrari to learn how simulation software can help you realize your product promise.