ANSYS

ADVANTAGE™

Excellence in Engineering Simulation

VOLUME IX | ISSUE 2 | 2015

SIMULATION FOR THE AUTOMOTIVE INDUSTRY

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

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

Executive & Managing Editor
Chris Reeves

Senior Editor
Tim Palucka

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

Editorial Adviser
Tom Smithyman

Editorial Contributor
ANSYS Customer Excellence
North America

Art Directors
Ron Santillo
Dan Hart

Graphics Contributors
Gregg Weber
Michael Marchant

Design
Agency 1903

Join the simulation conversation
ansys.com/Social@ANSYS

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

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

Subscribe at
ansys.com/magazine

For ANSYS, Inc. sales information, call 1.866.267.9724.
Email the editorial staff at ansys-advantage@ansys.com.

For address changes, contact AdvantageAddressChange@ansys.com.

EDITORIAL

SHIFTING UP TO A BETTER AUTOMOTIVE PARADIGM

The fast lane is getting faster and more complex. Automakers that fall short of innovating within a shorter design cycle will lose the product race.

By Sin Min Yap, Vice President of Marketing, ANSYS

When I joined the Ford Motor Company back in 1997, simulation was far from pervasive in the product development (PD) process: It was used primarily for verification and validation much later in the design cycle. Today, there's practically no system or component that can't be improved via simulation. Even so, PD teams apply simulation only a fraction as much as they could. Automotive companies have reduced or eliminated physical testing significantly, which helps to shorten the cycle, but there is so much more that can be done to support the design trade-offs that engineers must make in creating world-class vehicles.

One of my early development projects at Ford was the F-150 truck. It took about four years from conceptual to design to production phases. Today, car makers have reduced that time by up to 50 percent; some even push for less time. They made these gains by consolidating vehicle platforms, re-using common parts, and deploying product lifecycle management (PLM) and CAD software for efficiency. Yet there's one similarity between my Ford days and today: The industry still performs design trade-offs in manual, suboptimal silo-like conditions, working in a serial manner instead of in parallel.

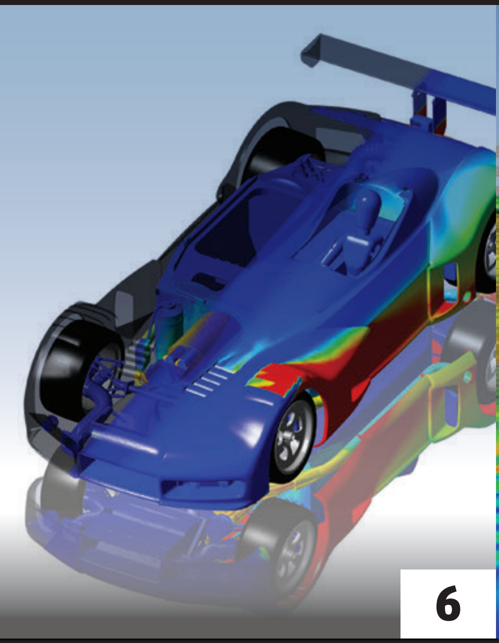
The silos cordon off the different physics/disciplines (mechanical, fluids, electronics and embedded software) as well as work related to the vehicle's many attributes (such as NVH, durability, safety, aerodynamics and electromagnetic compliance). Adding to the complexity, vehicle attributes compete with each other for performance and dollars. Leveraging the power of simulation throughout the PD process breaks down these barriers and clarifies the consequences of trade-offs. It enables more efficient and effective decision making, compressing the design cycle, reducing costs, and producing a more-robust and optimal product — but most important is that it puts dynamic tools within reach of engineers who have not traditionally used simulation in product development processes, like requirements management, concept development, systems design, change management and quality/reliability management.

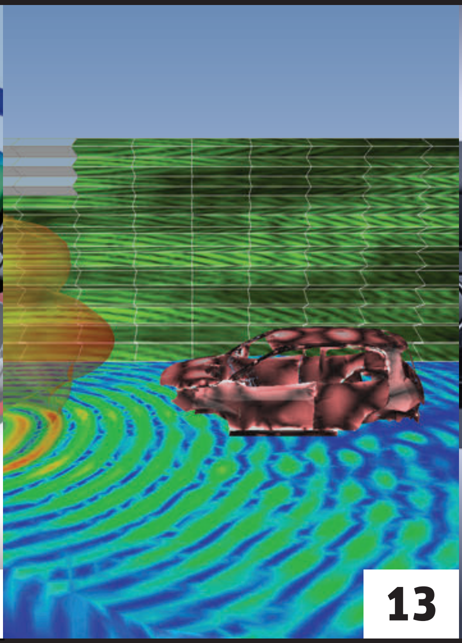
So will any simulation software help automakers innovate as fast as they need to? Yes, but only if the simulation provider offers a comprehensive approach to systems engineering and the ability to simulate complete virtual prototypes. As a result, PD teams can focus on engineering — not on running software — and engineers can collaborate and efficiently evaluate trade-offs.

ANSYS has spent decades building the technology and engaging the right people to achieve this. In fact, it is the only simulation company that offers such a solution. The company's products help automakers (and others) realize their product promise.

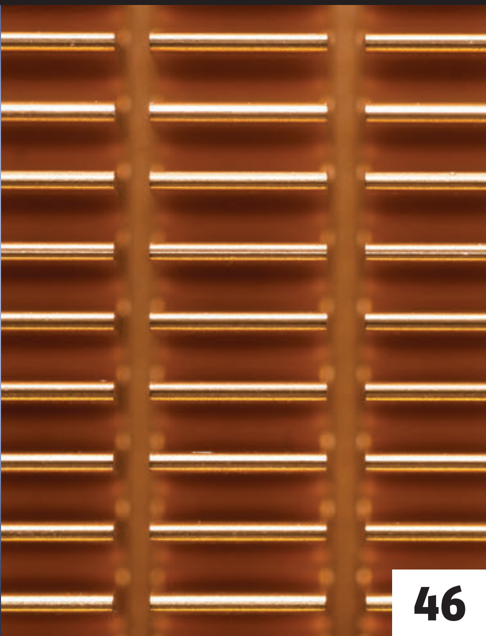
Eighteen years after starting my job at Ford Motor Company, technology and the ways we use it surely have evolved. But the silos remain and act as barriers on the road to real innovation. Today's complex systems and products require solutions that span all physics and disciplines. So let's embrace a paradigm shift and let the ANSYS simulation platform lead the way.

Today's complex systems and products require solutions that span all physics and disciplines.

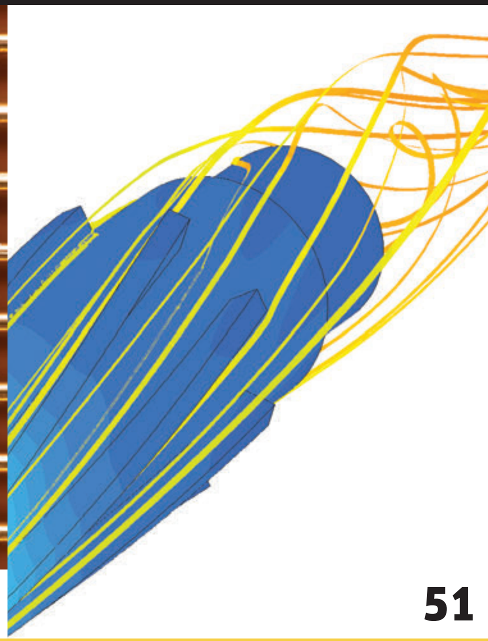




FEATURES



46



51

SIMULATION@WORK

DEPARTMENTS

WEB EXCLUSIVE

To view web exclusives or subscribe to ANSYS Advantage, *visit: ansys.com/magazine*

ABOUT THE COVER

The new LEGO® speed champions sets include the Ferrari 458 Italia GT2 75908 model. The model contains many blocks; this represents the true nature of the race car, which also comprises many interconnected components that must reliably work together to be a winner. *LEGO is a trademark of the LEGO Group, which does not sponsor, authorize or endorse this publication.*

Simulation in the News

LEGO SPEED CHAMPIONS 2015 SETS REVEALED

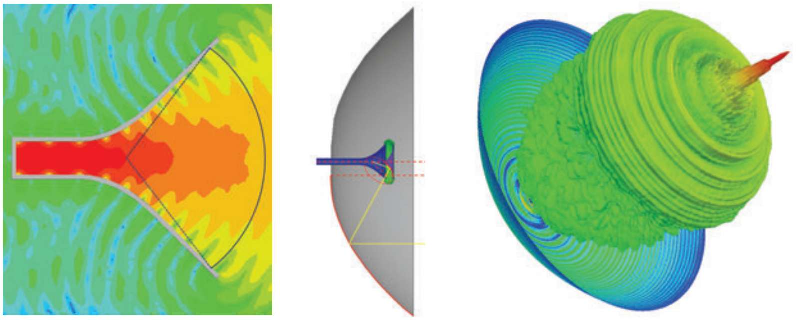
Bricks & Bloks
bricksandbloks.com, February 2015

In February, LEGO® introduced a new theme centered on speed champions including brands like Ferrari, Porsche and McLaren. The Ferrari model sports the ANSYS logo, as ANSYS is a team sponsor. The Ferrari team optimizes critical components of its race cars, like brake-cooling systems and full-body aerodynamics, using ANSYS solutions.

NUMERICAL MODELING OF ANTENNAS

Elektronik Praxis
elektronikpraxis.vogel.de, October 2014

A series of articles examines simulation-based development of advanced antennas, from basic numerical modeling to optimization and system design. ANSYS HFSS, designed for 3-D simulation of high-frequency electromagnetic fields, is used to analyze complex antennas that are difficult or impossible to understand.



▲ Dual reflector antenna simulated with ANSYS HFSS

ANSYS SOLUTIONS MEETING CUSTOMER NEEDS AT TIMKENSTEEL

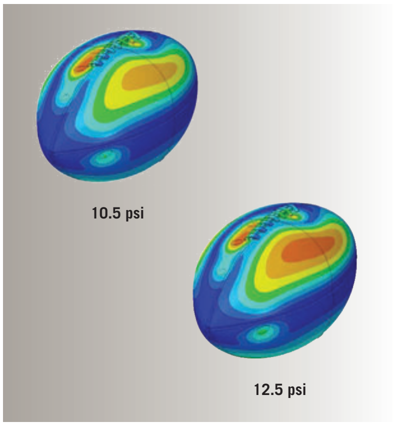
Industry Today
industrytoday.co.uk, February 2015

TimkenSteel, a leading manufacturer of special bar quality (SBQ) large bars and seamless mechanical tubing, uses ANSYS simulation to solve customers' most demanding engineering challenges. The company's experts in materials and applications work closely with customers to deliver solutions tailored to their applications and supply chains. TimkenSteel leverages ANSYS computational fluid dynamics (CFD) solutions to simulate material behavior during the heat treat process, reducing trial and error in the plant to optimize heat-treated steel products.

COMPUTER SIMULATIONS SHOW PATRIOTS' FOOTBALLS PROVIDE A NEGLIGIBLE ADVANTAGE

Sportsgrid
sportsgrid.com, January 2015

Hard science debunked any notions that a football deflated to 10.5 psi — 2 psi less than the NFL minimum — gave quarterback Tom Brady or the Patriots' running game any distinct advantage. *Popular Science* paired up with computer simulation specialist ANSYS to see how a particular grip would be affected by decreased air pressure inside the ball.



ANSYS MODERNIZES CODE FOR INTEL XEON PHI

insideHPC
insidehpc.com, March 2015

Collaboration between ANSYS and Intel® speeds structural code on Intel Xeon Phi™ coprocessors. The partnership ensures that simulation engineers performing structural analysis can expect seamless high-performance computing (HPC) operations with multi-core Xeon® E5 v3 processors and many-core Xeon Phi coprocessors. The result is the ability to run multiple, incredibly large simulations rapidly — which is becoming standard when developing today's innovative products.

ONE ON ONE

Semiconductor Engineering

semiengineering.com, February 2015

The starting point of electronics systems engineering is no longer the system on chip (SoC). *Semiconductor Engineering* talked with Walid Abu-Hadba of ANSYS about systems engineering, compute power and how simulation can be used to analyze a wide range of product behaviors.

"How does the SoC interact with everything around it? If you don't think of it as a system, you're making a mistake. The way we look at it is not from the chip up. It's from the system down. It's a different way of thinking about it."

— Walid Abu-Hadba, Chief Product Officer, ANSYS

DIESEL ENGINE GIANT CHOOSES TO STANDARDIZE ON ANSYS MULTIPHYSICS

Engineering.com

engineering.com, March 2015

Cummins has standardized its simulation software to the ANSYS portfolio, a move that will allow Cummins to innovate products and bring virtual prototyping earlier into the development cycle. For a company known for innovations within control, filtrations, air handling, power generation and aftertreatment systems, standardization of simulation tools should help it share results between disciplines and design teams.

SPACEX LEVERAGES HPC TO REACH ORBIT

HPCwire

www.hpcwire.com, January 2015

Much of SpaceX's future success hinges on its ability to leverage high-performance computing along with computer-aided engineering tools in the drive to reduce the cost of lifting a pound of payload to orbit. The pioneering company employs new computing tools to develop everything from reusable rockets to floating launch pads. SpaceX's primary computer-aided engineering solution is ANSYS software, since the company needs to investigate multiple types of physics when designing rocket engines and spacecraft.

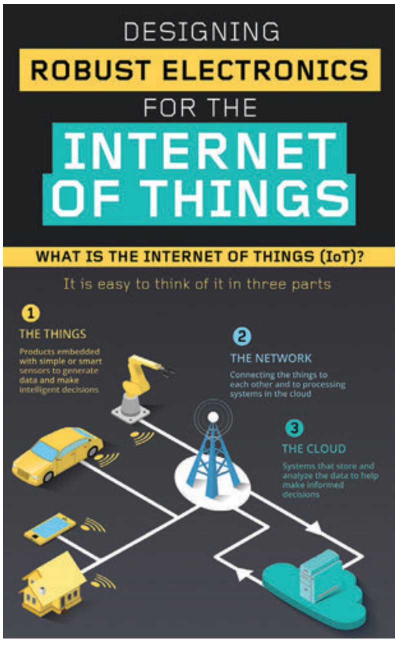
"We use ANSYS solutions to shorten the design analysis cycle, increasing the rate of design evolution while reducing test costs."

— Andy Sadhwani, Senior Propulsion Analyst, SpaceX

DESIGNING ROBUST ELECTRONICS FOR THE INTERNET OF THINGS

ansys.com, March 2015

The Internet of Things (IoT) will greatly streamline communication among our electronic devices, improving the way we live, work and play. In just a few years, more than 26 billion IoT devices will be installed. This means that the high-tech, automotive, healthcare, aerospace, industrial automation and energy sectors face complex design challenges — which can be resolved better, faster and less expensively using engineering simulation solutions from ANSYS.



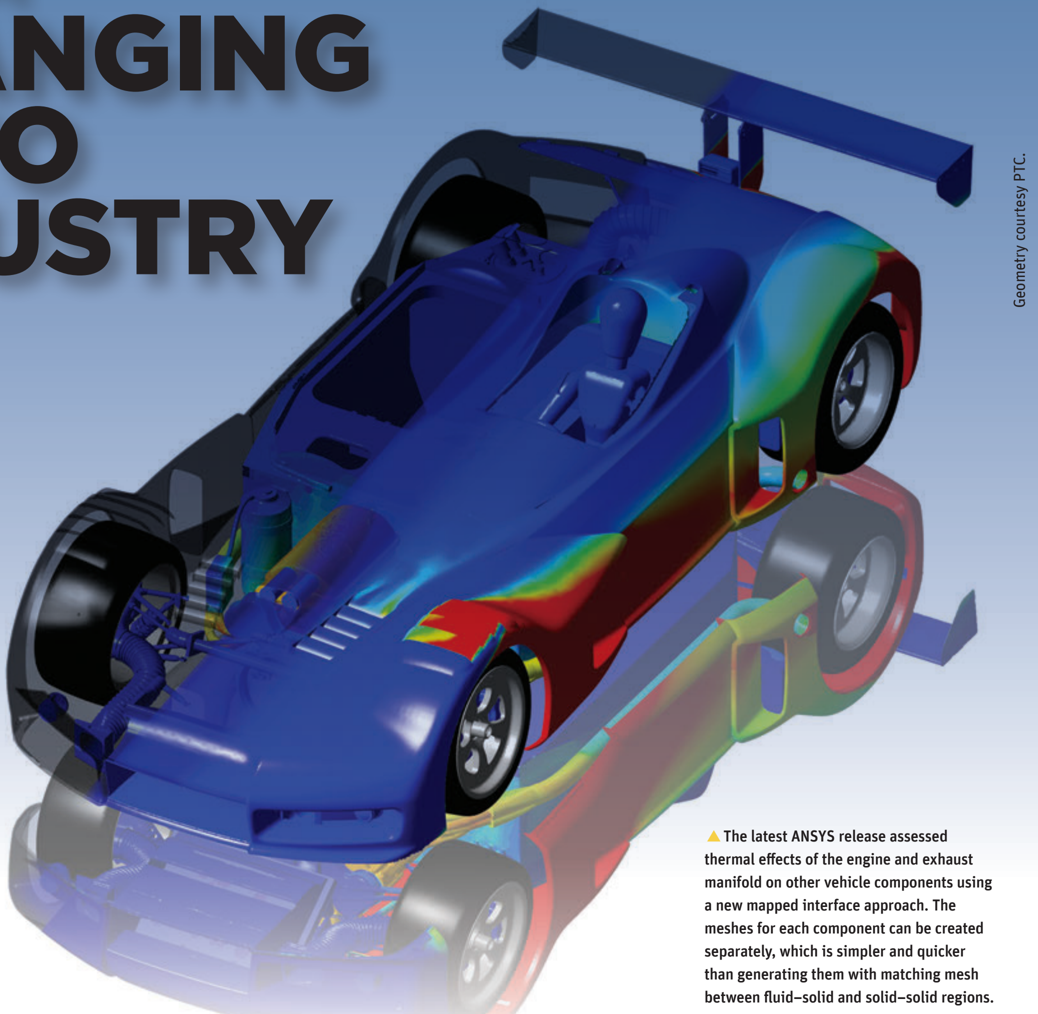
ATMEL SELECTS ANSYS SIMULATION SOLUTIONS TO POWER THE IOT

ELE Times

eletimes.com, March 2015

Atmel Corporation is using engineering simulation solutions from ANSYS to model, analyze and optimize its broad IoT product portfolio, from scalable embedded microcontrollers and microprocessors to wireless connectivity gateways. The software helps enable Atmel to meet stringent power/performance requirements, ensure reliable operations across a wide range of frequencies, and deliver products with tight time-to-market constraints.

A CHANGING SIMULATION PARADIGM *FOR A* CHANGING AUTO INDUSTRY



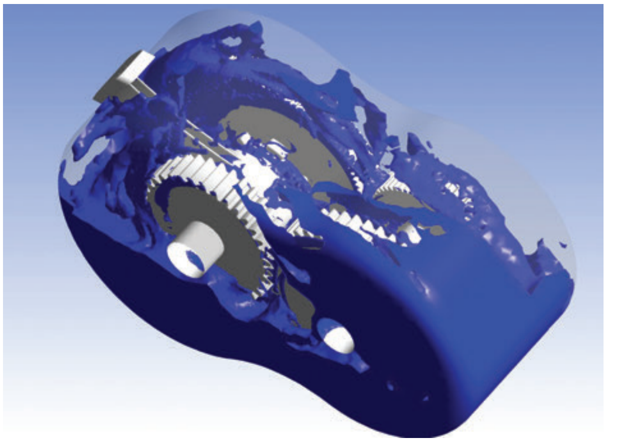
The latest ANSYS release assessed thermal effects of the engine and exhaust manifold on other vehicle components using a new mapped interface approach. The meshes for each component can be created separately, which is simpler and quicker than generating them with matching mesh between fluid–solid and solid–solid regions.

The automotive industry must fully embrace complete virtual prototyping with multidisciplinary simulation and multiphysics — and use it thoughtfully and systematically throughout the product development cycle — to see the real promise of technical innovation. Can vehicle companies shift the paradigm from simulation-in-silos to deploying a common, scalable enterprise-level simulation platform that enables thorough systems engineering?

By **Sandeep Sovani**, Director, Global Automotive Industry, ANSYS

A hundred years ago, Henry Ford promised customers that their car could be painted any color so long as it was black. Today, color is the least of the auto industry's challenges. The car of the 21st century must be fuel-efficient and robust, technologically savvy and affordable, and manufactured quickly on the line without defects. It must meet increasingly stricter government regulations. And the vehicle must incorporate fast-evolving electronic, communication and software technology that hardly existed a few years ago.

Automakers and their supply chain already apply engineering simulation to address some of today's problems: fuel-efficiency standards, potential warranty issues (with reputation and financial consequences), and the transition to hybrid and electric vehicles (H/EVs). They also embrace big ideas — disruptive technology — like self-driving cars. But the key to success is how the industry leverages the big-picture power of simulation to innovate, fulfill consumer demands, comply with stringent regulatory demands, and meet development time, cost and performance targets. Said another way, how effectively are companies deploying a common enterprise-level simulation platform?



Gear box simulation is useful in predicting flow pattern to ensure proper lubrication, computing temperature distribution in fluids and solid components, and predicting viscous losses. ANSYS offers tools that analyze not 20 or 30, but many hundreds of, vehicle shape variants with high-fidelity, detailed simulations.

The key to success is how the industry leverages big-picture power of simulation to satisfy consumer wants, comply with regulatory demands, and meet development time, cost and performance targets.

TODAY'S DRIVING FORCES IN AUTOMOTIVE INNOVATION

Fuel-Efficient, Cleaner Cars

It's no surprise that the industry's overriding trend is fuel efficiency and emissions reduction. Governments worldwide have established fuel-economy/greenhouse-gas emissions standards. By 2025, U.S. CAFE standards will jump to 54.5 mpg, a move backed by 13 major automakers. China and India, the world's largest automotive markets, have followed suit. The global initiative will cut emissions, save oil and take outdated cars off the roads.

Meeting these targets involves exploiting every opportunity for re-engineering. Using simulation sheds insight to quickly resolve a variety of trade-offs, such as reducing aerodynamic drag without sacrificing cooling/cabin quietness, or reducing vehicle weight while still meeting strength/durability specs.

Rising Vehicle Complexity and Disruptive Technology

Consumer appetites, government regulations and advancing technologies are transforming cars from mechanical machines to complex electronic ones. For instance, the door lock apparatus, once a small mechanical device, is now an electronic passive-entry system that comprises electronic sensors, controllers, actuators and advanced software algorithms.

Seven automotive companies have announced plans to sell or market some form of autonomous or self-driving car. Players are investing heavily in this application, achieving major milestones. Internet giant Google logged 125,000 miles in autonomous vehicles in its first year of testing alone, a mere five years ago. While these innovations intrigue consumers, they unnerve auto executives who confront the rising complexity of vehicle engineering — along with its effect on time and cost of development. The chance of an engineer failing to uncover and address potential catastrophes is directly proportionate to a vehicle's design complexity.

How can the industry accomplish such dramatic leaps in product technology in a short time? How can car makers ensure thorough safety of autonomous driving systems? Simulation is the product development tool of choice to tackle these mounting challenges, enabling engineers to model the entire vehicle as a single system. Furthermore, since a company typically employs thousands of engineers around the globe to design various aspects of a vehicle, providing a common collaborative simulation platform is a key enabler. Visionary companies are developing and deploying a common enterprise-level simulation platform — a best practice that delivers rapid testing for hundreds, even thousands, of operating conditions; achieves safe operating modes; and exploits hard-to-see optimization and innovation opportunities.

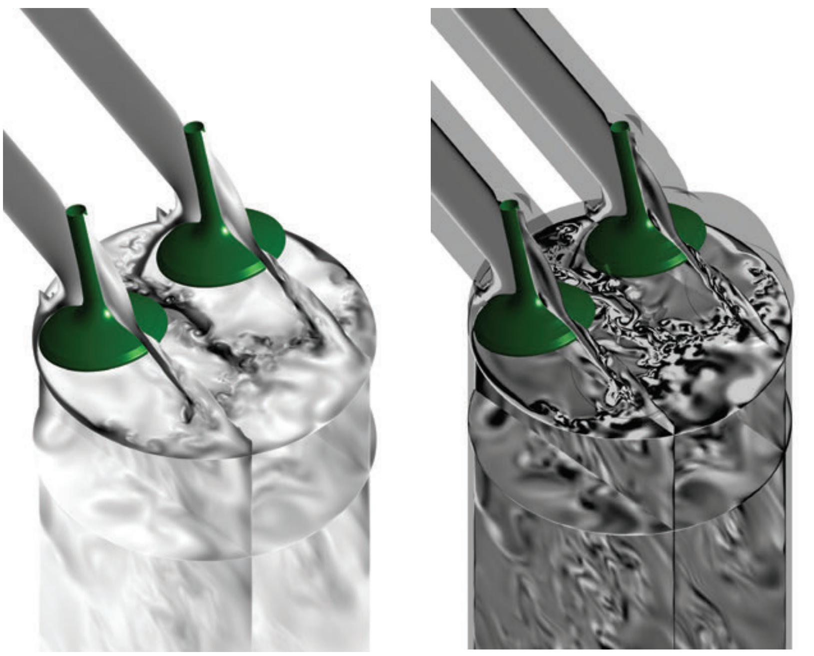
ADDRESSING ENGINEERING CHALLENGES OF INCREASINGLY COMPLEX AUTOMOBILES
ansys.com/92auto1

The Electronics of Things

With smart electronics usage in general on the rise, demand dictates more smart interfaces in vehicles. Infotainment comes in the form of satellite radio, GPS units, and touch-screens built into dashboards and headrests. Automakers now offer in-car 4G LTE Wi-Fi (along with the antennas that make it possible).

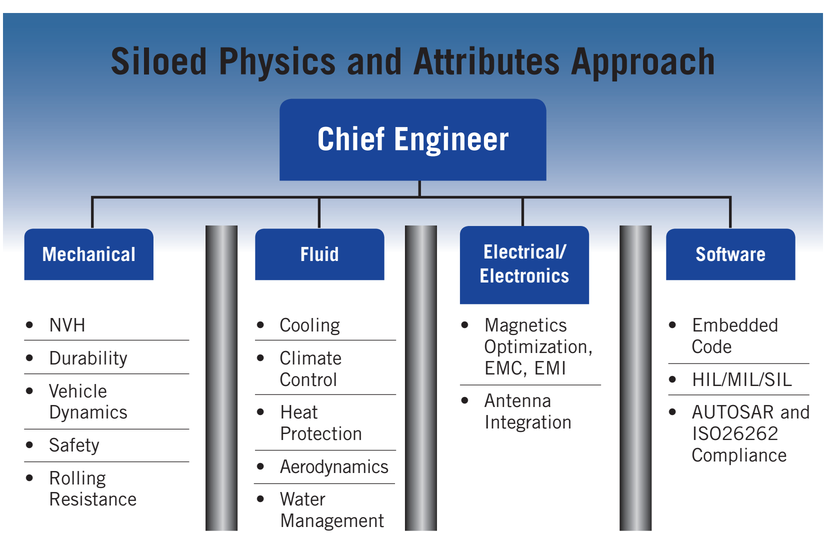
Components like keys and window/door locks are operated by software, as are gauges that read fuel consumption, mileage and emissions. Software controls rear-view cameras, batteries, acceleration and braking functions, to name a few. These systems together create a complicated network of embedded software and electronic signals that grows more complex as technology advances. The modern car is nothing short of a large computer cluster on wheels.

Such Internet of Things (IoT) functionality requires high-fidelity simulation tools for complex tasks — antenna and radar development, EMI–EMC prediction, signal integrity, chip–package system design and electronics cooling — in tandem with embedded code modeling and generation tools. The ANSYS software combination ensures reliable systems-based engineering across hardware and software, in a competitive time frame.



Courtesy Stefan Buhl, Chair of Numerical Thermo-Fluid Dynamics, TU Freiberg, Germany.

▲ Joint research indirectly advances the interests of industry. ANSYS engaged with TU Freiberg to study IC engine flow. The results show how HPC can deliver outstanding results in a time frame that fits into the product launch schedule.



▲ Most automakers focus on single-physics analysis, which lengthens the product development cycle and can result in disastrous consequences as the vehicle's various subsystems interact. Complex systems require best practices that break down the silos and span all physics/engineering disciplines.

STAYING ON TOP OF THE TRENDS: BREAKING DOWN THE SILOS

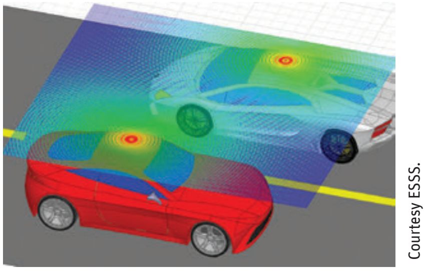
The automotive industry was an initial adopter of engineering simulation; it has leveraged this technology for several decades. The industry realized early on that simulation delivers virtual testing and analysis of an entire vehicle and its parts even before any prototypes are assembled. It's less expensive than physical testing and reveals results in a fraction of the time. Yet today, more than 85 percent of computer-aided engineering performed by automotive companies is single-physics simulation, which studies a sole physical effect in isolation. For example, mechanical strength of a brake rotor is studied separately from air flow that cools the brake. This approach will not recognize potential failure modes when the phenomena intersect — and the consequences of product failure can be disastrous. It also impacts development time, as simulation in various disciplines is performed sequentially instead of in parallel.

As vehicles become electrified, the interplay of multiple physics increases. In an electric traction motor, for example, electrical, magnetic, thermal, fluid, structural and acoustic aspects are all tightly coupled: Coolant flow affects temperature; temperature affects electromagnetics; these, in turn, affect motor efficiency as well as structural vibrations that result in noise. For such systems, making these trade-offs in a silo-like approach is suboptimal.

To make effective trade-offs, the ANSYS simulation suite incorporates coupled solvers, detailed models, submodels, methodologies and best practices across all prominent physics for analyzing key vehicle systems and components. The tools enable car companies to truly optimize designs, performing thousands of what-if, coupled-physics analyses that would be prohibitive via any other means. It serves as an exceptional common enterprise-level simulation platform due to its unparalleled breadth and depth of simulation solutions that are seamlessly hosted in a shared user framework.

SYSTEMATIC DEPLOYMENT OF SIMULATION AT AN R&D CENTER: EVOLUTION OF AUTOMOTIVE SUB-SYSTEM SIMULATION AT VALEO
ansys.com/92auto2

Since automotive products must perform reliably the first time, every time, engineers leverage simulation to address challenges in a risk-free manner.



Courtesy ESSS.

Without simulation, there is no way to properly design a car's antenna so that it is best prepared to deal with Wi-Fi.

SAFE AUTOMOBILE CONTROLS
ansys.com/92auto3

BROADENING DESIGN EXPLORATION AND USING OPTIMIZATION TECHNIQUES

Since automotive products must perform reliably the first time, every time, engineers leverage simulation to address challenges in a risk-free manner. ANSYS tools build in robustness while speeding design time. Parametric design exploration and parameterization enable a thorough view of a wide design space to uncover potential failure modes and quality issues; the methods incorporate design of experiments (DOE) analyses, response surface investigation and input constraint analysis in pursuit of optimal design candidates. Topology optimization takes bulk material out of components, reducing weight without sacrificing strength and durability. The adjoint method, which automatically shows exactly where and how to change a part's shape to improve performance, is especially beneficial in optimizing aerodynamics and duct flows. These methods maximize the ability to sort through thousands of product design concepts by testing for specific parameters, then adopt and refine only the best design candidates. High-performance computing (HPC) and the cloud's space and flexibility of storage ensure that wide design spaces are explored in short time.

LEVERAGING MODEL-BASED SYSTEMS ENGINEERING AND HARDWARE/SOFTWARE COSIMULATION

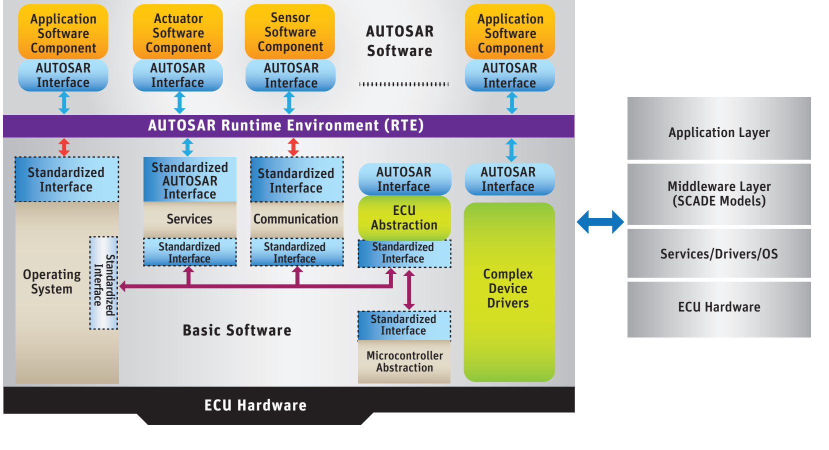
Auto companies must go even further to address complexity, employing simulation that combines high-fidelity component analysis with holistic system behavior models. The focus is on model-based systems engineering (MBSE), in which systems and components are simulated together. The solution includes reduced-order methods (ROMs) that enable effects of a subcomponent to be represented within an assembly without loss of accuracy — while greatly increasing computational speed. This approach accurately captures behavioral characteristics of individual components under real-life operating conditions as part of a bigger system.

Modern cars incorporate dozens of microcomputers that perform control and computational functions, with embedded software programs running them. Just like hardware components, embedded software must be tested for a variety of operating conditions to ensure flawless performance, especially in the case of safety-critical systems such as airbags. To ensure that hardware and software operate flawlessly in unison, they must be cosimulated. ANSYS advancements in MBSE and certified embedded code generation drastically reduce development time for embedded software while meeting the highest safety standards, such as ISO 26262 ASIL-D.

THE NEXT GREAT IDEA

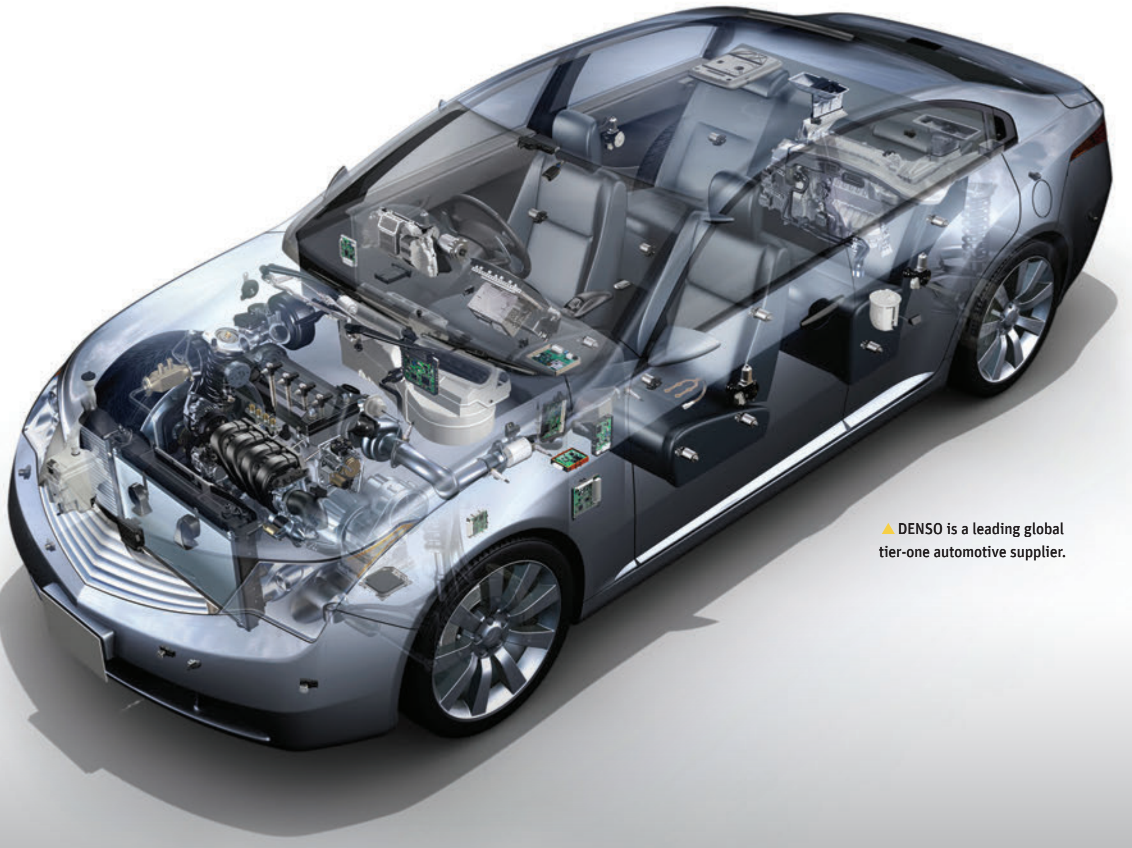
Simulation is not new to cars, but it has become such a crucial element of automotive engineering that visionary companies are making the paradigm shift of deploying a common enterprise-level simulation platform. It helps automakers keep pace with trends in everything from fuel efficiency to self-driving cars. Tomorrow, the trend-setters will look to simulation to set the pace.

This issue of *ANSYS Advantage* details exemplary practices that launch more-reliable products faster. "On Top of the World" (page 10) relates how DENSO engineers use the mechanical suite to expedite product development, cut costs and boost competitiveness across its product portfolio. "In the Loop" (page 22) addresses the role of embedded-code simulation tools in designing vehicle automation and driver assistance concepts. Insights into designing better onboard vehicle electronics appear in "Test Drive for EMI" (page 13). And Tenneco in China utilizes fluid dynamics to address emissions (page 29). Enjoy this automotive edition of *ANSYS Advantage*.



ANSYS SCADE was deployed for the hybrid vehicle management system named electric brain unit (EBU) for a Subaru production vehicle. Software architecture in EBU control software is layered to comply with the AUTOSAR standard.

ON TOP OF THE WORLD



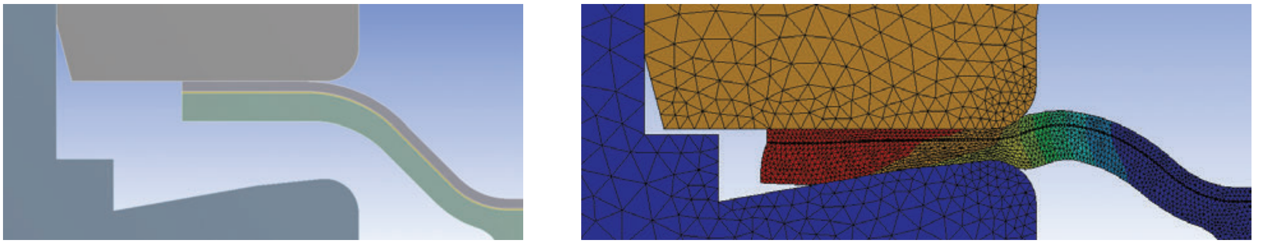
▲ DENSO is a leading global tier-one automotive supplier.

DENSO Corporation standardizes on ANSYS structural software to expedite global product development.

By Shigeru Akaike, Project Director, CAE Design Promotion, DENSO Corporation, Kariya, Japan

Competition is intense in the automotive systems and components business. Best-in-class simulation capabilities are necessary to thrive amid global competition. DENSO Corporation — a leading supplier of advanced automotive technology, systems and components for all the world's major automakers — faced the need to reduce software licensing expenses to remain cost-competitive and to develop world-class products. DENSO performed a rigorous benchmarking process and selected ANSYS simulation software as its standard tool to expedite product development, cut costs

DENSO has developed a strategy to embed CAE fully into all phases of the global product development process.



▲ Using large deformation analysis allows the DENSO team to predict strength and the shape profile. With some compressions at more than 50 percent of the original thickness, structural simulation helps the company to ensure product reliability.

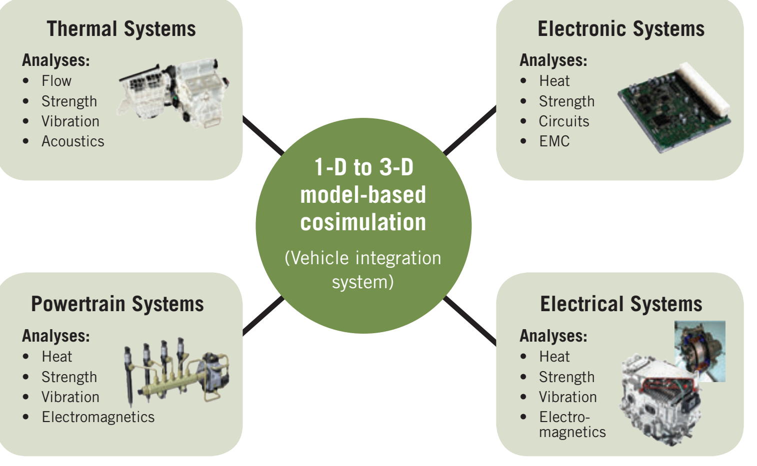
ENTERPRISE-WIDE SOLUTIONS: PRODUCT DESIGN, ANALYSIS AND MANAGEMENT THROUGH ENGINEERING TEAMWORK
ansys.com/92world

and boost competitiveness across its product portfolio, which includes automotive powertrains, advanced electronics, heating and cooling systems, and many other products.

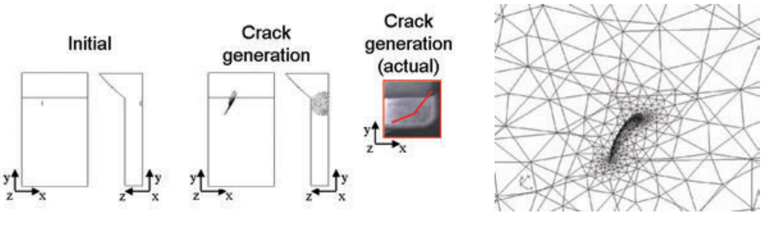
DENSO BACKGROUND

With sales of $39.8 billion U.S. in the latest fiscal year, DENSO is organized into business groups focusing on powertrain controls, electronic systems, thermal systems, and information and safety systems. A small group of DENSO analysts began using computer-aided engineering (CAE) about 30 years ago to diagnose problems that had been revealed during the prototyping process. In the ensuing years, new applications for CAE have been employed, including its use as a presentation tool during the sales process, as an engineering tool to develop new ideas, and as a partial alternative to physical prototyping for evaluating proposed designs. To take full advantage of this technology, the company has added many new CAE professionals over the years: experts responsible for customizing CAE tools for a single physics, specialists who customize tools for multiple physics, and CAE engineers who develop new tools for single and multiple physics.

Over the years, DENSO accumulated licenses for nearly 70 commercial CAE codes and customized many of these codes to meet its special needs. But in 2011, corporate budget cuts made it necessary to reduce software costs. The company made the decision to benchmark its portfolio of CAE codes, comparing codes employed for



▲ DENSO continues to embed CAE more deeply within its global product development process, and the company maintains its strong strategic partnership with ANSYS by sharing visions and goals, as well as through ongoing and future joint projects.



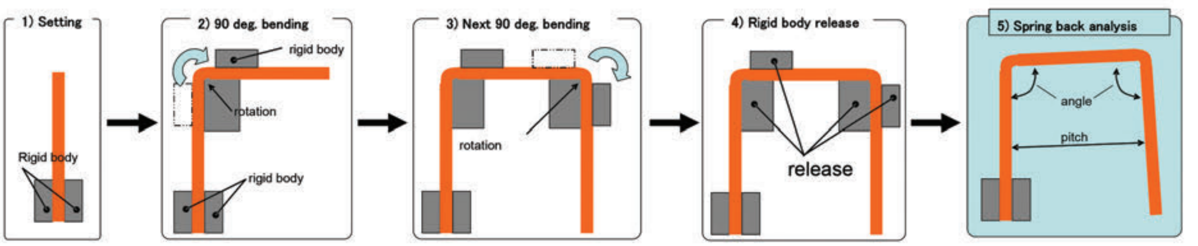
▲ Crack growth analysis allows DENSO to predict when failure might occur. This helps to ensure product reliability by visualizing the stress distribution based on crack profile changes.

similar purposes, identifying the best one in each category and standardizing on the best code company-wide. In the analysis code category, DENSO identified 69 capabilities that were needed for structural simulation and asked the two leading code vendors in this category what capabilities they could provide now and in the future.

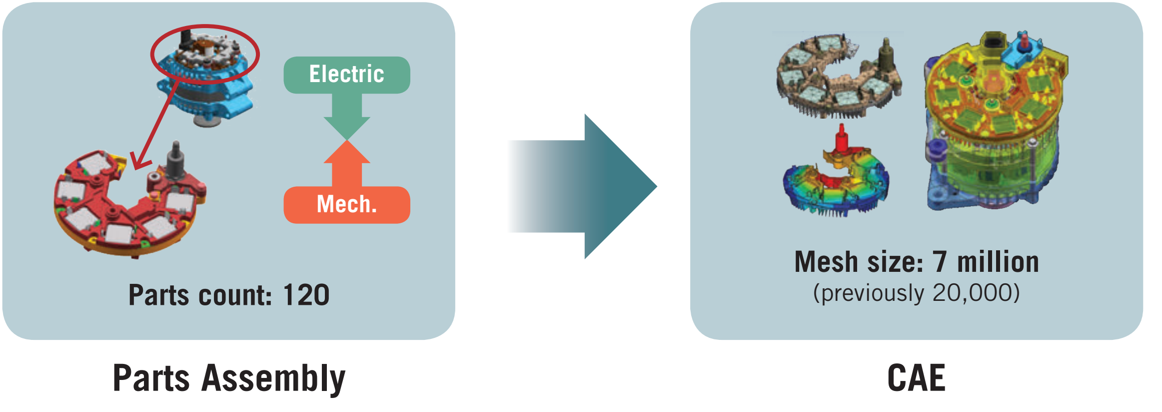
BENCHMARK STUDY IDENTIFIES ANSYS AS TECHNOLOGY LEADER

ANSYS delivered 38 of these capabilities directly and provided another 19 with workarounds. ANSYS also provided a timetable for delivering another three functions and promised to develop the remaining nine in the near future. The other vendor could offer only eight of the 69 capabilities outright and provide a workaround for 12 more. That vendor also promised to develop 42 more functions at some unspecified point in the future and could make no promises for the final seven. DENSO also surveyed its user community and found that 80 percent favored adopting ANSYS Mechanical as the company standard while only 20 percent favored the other vendor's software.

The benchmark study focused on finite element analysis software, which was the primary analysis used at DENSO at the time of the study. DENSO selected



▲ By using simulation to predict profiles after the forming process, geometric accuracy can be improved.



▲ By simulating the entire assembly, DENSO can predict strength and fatigue to ensure product dependability.

ANSYS software largely because of ANSYS Mechanical's advanced analytical abilities for structural linear, nonlinear and dynamics analysis; its ability to model with elements; its library of material models and equation solvers; and its scalability in efficiently solving a range of engineering problems and scenarios. DENSO also found the support provided by long-term ANSYS channel partner Cybernet Systems to be especially valuable. After the study was completed, DENSO increased its usage of additional ANSYS multiphysics capabilities including CFD and electromagnetic simulation.

EMBEDDING CAE INTO THE PRODUCT DEVELOPMENT PROCESS

DENSO has developed a strategy to embed CAE fully into all phases of the global product development process. The mission of the Digital Engineering Department is to enhance CAE technology in each of the company's business groups by developing methods to solve typical product development problems. Each business group has a CAE team responsible for developing product-specific CAE tools for use in the design process. Engineers from overseas business units are trained at headquarters to enable joint development of CAE technology in the future.

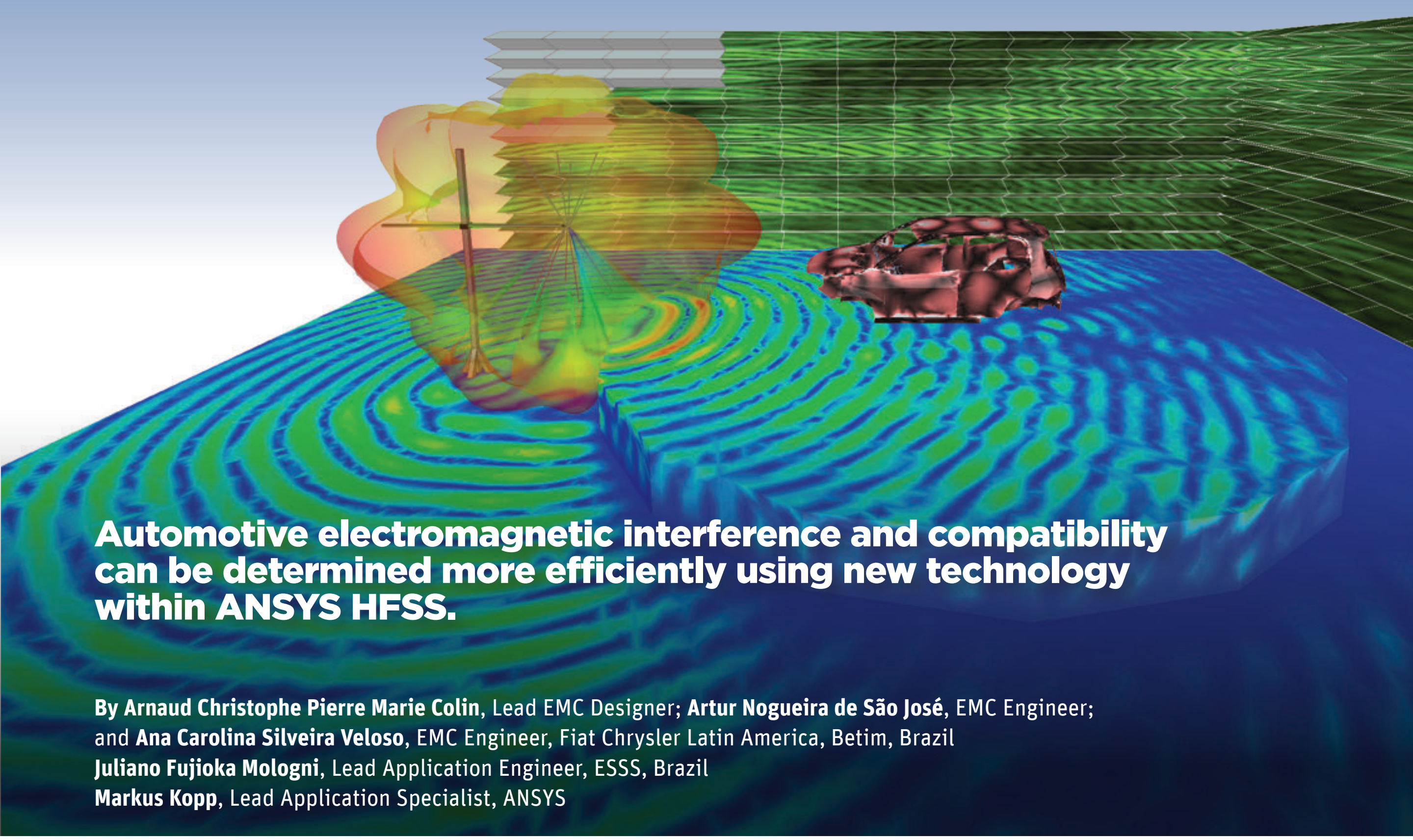
DENSO has determined that its need for multiphysics analysis will increase greatly in the future. For example, research and development teams working on hybrid vehicle/electric vehicle motor generator design must address structural and thermal considerations along with electromagnetic design constraints. Kinematics–vibration–noise analysis is required to address environmental problems in the design of turbomachinery, compressors and belt drives. Because of the risk of supply cutoffs, multiphysics analysis is vital in developing alternative materials that frequently need to be considered.

DENSO has determined that its need for multiphysics analysis will increase greatly in the future.

Collaboration with academia helps DENSO develop basic theories that can be embedded into the software. The company is working with universities on the flow-by-particle method, high-precision electromagnetic fields, polymeric heat transfer characteristics, metal-to-metal joints and magnetic particle compression.

As a result of these efforts, DENSO has improved product quality by considering more alternatives upfront and has compressed the product development process. DENSO intends to continue to embed CAE more deeply within its global product development process and maintain its strong strategic partnership with ANSYS through sharing visions and goals, as well as ongoing and future joint projects.

TEST DRIVE FOR EMI



Automotive electromagnetic interference and compatibility can be determined more efficiently using new technology within ANSYS HFSS.

By **Arnaud Christophe Pierre Marie Colin**, Lead EMC Designer; **Artur Nogueira de São José**, EMC Engineer; and **Ana Carolina Silveira Veloso**, EMC Engineer, Fiat Chrysler Latin America, Betim, Brazil
Juliano Fujioka Mologni, Lead Application Engineer, ESSS, Brazil
Markus Kopp, Lead Application Specialist, ANSYS

Automobiles are fast becoming mobile hotspots. Components such as wireless links, multimedia devices, electronic control modules and hybrid/electric drives are continually being added to vehicles, which makes designing for electromagnetic interference (EMI) and electromagnetic compatibility (EMC) increasingly important. At Fiat Chrysler in Brazil, a team of engineers is certifying the complete product integrity by investigating potential EMI on its vehicles using ANSYS HFSS and full-vehicle testing.

Because electronics have been rapidly added to automobiles, a number of guidelines have been developed, including legislation, industry association standards and even regulatory limits that are specific to a particular automotive manufacturer. One of the earliest industry directives was issued in Europe in

As automobiles become mobile hotspots and electronic components are continually added to vehicles, designing for EMI and EMC becomes increasingly important.

1972 to deal with electronic spark plug noise; since then, many organizations have created a variety of standards specifically for the automotive industry.

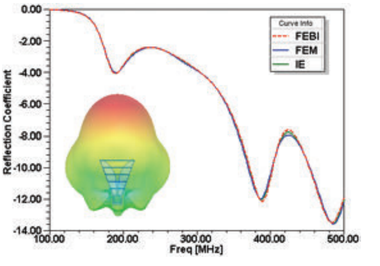
Many standards, directives and regulations are designed with vehicle safety in mind. This ensures that all onboard systems continue to function properly during exposure to EMI and then return to normal, either automatically or by a manual reset operation after exposure. A major concern for Fiat Chrysler's engineers is the amount of wires a car now contains — as much as 5 kilometers per vehicle. While cabling is an obvious source of EMI, there are a number of other sources in modern vehicles that are packed with electronics. In addition, drivers introduce potential EMI sources in the form of mobile phones, tablets and Bluetooth®-enabled devices. Automobile manufacturers that create smart vehicles need to meet standards to reduce risk of failure.

Conventional EMI/EMC procedures and techniques are no longer appropriate for the latest generation of electronic devices and components. A few automotive standards have been developed that use laboratory tests in an attempt to reduce the probability of EMI occurring in vehicles. One important international lab-based standard is ISO 11451-2. This standard calls for testing a source antenna that radiates throughout the vehicle in an anechoic (echo-free) chamber; the performance of all electronic subsystems must not be affected by the electromagnetic disturbance generated by the source antenna.

ISO 11451-2 is meant to determine the immunity of private and commercial road vehicles to electrical disturbances from off-vehicle radiation sources, regardless of the vehicle propulsion system (including hybrid/electric vehicles). The test procedure prescribes performance on a full vehicle in an absorber-lined, shielded enclosure, creating a test environment that represents open-field testing. For this test, the floor generally is not covered with absorbing material, but such covering is allowed.

Testing for the standard consists of generating radiated electromagnetic fields using a source antenna with radio frequency (RF) sources capable of producing the desired field strengths ranging from 25 V/m to 100 V/m and beyond. The test covers the range of frequencies from 10 kHz to 18 GHz. During the procedure, all embedded electronic equipment must perform flawlessly. This flawless performance also applies to the frequency sweep of the source antenna.

Physically performing the ISO 11451-2 standard test can be a time-consuming process that requires costly equipment and access to an expensive test facility. Numerical simulation can be a cost-effective, alternative means to reduce the product design cycle and its associated R&D costs. Full-vehicle finite element method (FEM) simulation has become possible within the past few years using the domain decomposition method (DDM), which was pioneered by ANSYS HFSS software. DDM parallelizes the entire simulation domain by creating a number of subdomains, each solved on different computing cores or various computers connected to a network. While DDM allows engineers to simulate entire vehicles, there is another approach available within HFSS for solving large electromagnetic structures: a hybrid finite element–boundary integral (FE-BI) methodology.

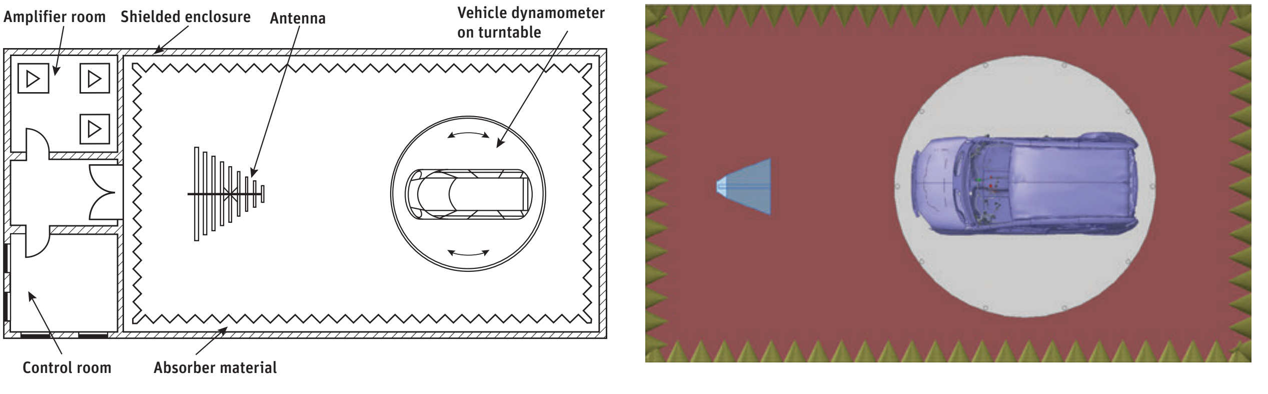


A comparison of far-field behavior shows that the FE-BI method is accurate compared to traditional methods.

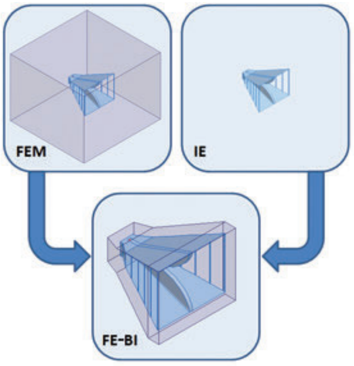
ELECTROMAGNETIC SIMULATION OF ANTENNAS INSTALLED INSIDE VEHICLES: AN AUTOMOTIVE EMC APPROACH
ansys.com/92emi

FE-BI uses an integral equation (IE)–based solution as a truncation boundary for the FEM problem space, thus bringing together the best of FEM and IE. This combination of solution paradigms allows Fiat Chrysler engineers to dramatically reduce the simulation's solution volume from that required by the FEM method. Because the distance from radiator to FE-BI boundary can be arbitrarily small, solution time is decreased, as is the overall computational effort.

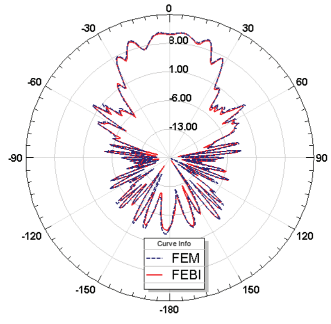
To demonstrate the capability of the FE-BI methodology, the Fiat Chrysler team worked with ESSS, the ANSYS channel partner in South America, to conduct



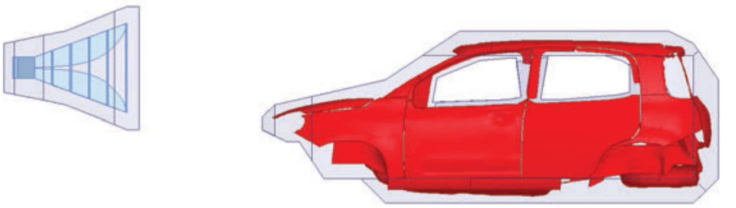
ISO 11451-2 test apparatus (left). Virtual test chamber used for ANSYS HFSS simulation (right)



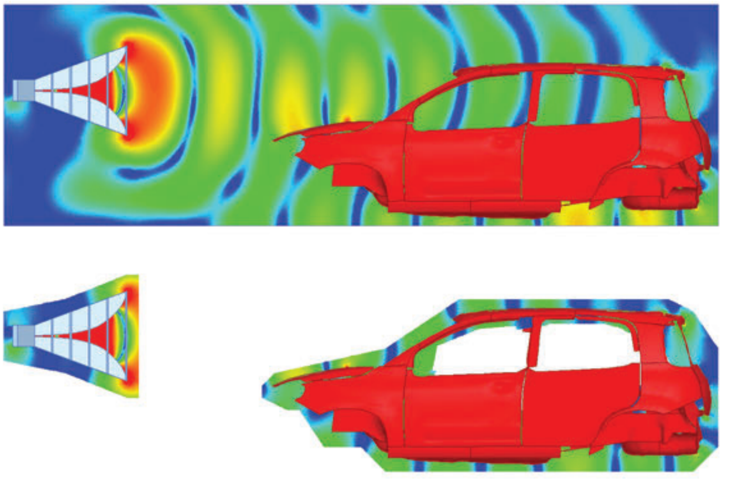
▲ Comparison of FEM, IE and FE-BI models. The reduced size of the solution region when using FE-BI (as compared to traditional FEM or other numerical 3-D field solvers) leads to a faster simulation.



▲ Antenna far-field pattern at Φ=90 degrees for the whole model



▲ Two subregions were solved simultaneously using the HFSS FE-BI solver. The air region is shown in light blue, and the majority of the air volume has been removed.



▲ The electric field on both the surface shell and a plane that bisects the solution volume of the vehicle for traditional FEM (top) and FE-BI (bottom) results

a full-vehicle simulation using the FE-BI capability. The team then applied the results to the ISO 11451-2 standard to determine EMI of an electronic subsystem. For the simulation, the team reduced the large air region in the test chamber to two much smaller air boxes that more closely conformed to the structures they contained. The surfaces of these air regions were located close to the antenna and the vehicle.

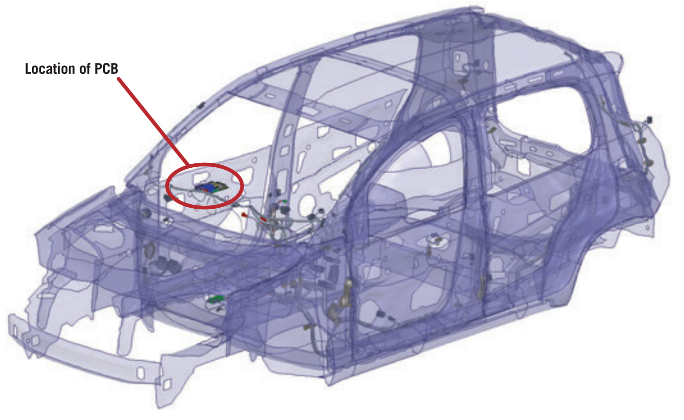
Fiat Chrysler engineers did not model the absorber elements in this simulation because the IE boundary in FE-BI is equivalent to a free-space simulation, which is the same as absorbing material used in a physical measurement. The total computation time of just 28 minutes represents more than a 10-fold speedup when compared to a traditional FEM solution. Additionally, the total amount of RAM required for the FE-BI simulation was 6.8 GB, which is also more than a 10-fold decrease compared to previous work using FEM.

Solution results using the FE-BI method showed that the predictions for the quantities of interest were in excellent agreement with those obtained using FEM. The electric fields calculated on both the vehicle's surface shell and a plane that bisects the car were very similar for both solution methods, as were the total far-field patterns of the entire model.

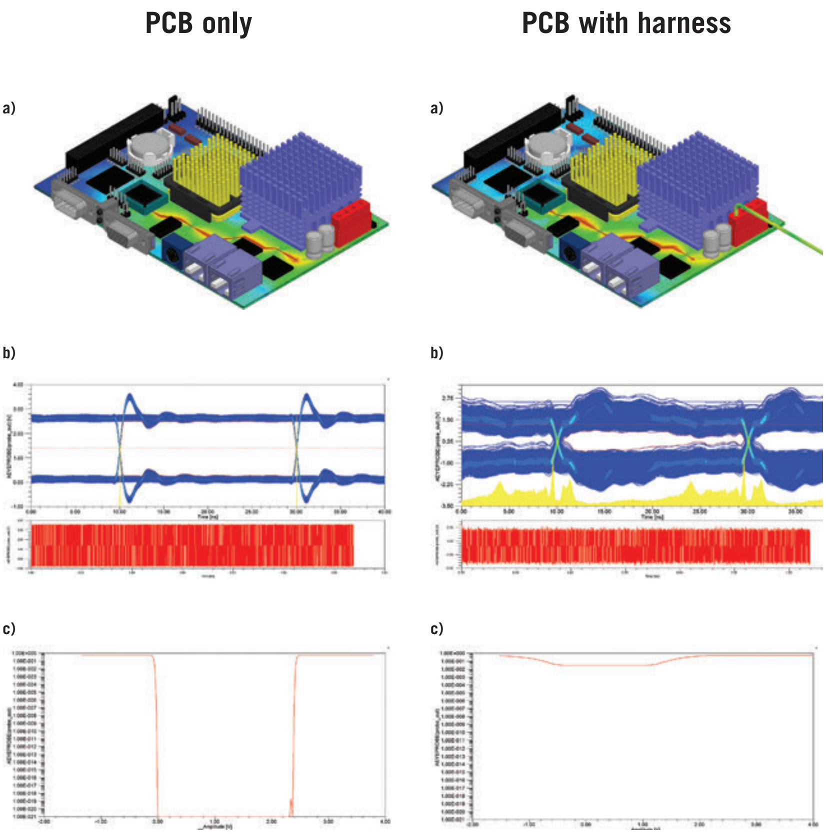
The FE-BI approach also can be used to test the immunity of embedded control unit modules. To demonstrate this capability, the engineering team introduced a printed circuit board (PCB) connected to the engine wiring harness into the simulation. The transmitted signal travels from a sensor, located at the bottom of the engine, to the PCB using a wiring harness that is routed around the engine. The wiring harness end is attached to the PCB using a red four-way connector. One of the four-way connector's pins is soldered to a trace that begins in the top side of the PCB on the connector side and then goes through a via to the bottom side, where it is connected to the microcontroller. In this case, the team analyzed only a single onboard diagnostic (OBD) protocol CAN J1913 signal.

The wiring harness plays a vital role in EMI because the harness can act as a radiation source. To better understand the effect of the wiring harness, ESSS engineers performed two simulations. The

The ANSYS HFSS FE-BI capability is clearly advantageous as a numerical technique to simulate a full vehicle according to automotive EMC standards.



▲ Location of PCB relative to vehicle



▲ Simulation of CAN J1939 signal in PCB alone (left) and in PCB with wiring harness (right): a) Electric field plot distribution. b) Eye diagram of received signal at microprocessor. EMI is observed when the harness is attached. c) Bathtub diagram for signal being received at microcontroller. The bathtub curve is greatly affected by the EMI source, with a final bit error rate of 1E-2. This means that one bit out of every 100 will be incorrectly interpreted by the microcontroller.

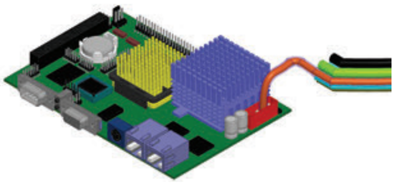
first included the PCB and wiring harness along with the chassis and source antenna. For the second simulation, the team removed the wiring harness, and the random CAN J1939 signal was applied directly into the PCB connector instead of at the sensor location at the bottom of the engine.

Using the FE-BI solver in HFSS, the team easily calculated the electromagnetic fields and the scattering parameters of the two simulations (with and without wiring harness). The simulations showed a resonance on the PCB when connected to the wiring harness. The frequency of this resonance is a function of the length of the cable attached to the PCB. When attached to the PCB, the harness increased the coupling between source antenna and PCB by more than 30 dB between 152 MHz and 191 MHz.

Finally, the engineers dynamically linked the 3-D electromagnetic model to the ANSYS circuit solver available in HFSS to simulate the CAN J1939 signal in the wiring harness and PCB. The frequency-domain field results produced by HFSS were seamlessly combined with time-based signals using ANSYS Designer software. In Designer, it is possible to specify the various signals that excite both the antenna and the wiring harness. For these simulations, the team set the antenna excitation to a constant 150 V sinusoidal signal, with a delay of 8 µs and a frequency sweep varying from 10 MHz to 500 MHz. The initial time delay was set to clearly see the effect of EMI on the transmitted signal. The team generated the CAN J1939 signal at the sensor end of the harness for the first simulation; for the second simulation, the signal was injected directly at the connector with no wire harness present. Simulation with the harness shows that the overall sensor system performance will be greatly affected by incoming radiation in the 152 MHz to 191 MHz band.

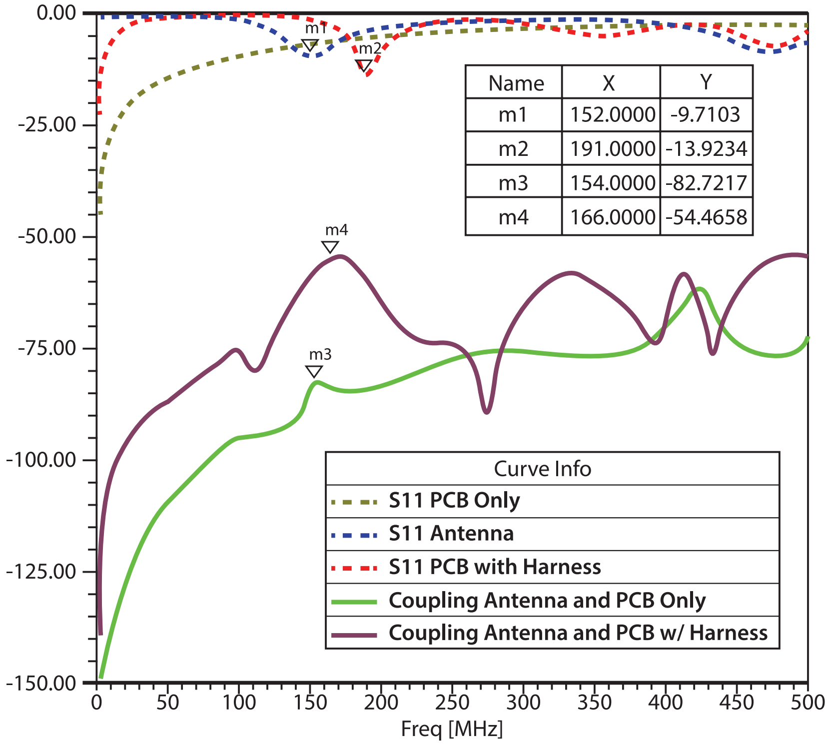
The ANSYS HFSS FE-BI capability is clearly advantageous as a numerical

Simulation allows for accurate what-if analysis to determine potential EMI issues caused by driver- or passenger-introduced electronic communications devices.



Point where wiring harness attaches to PCB

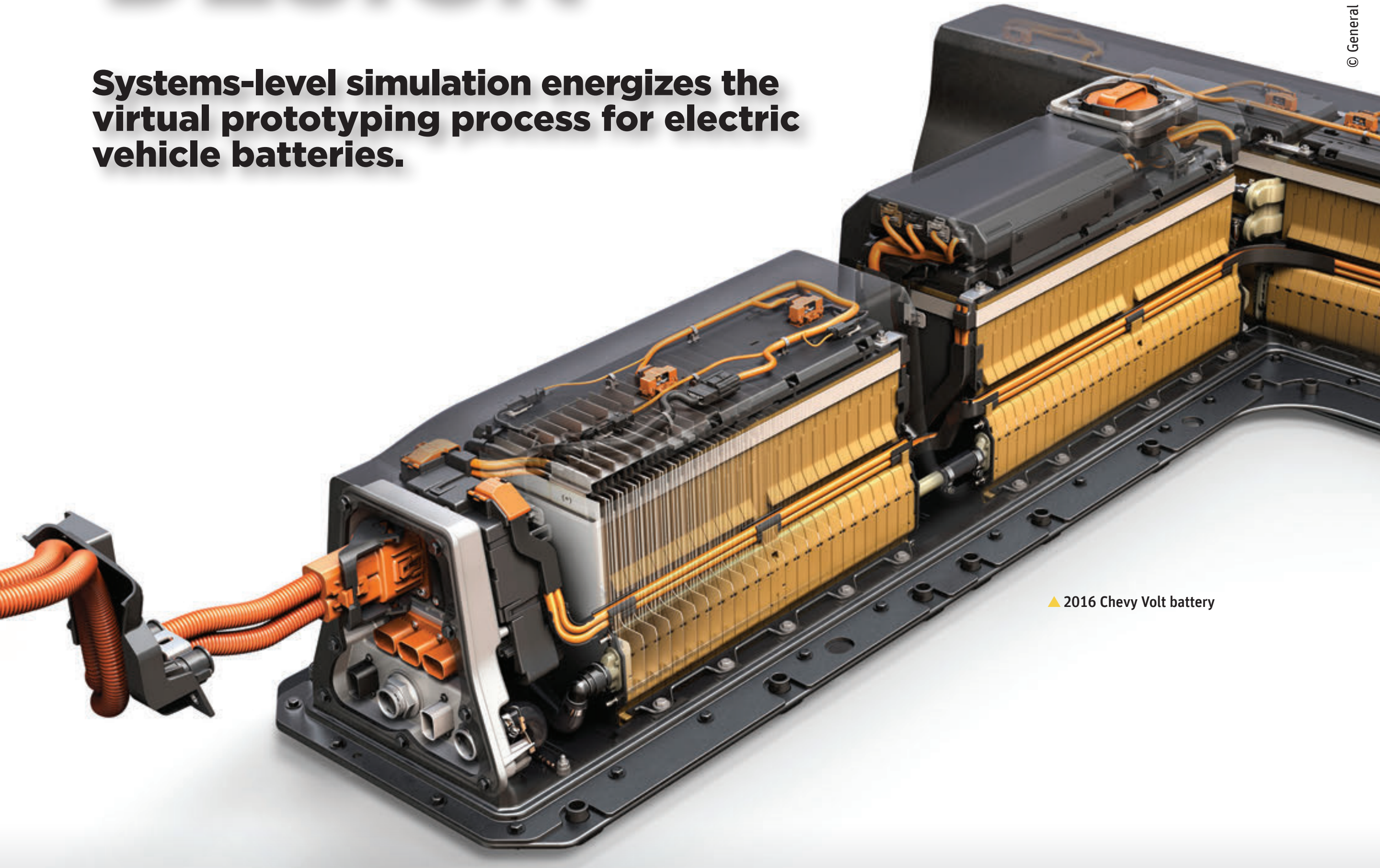
technique to simulate a full vehicle according to automotive EMC standards. The FE-BI technique was over 10 times faster and required 10 times less computational effort than a traditional FEM simulation. As a result, EMI/EMC engineers can begin to simulate entire vehicles and their subsystems in virtual anechoic chambers to meet EMC and EMI standards. Using simulation also allows for accurate what-if analysis to help determine potential EMI issues caused by driver- or passenger-introduced electronic communications devices. It also leads to a better understanding of transient noise issues caused by the myriad of motors included in every vehicle.



S–matrix with PCB, both alone and connected to wire harness

AUTOMATING BATTERY PACK DESIGN

Systems-level simulation energizes the virtual prototyping process for electric vehicle batteries.



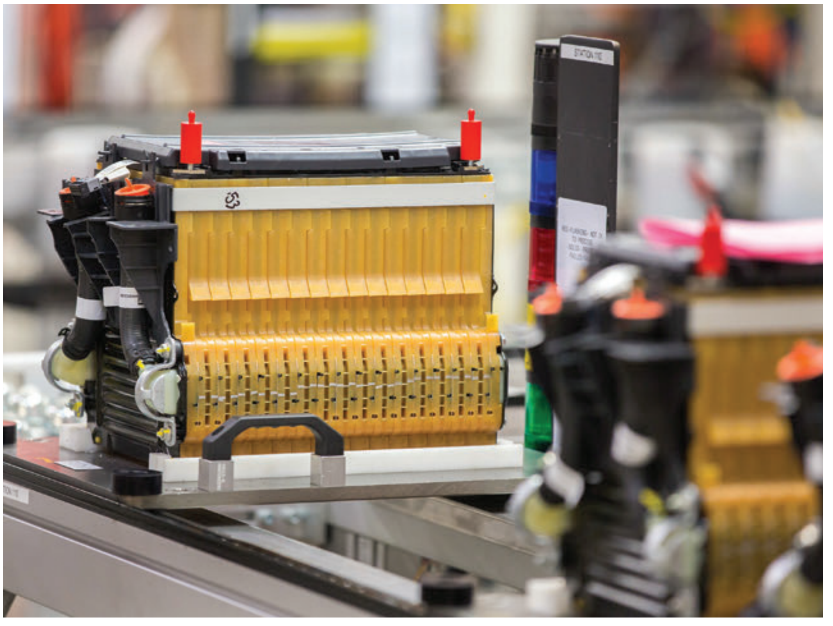
© General Motors.

2016 Chevy Volt battery

By Erik Yen, Senior Researcher, and **Taeyoung Han**, Technical Fellow, Vehicle Systems Research Laboratory, General Motors Research and Development Center, Warren, U.S.A.
Sameer Kher, Senior Manager, Software Development, ANSYS

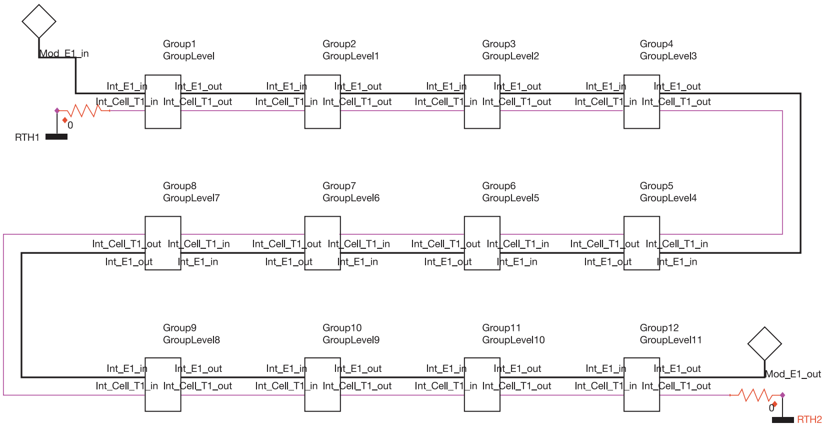
Since their commercial introduction in the early 1990s, lithium-ion batteries have emerged as the most popular form of rechargeable energy storage devices in the portable electronics and electric vehicle markets. The lightweight lithium compounds that comprise the electrodes result in a high specific energy (watt-hours/kilogram) as compared to other types of batteries. While a few battery cells may be sufficient for a phone or laptop, it is necessary to connect many hundreds of individual cells together as part of a much

For electric vehicle makers, designing an efficient and robust cooling system for the battery pack is a key engineering task.

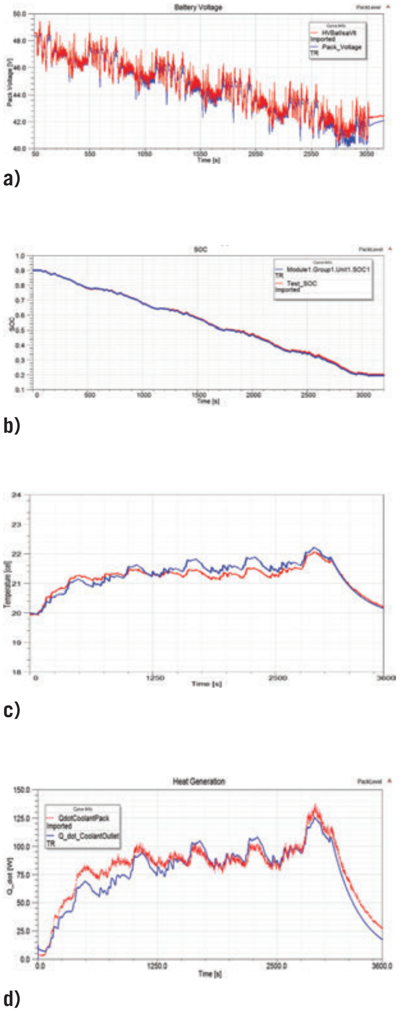


© General Motors.

▲ General Motors electric vehicle battery production



▲ ANSYS Simplorer model of 24-cell battery module, consisting of 12 two-cell units with automatic electrical and thermal connections



▲ ANSYS Simplorer predictions for a 24-cell battery module in blue as compared to experimental data in red measured using the US06 driving schedule for overall pack voltage a), cell state-of-charge b), average cell temperature c), and pack heat generation d).

larger battery pack system to power an electric vehicle.

Seeking to further increase the specific energy of electric vehicle (EV) batteries, while also reducing the overall size and weight of the battery system and maintaining safe operating conditions, automakers and their suppliers have worked together — with the support of the U.S. Department of Energy (DOE) Vehicle Technologies Office — to attack several grand challenges put forth in the EV Everywhere technology blueprint. To meet the ambitious goals of EV Everywhere — which include reducing energy costs to $125 U.S.D./kilowatt-hour by 2022 — the use of simulation tools to design battery systems and accurately predict their performance is a vital component of the R&D strategy.

Beginning in 2012, General Motors led a team working under a program administered by DOE's National Renewable Energy Laboratory known as the Computer-Aided Engineering for Electric Drive Vehicle Batteries (CAEBAT) project. The team consisted of GM researchers and engineers, ANSYS software developers and applications engineers, and the staff of ESim LLC. One of the objectives of the GM CAEBAT project has been development of battery pack design tools, which included leveraging and extending the capabilities of systems-level simulation packages.

CAEBAT BATTERY THERMAL MANAGEMENT PROJECT BY GENERAL MOTORS, ANSYS AND ESIM
ansys.com/92battery

PACK-LEVEL ANALYSIS

Because a vehicle battery pack may contain hundreds or even thousands of cells that exhibit tightly coupled electrochemical and thermal behavior, one particular challenge is to maintain an optimum range of system operating conditions to minimize material degradation and loss of capacity. From the perspective of an automotive OEM, keeping the whole pack within the temperature range of 25 C to 35 C (77 F to 95 F) is crucial for the reliability of

Incorporating this kind of simulation into the process helps guide the overall pack design direction as automakers seek to meet DOE's programmatic goals and address the demands of the growing EV consumer market.

FAST-CHARGING BATTERY DEVELOPMENT
ansys.com/92battery2

the system. Because temperatures in the surrounding environment can span from –40 C to 50 C (–40 F to 122 F), the temperature uniformity of the individual cells is maintained by a dedicated thermal management system. For electric vehicle makers, designing an efficient and robust cooling system for the battery pack is a key engineering task.

To analyze the coupled electrochemistry and heat transfer of a pack, it is desirable to have predictions based on fine spatial resolution of the entire system of battery cells. However, such information may be available only through resource-intensive, time-consuming full-field simulation, which is not always practical during tight vehicle development cycles. In addition, engineers must capture transient conditions that affect the load on the pack during a variety of driving schedules, such as the EPA's US06 cycle that represents aggressive driving behavior with a variety of brisk changes in speed. A systems-level approach using ANSYS Simplorer can provide an effective solution when complete field data is not necessary. Automotive engineers require quick turn-around time between design iterations to evaluate potential cooling system designs.

A SYSTEM OF UNIT MODELS

To address these kinds of design challenges, GM researchers deconstructed the full pack domain in Simplorer to first create a representation of a battery unit model. The unit model is a combination of one or more battery cells and the adjacent cooling channel. Using off-the-shelf Simplorer components to represent the internal resistors, capacitors, and sources of both electrical

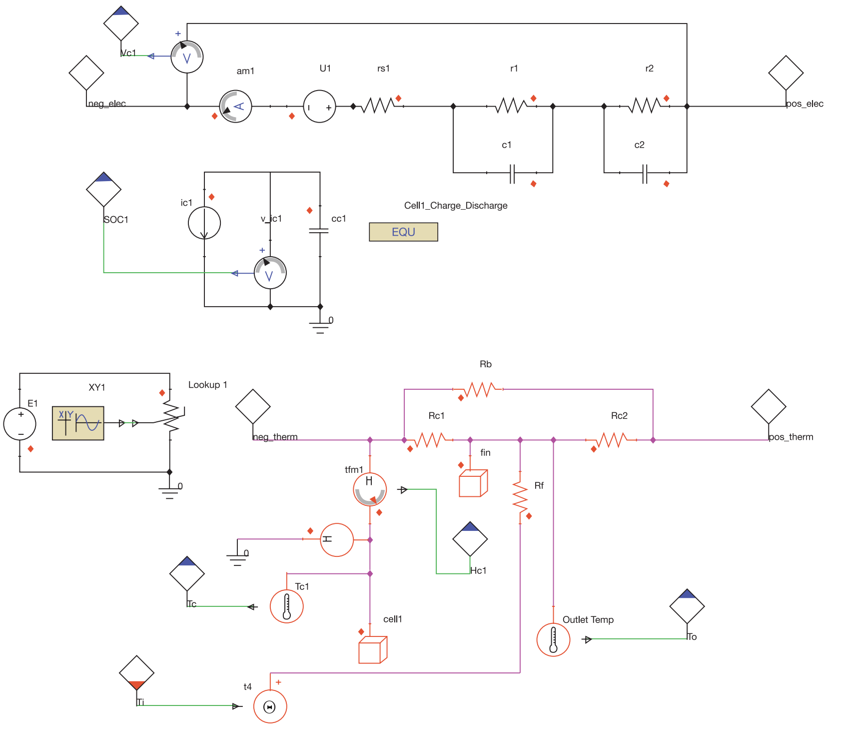
© General Motors.

▲ Chevy Bolt concept battery electric vehicle and thermal behavior, the research team created several unit-model variations based on theoretical and empirical formulations for both circuit and heat-transfer modeling. Once completed, these units are easily stored in a Simplorer user library for later use by a production pack designer.

Within the pack, the individual cells are wired electrically in parallel to form groups, and the groups are wired in series to form modules. To automate the process of replicating and connecting units, groups and modules together into a pack, the CAEBAT team created a Python-scripted extension to the Simplorer user interface that requires just a few integer value inputs to specify the pack configuration. With the positioning, wiring and hierarchical layout complete, the Simplorer extension then adds custom components written in the VHDL-AMS modeling language to represent the coolant manifold, along with the transient load to represent the driving schedule. The pack designer can then change the parameters for any individual unit in the pack to analyze potential thermal runaway, or can replace units with others from the user library to consider the effects of cell-to-cell manufacturing variations. This combination of automation and flexibility enables the CAEBAT team to evaluate numerous pack configurations, consider different profiles for the coolant flow rate, and predict the thermal and electrical responses to driving schedules like US06.

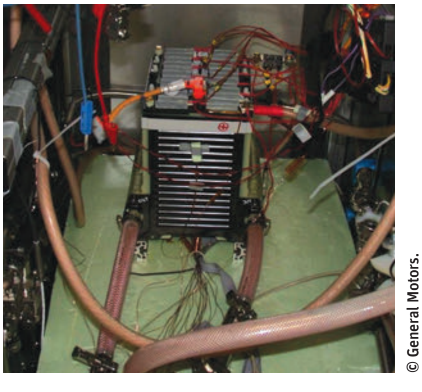
VALIDATION AND FUTURE WORK

GM's researchers validated the systems-level approach by comparing the Simplorer model of a 24-cell reference battery module to experimental test results. The GM unit model included a six-parameter electrical circuit submodel coupled to a thermal circuit sub-model. Resulting Simplorer predictions of the overall pack voltage, the state of charge and the average temperature of each cell closely followed the trends observed in laboratory experiments. In the longer term, CAEBAT team partners are investigating further enhancements to the systems-level simulation approach. These include the addition of battery-life modeling to predict the capacity fade of cells over long-term use, and expanding the capability to examine individual cells in more detail by replacing selected units in the Simplorer pack model with full 3-D ANSYS Fluent cell models as well as reduced-order models. The information provided by the systems-level approach will be especially critical to GM for trade studies regarding questions — such as air cooling versus liquid cooling, battery form factor or effects of battery management system control logic — that must be answered before building costly prototype hardware. Incorporating this kind of simulation into the process helps to guide the overall pack design direction as automakers seek to meet DOE's programmatic goals and address demands of the growing EV consumer market.



Example ANSYS Simplorer unit model, including six-parameter electrical circuit model (top) and thermal model representing a battery cell and cooling channel (bottom)

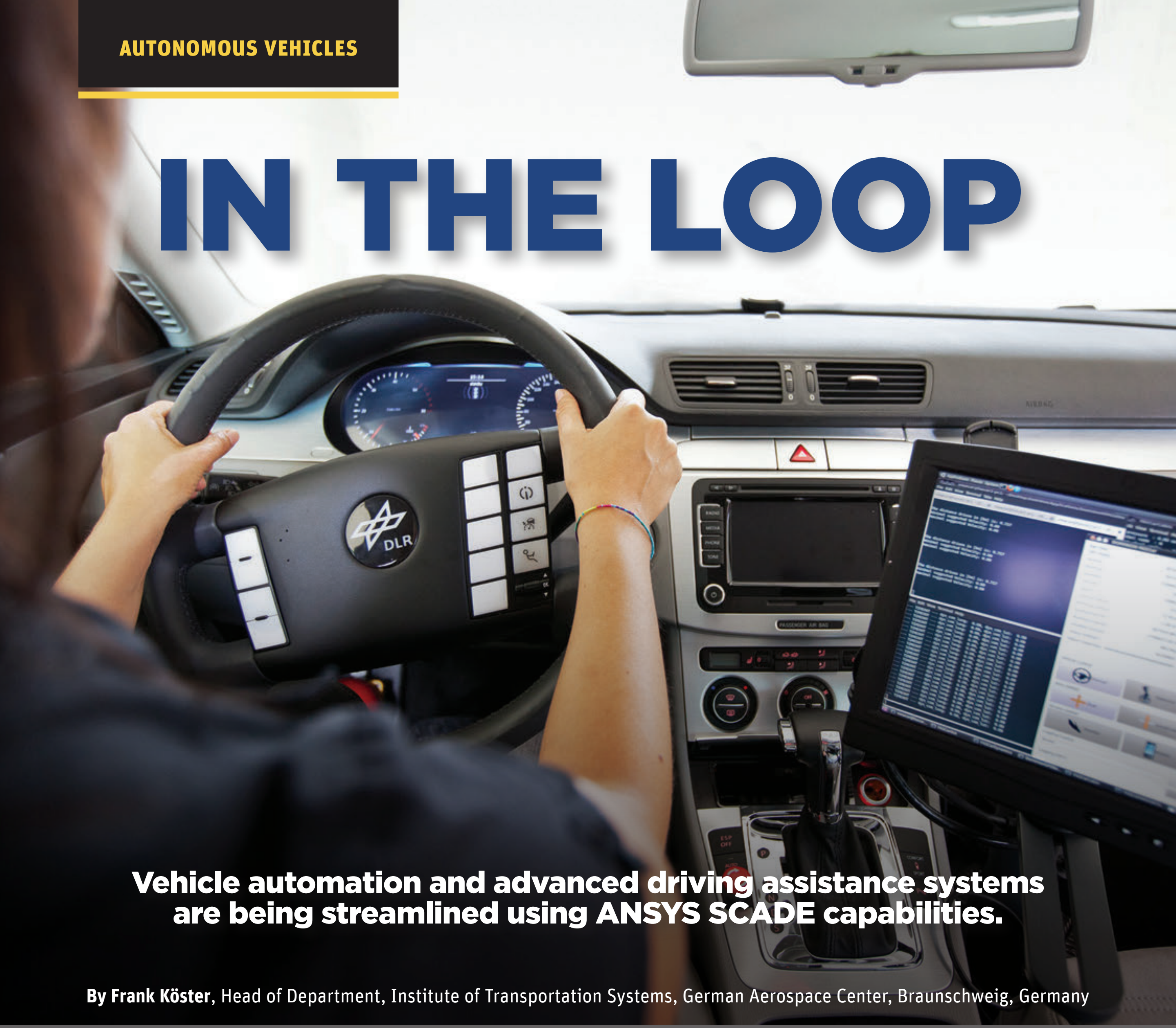
The combination of automation and flexibility enables the CAEBAT team to evaluate numerous pack configurations.



GM prototype 24-cell module featuring steady-state liquid cooling used for experimental validation

ACCELERATING DEVELOPMENT OF EV BATTERIES THROUGH COMPUTER-AIDED ENGINEERING
ansys.com/92battery3

IN THE LOOP



Vehicle automation and advanced driving assistance systems are being streamlined using ANSYS SCADE capabilities.

By Frank Köster, Head of Department, Institute of Transportation Systems, German Aerospace Center, Braunschweig, Germany

Vehicle automation offers the potential to substantially reduce the hundreds of thousands of deaths — including motorists, motorcyclists, bicyclists and pedestrians — that occur each year around the world in automobile accidents. This automation provides the opportunity to improve traffic flow, increase driver comfort, and reduce fuel consumption and emissions. Many vehicles already have automated lighting, intelligent parking-assist systems, proximity sensors with alarms and other automated systems. However, there are many technical, regulatory and legal obstacles to fully autonomous vehicle operation or self-driving cars. Only a few U.S. states currently allow semi-autonomous vehicle operation (for example, systems that can take over control of the vehicle if the driver makes a mistake), and fully autonomous, driverless vehicles are not allowed anywhere in the United States. For the foreseeable future, vehicle automation and advanced driving-assistance systems (ADAS) will supply assistance and automation ranging from full control by the driver to full control by the automation system. When developing these systems, a major challenge is transitioning between these different levels of automation, including transitions initiated by the driver and those initiated by the automation system.

Vehicle automation offers the potential to substantially reduce hundreds of thousands of deaths.

The Institute of Transportation Systems at the German Aerospace Center (DLR) is working in cooperation with leading automobile original equipment manufacturers (OEMs) to develop vehicle automation and ADAS that will overcome this and other challenges. DLR is combining its technological expertise with psychological and ergonomic research to produce vehicle automation systems that can be tailored to the capabilities and expectations of each driver. Systems that are currently under development involve integration of the

Dynamic driving simulator

MODEL-BASED SYSTEMS ENGINEERING: BUSINESS OPPORTUNITIES AND OVERCOMING IMPLEMENTATION CHALLENGES
ansys.com/92loop

driver and automation system so that, for example, when the limits of the automation system are reached, control is handed back to the driver. In this situation, the driver needs to be presented with the right information at the right time by the human–machine interface (HMI) so that he or she can safely resume control of the vehicle. DLR uses ANSYS SCADE Suite and ANSYS SCADE Display to develop HMIs in the model environment using prebuilt and specific components. By simulating behavior using the model, it is possible to identify and correct defects as well as gain critical insights early in the design process to rapidly improve system performance.

HMI on heads-up display for highly automated driving

By simulating behavior using the model, it is possible to identify and correct defects as well as gain critical insights early in the design process to rapidly improve system performance.

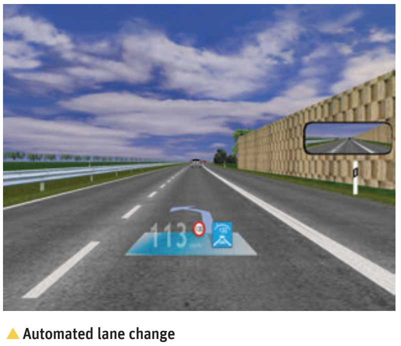
ROLE OF THE HUMAN–MACHINE INTERFACE

One example of how the HMI operates occurs when the automation system cannot sense the lane markings due

The time required to develop and validate the HMI was been substantially reduced by moving the development process to SCADE Suite and SCADE Display.

to dirt on the road, in which case it may need to return control to the driver. The HMI generates acoustic, tactile and visual alarms to bring the driver back into the loop; it also performs various checks to confirm that the driver has taken over as intended, for example, by sensing that the driver has gripped the steering wheel. If the driver does not react, the automation system brings the vehicle to a safe stop. In the reverse situation, when the driver is controlling the vehicle and the automation system senses sudden danger, the system may issue a warning to the driver and take over control to avoid an accident.

Management of this handover process is just one of the many functions performed by HMIs that are continually gaining functionality as vehicle automation systems continue to mature. As a result, the HMI development process has become increasingly challenging. In the past, when HMIs were developed using manual coding methods, developers typically did not receive feedback until the code was compiled and run on the expensive and complicated target hardware environment. Making changes to the HMI was difficult because the engineer making the changes had no way to validate them until the code could be run on the target. Many different scenarios had to be evaluated in the target environment for each iteration of the HMI, which was a long process. A considerable manual coding and testing effort was required to make changes to the HMI, such as moving an element from one display to another.



▲ Automated lane change

▲ Warning presented on HMI

TRANSITION TO MODEL-BASED DEVELOPMENT

The time required to develop and validate the HMI has been substantially reduced by moving the development process to SCADE Suite and SCADE Display. Functional requirements and test cases are linked to the SCADE model using the SCADE Requirements Management Gateway. DLR engineers now use a model-based design approach built on the creation of an executable model in a block-diagram design environment. Engineers define the functionality of the HMI using blocks that represent algorithms or subsystems. They have created a library of blocks in the SCADE environment that perform and display common vehicle automation HMI functions, so the development process largely consists of selecting and adapting existing blocks and connecting their outputs and inputs.

Engineers simulate the behavior of the model and receive immediate feedback on its performance. Test cases are run in the virtual PC environment rather than in the more-complicated and expensive target environment. For example, for each new iteration of the code, engineers must check hundreds of different scenarios to ensure that certain information is presented on the screen at critical points, such as handoff from the automation system to the driver. In the past, this involved a lengthy manual process. Now, the engineer developing the model can run an automated routine that quickly evaluates each scenario.

AUTOMATIC CODE GENERATION

After the model has been validated, the SCADE KCG code generator produces code for the target environment. The SCADE Suite KCG C code generator provides complete traceability from model to generated code by establishing an unambiguous one-to-one relationship between the model and the code. The code is first run in different DLR driving simulators, including, finally, DLR's dynamic driving simulator, which combines a high-fidelity immersive visual system with an integrated cockpit and a hydraulic motion system to form a realistic driving environment to test prototype automation systems. Here engineers can evaluate HMI performance in driving scenarios that are very close to reality, such as bringing the driver back into the control loop when the system reaches its limits because of a construction detour. Once the HMI's operation is verified on the driving simulator, code is then generated for the test vehicle that can be controlled by a virtual copilot for system evaluation.

SCADE Suite and SCADE Display make it easy to modify the HMI to evaluate different alternative designs and to produce different variants of the HMI for different vehicles. DLR engineers can re-arrange the way that elements are positioned on the different displays of a vehicle simply by re-arranging blocks in the model. In the past, this would have required major amounts of manual coding. ANSYS SCADE Suite and SCADE Display have substantially improved the process of developing HMIs by continually testing and validating the HMI, first in the model phase, then in the vehicle simulator and finally in the target environment on the test vehicle, so that problems can be identified and corrected at the earliest possible stage.

MORE MUSIC, LESS NOISE

As automotive infotainment units become more complex, designers turn to simulation at the chip level to ensure reliable, noise-free performance.

By Jacob Bakker, Consultant Physical Verification, NXP Semiconductors N.V., Eindhoven, The Netherlands

Digital radios are becoming increasingly common accessories in automobiles, but their design poses a major challenge: The integrated circuit (IC) contains both analog and digital components that need to work together without disturbing each other. The power supply, external interfaces (USB, HDMI, VGA, etc.), radio signal receiver and sensor actuator are analog devices, while the data processing and memory storage units are digital. If they could be widely separated — for example, on two different ICs on one printed circuit board — then noise would not be a problem. But as designers continually compress components

The integrated circuit board contains both analog and digital components that need to work together without disturbing each other.

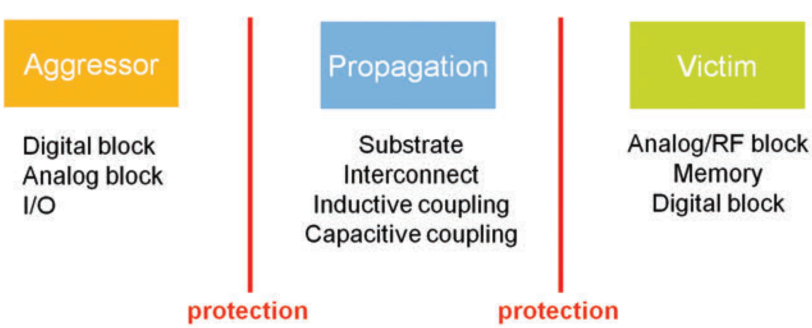
ANSYS RedHawk selects the worst power cycle automatically, so it is easy for engineers to see if their design meets specifications even in the worst-case scenario.

into smaller and smaller footprints to save space and cost, analog and digital signals are pushed closer together, increasing the chances of audible noise and sound disturbances coming from the radio.

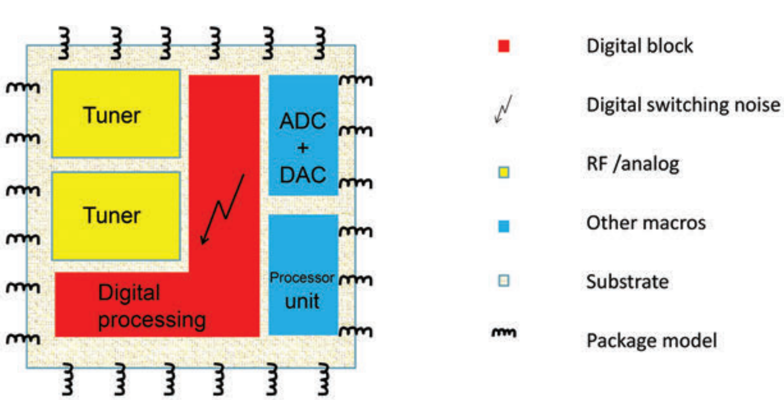
NXP Semiconductors N.V. in Eindhoven, The Netherlands, is the world's market leader in digital radio ICs for automobiles. As a global supplier, its products must be capable of decoding signals from all three major digital terrestrial radio standards that are popular in different parts of the world. Continuing a tradition of innovating complex electronics, the company recently produced a novel series of digital radio chips, dubbed SAF360x — integrating all three radios and up to six ICs into a single IC with a footprint that is 75 percent smaller and much more cost effective than its predecessor — using ANSYS Q3D Extractor and ANSYS RedHawk electronics simulation software to minimize analog–digital interference.

DIGITAL AGGRESSORS AND ANALOG VICTIMS

When radio frequency (RF) analog and digital baseband circuitry are combined on a single IC, the switching (from digital zero to one and vice versa) of the digital circuitry creates noise. Each switching event corresponds to a voltage swing between ground and the operating power-supply level. If too many



▲ Aggressor–victim model of noise propagation in digital–analog IC chips



▲ IC block floorplan used for simulation by NXP engineers

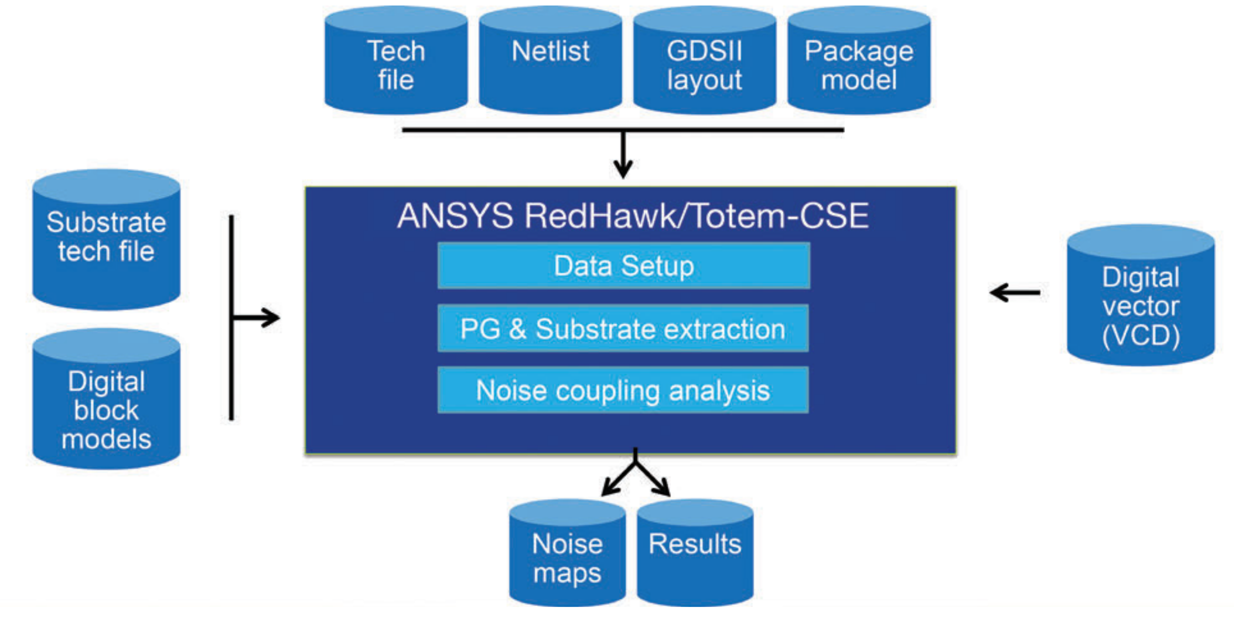
CHIP-PACKAGE METHODOLOGIES FOR AUTOMOTIVE AND EMBEDDED APPLICATIONS
ansys.com/92music

transistor gates switch at once, voltage on the power supply grid drops and ground bounce occurs, which can propagate from the digital side to the analog through the common substrate of the IC, interfering with analog devices and reducing performance. In the case of the SAF360x, the radio's sound quality could be impacted. The digital circuitry is referred to as the "aggressor" and the RF analog circuitry as the "victim." In digital radio ICs, the digital components are usually responsible for about 90 percent of the noise, caused by millions of switching transistors. When the same IC has multiple RF circuitry integrated, the noise coupling problem is huge. Both the frequency of digital switching and the proximity of the digital to the analog components are major design criteria in these circuits.

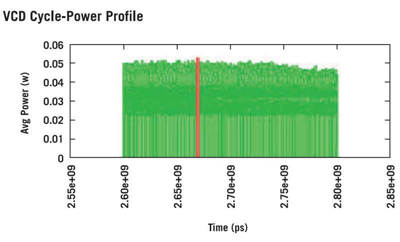
The noise signals must propagate through some medium connecting the analog and digital sections of the IC to produce interference. Often the silicon substrate is the major contributor to the propagation path. The noise within the IC travels through the resistive interconnects, capacitive coupling between the transistor junctions, or through the substrate. At the package level, the inductive coupling from I/O bonding wires serves as a medium.

SILENCERS

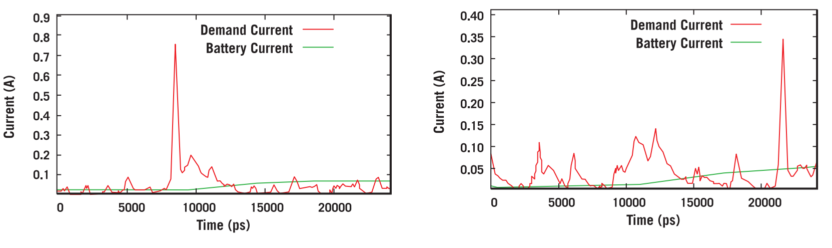
Two approaches are used to minimize interference in these digital–analog ICs: passive isolation structures that act as shields to prevent noise transmission, and optimal design of both analog and digital circuits to minimize noise concerns from the start. Passive isolation structures like guard rings provide a low-resistance connection to the substrate, ensuring that the



Extraction and analysis workflow for simulation



RedHawk automatically selects the worst power cycle (indicated in red).



In use case 1 (left), the digital switches were allowed to happen naturally, causing most switches to occur around 9,000 picoseconds; in use case 2 (right), engineers programmed the digital processing unit to spread switching out over a wider interval of time, resulting in lower, more-uniform current spikes. (Note the different current scales on the y axes.)

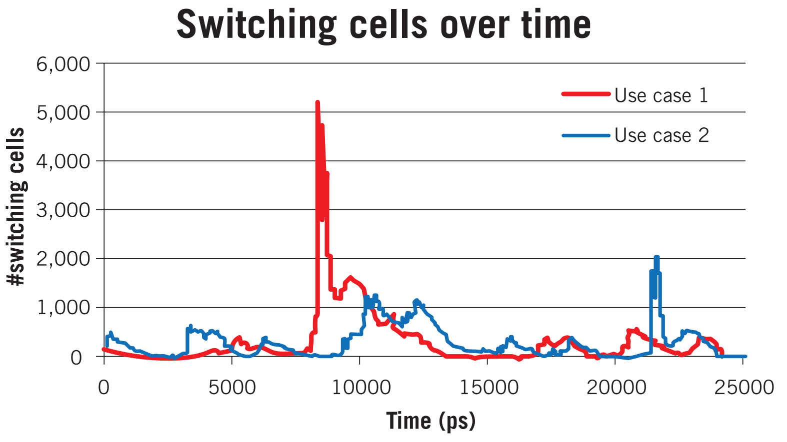
substrate is grounded at the location of the guard ring, thus making both analog and digital circuits less susceptible to noise; often multiple, parallel guard rings are needed. Deep N-well shields are another means of passive isolation, in which n-type (negatively charged) wells of material are implanted deep in the p-type (positively charged) substrate to minimize propagation of noise through the substrate. Optimal design involves practices that ensure that radio frequency and analog components are noise tolerant to begin with — perhaps through use of differential signals — and that digital circuits generate a minimum amount of noise when switching.

Using ANSYS RedHawk and ANSYS Q3D Extractor, NXP engineers designed a new IC chip for a digital automobile radio with a 75 percent footprint reduction, lower costs and superior sound quality.

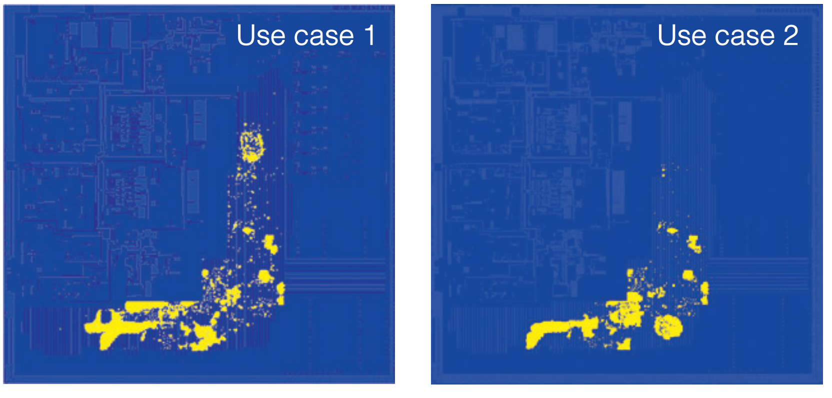
Once both of these approaches have been implemented (passive isolation and noise tolerance/noise minimization) to the best of the designer's ability, the team tests the resulting noise profiles for each design using simulation. The engineers at NXP used an IC block floorplan consisting of two radio-frequency/analog tuners, a digital processing block, an analog-to-digital converter (ADC) and digital-to-analog converter (DAC) block, a processor unit block, and a package model for the simulation.

SIMULATING NOISE GENERATION AND PROPAGATION

Because the inductance (L) of the semiconductor package is important —



▲ Use case 1 (red plot) shows natural switching of the cells, with most of them occurring at about 9,000 picoseconds, causing a large current spike. In use case 2 (blue), cells that were preprogrammed to switch over a longer time interval result in a smaller current spike due to fewer cells switching simultaneously at about 22,000 picoseconds.



▲ Programming the locations of switching cells (yellow) for two use cases can affect noise propagation to a small degree.

the L di/dt effect is dominated by package inductance — NXP engineers used ANSYS Q3D Extractor to extract the package model for the analysis. They used CSE, the substrate analysis kernel for the solver engine, that is an add-on to ANSYS Totem. Inputs for the simulation included the tech file, which contains images of the metal layers, dielectric constants and other physical properties of the design; netlist, which describes how the IC components are assembled; GDSII layout, which contains the geometries; and substrate tech file, which contains the foundry-supplied substrate physical properties. Another input type was the digital block models contained in the RedHawk power library; for each digital cell, this model contains the current waveforms when the cell is switching as well as the effective series capacitance and resistance when the cell is not switching. Engineers then input the digital vector data in the voltage change dump (VCD) file to describe the switching of the actual cells being analyzed.

For each use case, 200,000 digital cells were included in the model. In the initialization step, all inputs were processed and formatted for RedHawk. Next, the power and ground (PG) network and substrate extraction were done to account for all propagation paths in the system. Finally, RedHawk performed the noise coupling analysis.

The simulation portion of this analysis took less than 60,000 MB and a maximum time of approximately 175 minutes.

TAKING THE NOISE OUT

Noise maps and waveforms are the primary results of the simulation. ANSYS RedHawk Explorer displays the power profiles, showing average power over time, for all cycles of the simulation. RedHawk automatically selects the worst power cycle in the VCD file, so it is easy for engineers to see if their design meets specifications even in the worst-case scenario.

Simulation results helped to identify voltage drop scenarios in the system. Graphs of demand current and battery current over time reveal the spikes in current for two different use cases. Measuring the noise generated naturally by the IC and when the gate switching is preprogrammed to occur over a wider time interval gives engineers valuable optimization data.

The current spikes correspond well with the times when the most digital gate switches occurred. Spreading the gate switches out in time reduces the noise propagation level in the combined digital–analog IC chip.

The programming of digital circuitry also affects which gates switch by location on the IC. This can help to reduce the noise levels since the distance between the digital switching gate "aggressor" and the analog "victim" might change. This effect, however, is less prominent compared to the timing changes.

So what can be done to minimize noise propagation in a digital radio IC chip? Because engineers have a programmable digital block, they can control the number of cells that switch at a given time. By minimizing simultaneously switching circuits, they can reduce the current spike and, hence, the voltage drop in the system that leads to noise propagation from the digital side to the analog side of the IC.

Using ANSYS RedHawk and ANSYS Q3D Extractor, NXP engineers designed a new IC chip for a digital automobile radio with a 75 percent footprint reduction, lower costs and superior sound quality. But they are not stopping there. The team is already designing its next product, and with ever-shorter design times, it could be on the market in short order, helping NXP retain its position as global leader in ICs for digital automobile radio applications.

COMING CLEAN

To meet China's tough new emissions requirements, Tenneco used ANSYS CFD to optimize the design of a new selective catalytic reduction system.

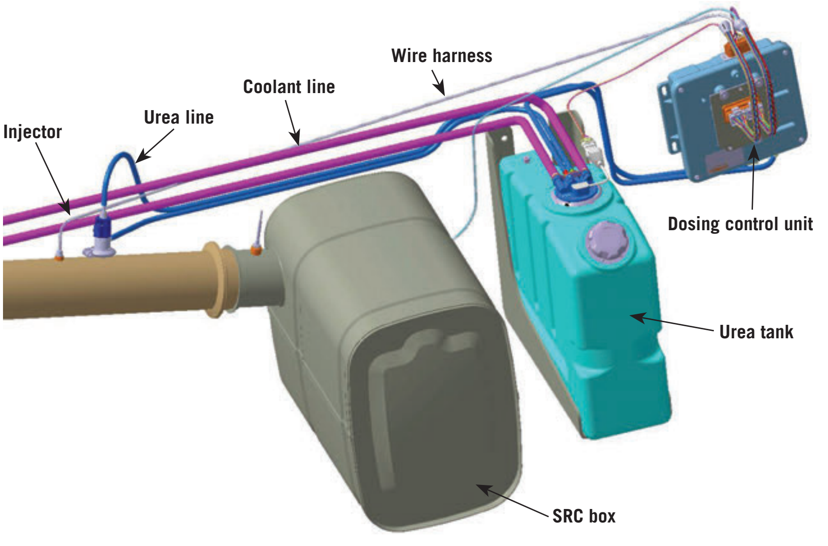
By Zhiguo Zhao, CFD Group Supervisor, Tenneco China Research Center, Shanghai, China

Although heavy trucks account for only about 5 percent of vehicles on China's roads, they account for about 80 percent of nitrogen oxides (NOx) emissions. China recently implemented demanding Stage IV emissions requirements that are nearly identical to the Euro 4 emissions standards. The new requirements reduce NOx emissions to 3.5 grams per kilowatt-hour (g/kWh), compared to 5.0 g/kWh in Stage III standards. Selective catalytic

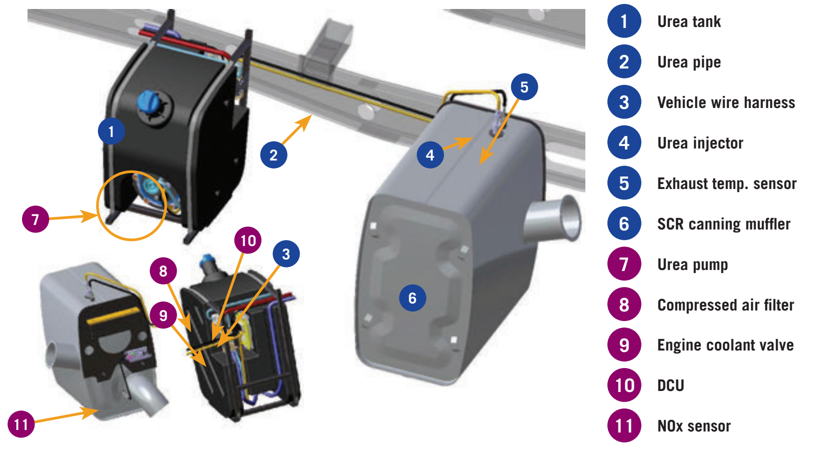
China recently implemented demanding Stage IV emissions requirements.

reduction (SCR) has become the go-to technology for meeting tough diesel-engine NOx emissions requirements. SCR technology involves injection of urea, a reducing agent, into the exhaust stream. The exhaust stream then flows through a catalyst, resulting in the reduction of nitrogen oxides into nitrogen, water and tiny amounts of carbon dioxide, all of which are natural components of air.

In traditional SCR design, the urea injector is mounted to the exhaust line leading into the SCR box that holds the catalyst. Tenneco — one of the world's leading designers, manufacturers and distributors of clean-air and ride-performance products and systems for the automotive, commercial truck and off-highway markets and the aftermarket — has developed a new injector-integrated SCR design in which urea is injected directly into the SCR box through an internal mixing pipe. The new design reduces the cost of the inlet pipe and improves design flexibility by eliminating restrictions on inlet pipe length and geometry. Tenneco engineers faced the challenge of optimizing the new design to eliminate the tendency for urea to build up on the mixing pipe in the SCR box. This buildup limits the life of the unit. They used ANSYS Fluent computational fluid dynamics (CFD) software to simulate the complex SCR physics, including droplet evaporation, droplet–wall interaction and NOx reduction. Engineers evaluated multiple design iterations and optimized the SCR design, eliminating urea deposits and meeting other design requirements.



▲ Traditional SCR design



▲ New design with injector fixed at SCR box

Engineers evaluated multiple design iterations and optimized the SCR design, eliminating urea deposits and meeting other design requirements.

SCR DESIGN CHALLENGES

SCR technology involves injecting aqueous urea into the exhaust system. Urea decomposes into ammonia and isocyanic acid and reacts with NOx on the surface of the SCR catalyst. A good SCR system has high NOx conversion efficiency, low urea consumption, long life, and no or little ammonia slip. Ammonia slip refers to ammonia that passes through the SCR without reacting with NOx. The first of two critical phases in the SCR operation is the mixing of ammonia and exhaust gases to achieve a uniform mixture prior to exposure to the catalyst. The second critical phase is the catalytic NOx reduction to optimize NOx conversion while minimizing ammonia usage and slip.

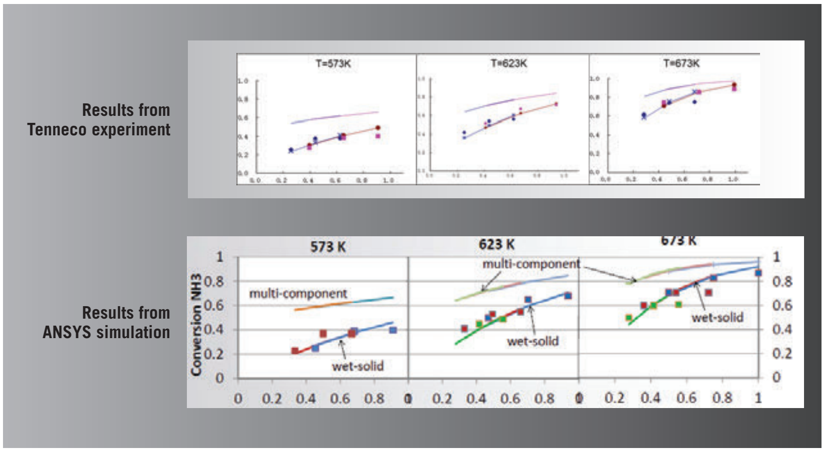
CFD modeling is used in the design process to ensure that a proposed SCR design delivers the required level of NOx reduction over the full operating cycle of specific engine models. The complex physics involved in SCR operation, including the interaction of gas and liquid phases, complex chemical reactions and spray–wall interaction, make it difficult to simulate. Another challenge is

Tenneco engineers have successfully used CFD to optimize the design of SCR systems, helping the company to continually improve its products and competitive position.

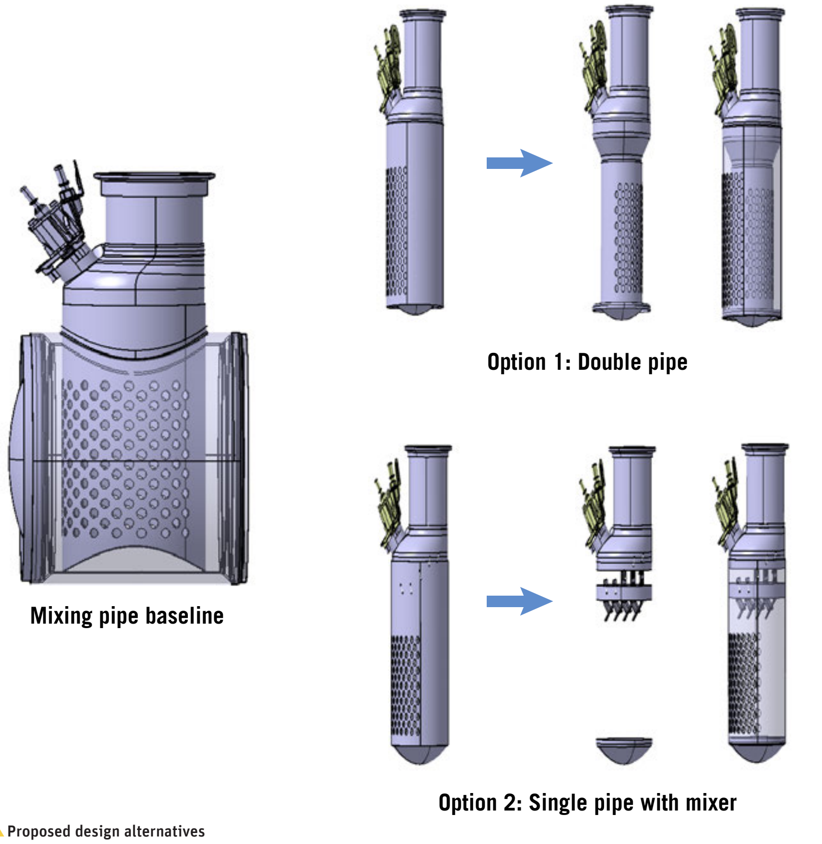
AUTOMOTIVE EXHAUST AFTERTREATMENT SIMULATIONS
ansys.com/92clean

the wide range of time scales involved: Spray dynamics is measured in milliseconds, wall film formation in seconds and catalyst transients in minutes. In addition, there is a scarcity of validated physical models, lack of established simulation best practices, and the legislated need to validate design performance over test cycles that often last 30 minutes or more.

A typical SCR design includes the urea delivery system, SCR box and dosing control unit. The injector is traditionally placed at the inlet pipe. In Tenneco's injector-integrated SCR design, the injector is fixed at the SCR box. The new design provides a significant cost reduction by avoiding the need to use high-grade steel for the inlet pipe. Also, by not mounting the injector on the inlet pipe, engineers can be flexible with their inlet pipe design, making it easier to adapt the SCR to different engine and truck configurations. On the other hand, in the new design, the mixing pipe extends into the SCR box; it must be constructed so that urea does not deposit on the pipe and lead to premature failure of the SCR system.



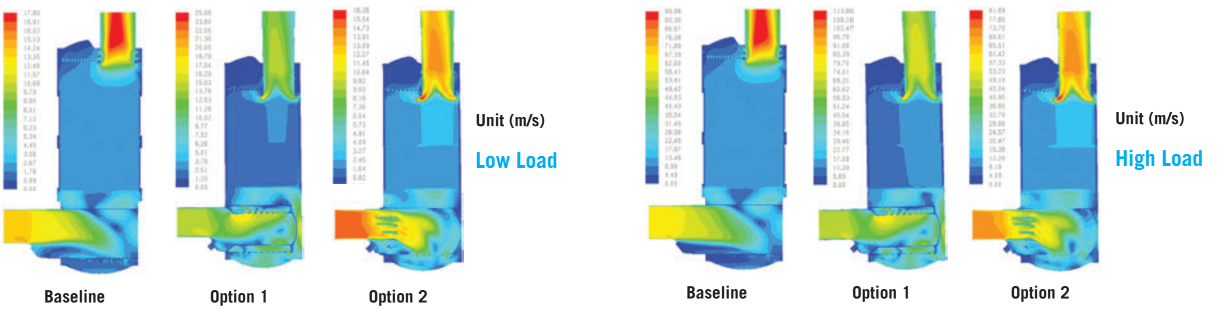
Results from the ANSYS CFD simulation (bottom) closely matched experimental results from Tenneco (top) using the urea reaction model.



Proposed design alternatives

VALIDATING CFD FOR SCR DESIGN

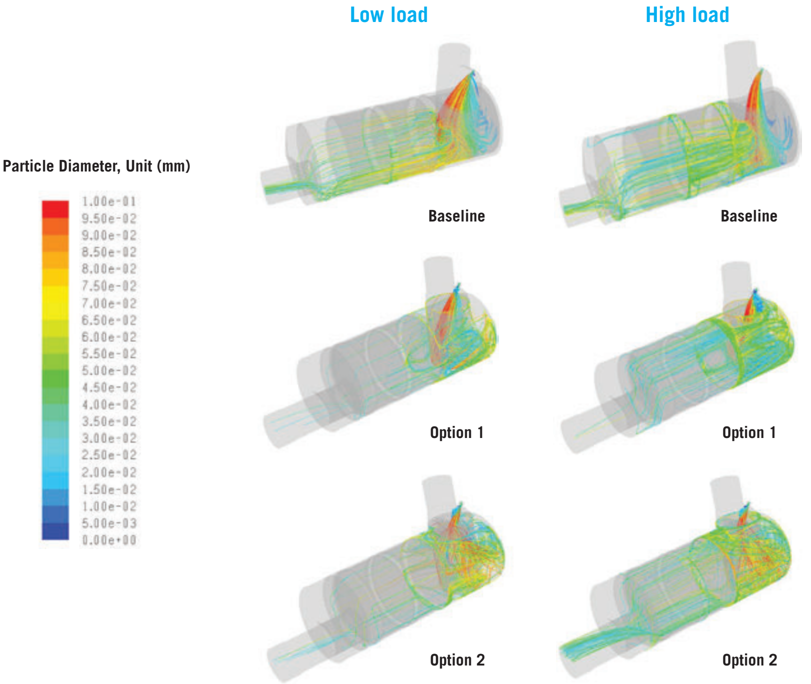
Prior to analyzing the new design, Tenneco engineers validated the accuracy of their simulation methods by performing physical experiments and simulating the test setup for CFD. Individual urea droplets were suspended on the tip of a fiber in a heated environment. Images of the evaporating droplet were recorded with high-speed cameras. The droplets were 32.5 percent urea by weight. Measurements were conducted while changing droplet ambient temperature by 50 K. In the CFD simulation, the droplets were held in the same position for the duration of the evaporation process. The evaporating droplet diameter and time history were evaluated. A simple 3-D planar mesh was created, and the droplet was positioned at its center. Multi-component and wet-solid



▲ Velocity contours under two load points

material properties were selected from Fluent's property database. The simulation results closely matched the experimental measurements.

Tenneco performed another droplet evaporation experiment based on work by J.Y. Kim in spray-induced mixing and thermal decomposition of urea solution in an SCR system. An aqueous solution containing 40 percent urea by weight was injected in the hot gas stream flowing inside a circular duct and was converted to ammonia. The ammonia concentration was measured at three downstream sampling points to determine conversion. CFD simulation results closely matched the experimental measurements.



▲ Particle trajectories

▲ Urea deposit in bottom of pipe in original design (left). New design eliminates urea deposits (right).

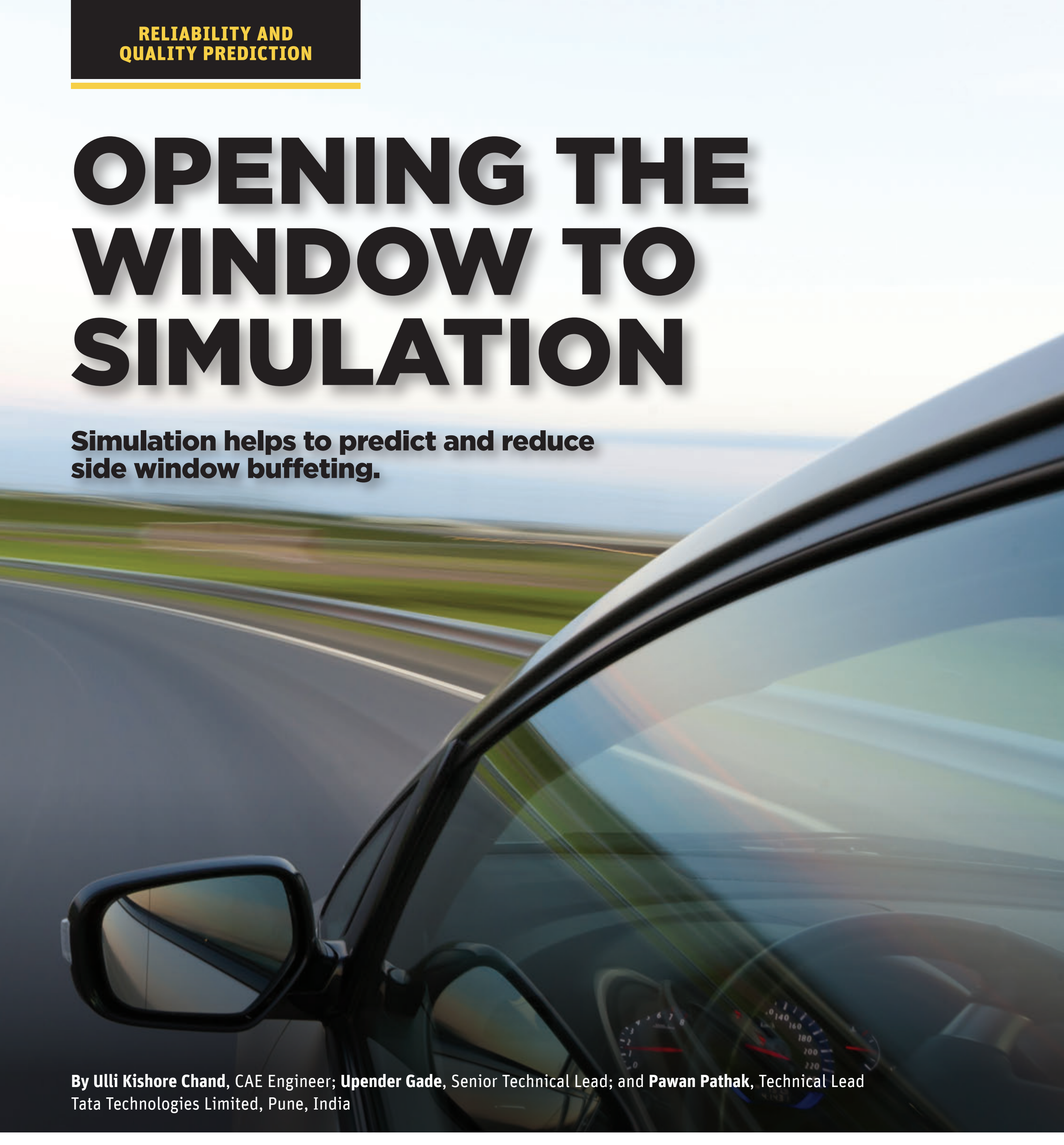
OPTIMIZING DESIGN OF NEW SCR

Tenneco engineers recently designed an injector-integrated SCR for a 12-liter diesel engine. The prototype of the original design showed urea deposits at the bottom of the pipe during the specified cycles run for emissions testing. Tenneco engineers proposed two new designs as potential solutions to the urea deposit problem. One design used a double mixing pipe with two pipes concentric to each other; the other design used a single pipe with a static mixer near the inlet.

Tenneco engineers used ANSYS CFD to simulate the baseline, double pipe and mixer cases. CFD results showed the velocity contours and particle trajectories for the three cases. Based on a comparison of the results from the three cases, Tenneco engineers felt confident that the double-pipe design would solve the problem. They built a prototype of this design and found that it eliminated the urea deposits. This case is one of several in which Tenneco engineers have successfully used CFD to optimize the design of SCR systems, helping the company to continually improve its products and competitive position in the heavy truck market through innovative engineering.

Reference

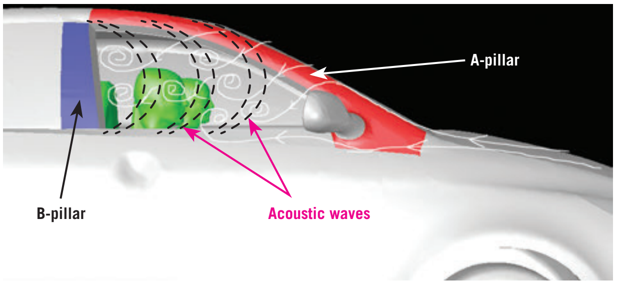
Kim, J.Y.; Ryu, S.H.; Ha, J.S. Numerical Prediction on the Characteristics of Spray-Induced Mixing and Thermal Decomposition of Urea Solution in SCR System. Proceedings of ICEFA04. *ASME 2004 Internal Combustion Engine Division Fall Technical Conference*. **2014**. pp. 165–170.



OPENING THE WINDOW TO SIMULATION

Simulation helps to predict and reduce side window buffeting.

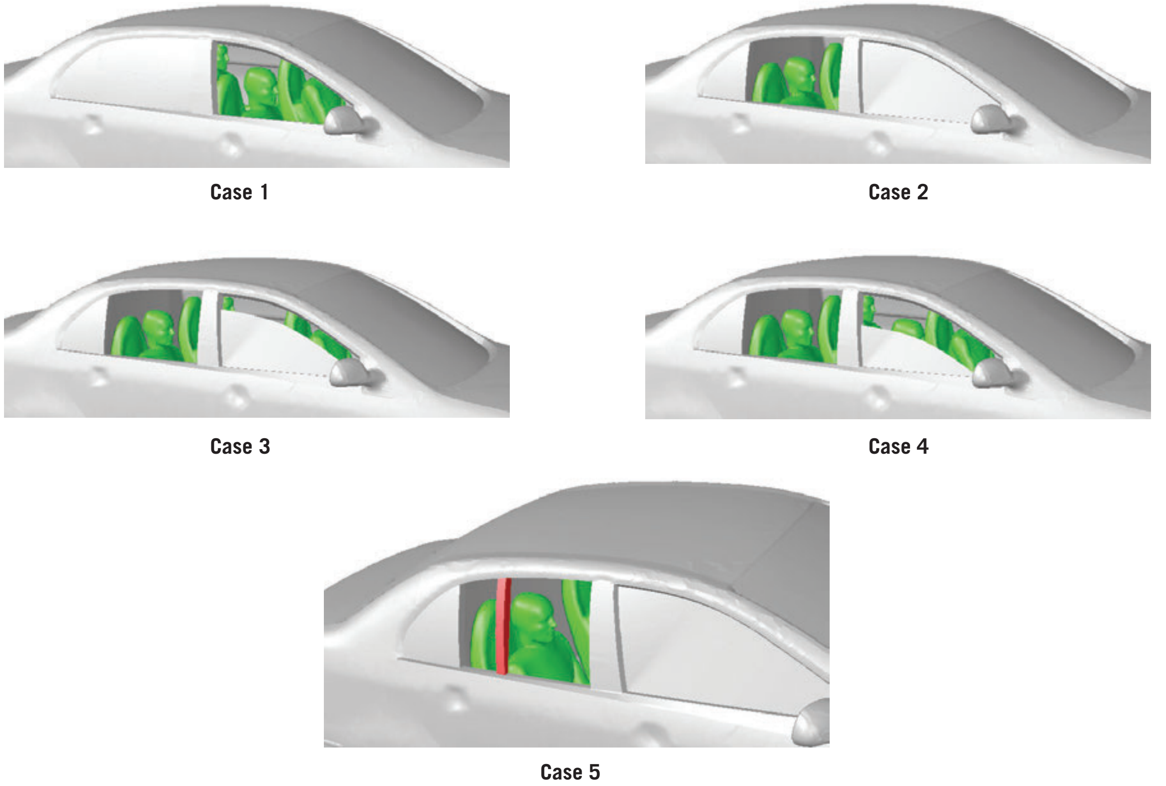
By Ulli Kishore Chand, CAE Engineer; **Upender Gade**, Senior Technical Lead; and **Pawan Pathak**, Technical Lead
Tata Technologies Limited, Pune, India



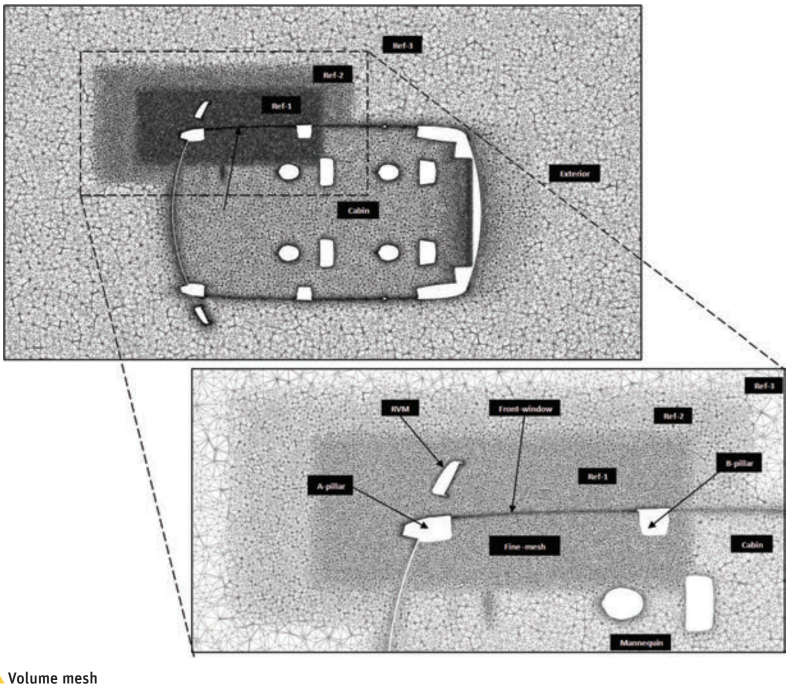
▲ Wind buffeting can intensify due to flow-acoustic feedback.

Imagine that while driving you open the window to let in cool, fresh air. But soon, you feel uncomfortable pulsations and hear noise, called wind buffeting or wind throbbing. Other sources of vehicle noise have been reduced, so automotive engineers are spending more time and effort addressing wind buffeting. Traditionally, engineers have built and tested each design to determine how it performs with regard to wind buffeting. However, Tata Technologies engineers are now accurately simulating wind buffeting with ANSYS Fluent computational fluid dynamics (CFD) software, making it possible to evaluate many different designs without the time and cost involved in building a prototype.

CFD makes it possible to quickly and inexpensively evaluate possible corrective measures and provide diagnostic information.



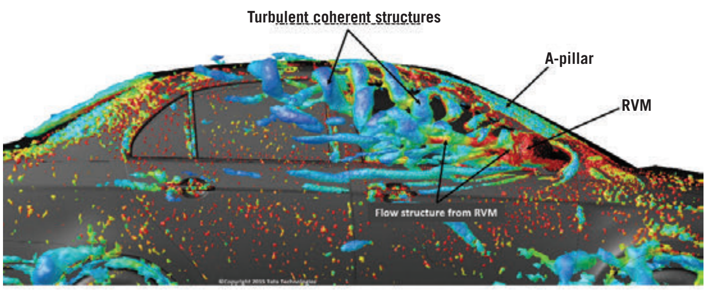
Five wind buffeting cases studied by Tata Technologies



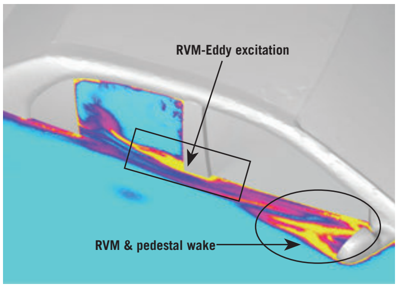
Volume mesh

ADVANCES IN CORE CFD TECHNOLOGY: MEETING YOUR EVOLVING PRODUCT DEVELOPMENT NEEDS
ansys.com/92window

Wind buffeting typically occurs when an unstable shear layer is established at the upstream edge of the window opening, along the A-pillar in the case of front-side window buffeting. Vortices are shed from this location and travel downstream along the side of the vehicle to the structure. When these vortices reach the B-pillar, they interact with the B-pillar to generate acoustic waves that propagate inside and outside the passenger compartment. When the forward-moving waves reach the A-pillar, they trigger more vortices that move back to the B-pillar. At certain travel speeds and for certain window and cabin geometries,



▲ Turbulent structure visualization around A-pillar



▲ Flow visualization shows the importance of the rear-view mirror in exciting wind buffeting in case 2.

this process generates self-sustaining oscillations that can create large pressure variations in the cabin that are uncomfortable and annoying for vehicle passengers.

LIMITATIONS OF PHYSICAL TESTING

To increase passenger comfort, automotive manufacturers have been steadily reducing structure-borne and airborne noise. Vehicles now operate much more quietly, but this makes wind buffeting more noticeable than it had been in the past. The frequency of wind buffeting is often below the range that can be heard by human ears, yet passengers can still experience an unsettling fluctuating force. Engineers measure these pressure fluctuations generated by wind buffeting with microphones. They then typically perform Fourier transform analysis on the analog signal to convert it to an acoustic spectrum in which amplitude is plotted as a function of frequency to better understand the causes of buffeting and its effects on vehicle passengers. These measurements can be performed only relatively late in the design process when vehicle prototypes have been created. At this point, many design decisions have been made, so changes are expensive and run the risk of delaying product introduction. Physical testing and design validation are conducted in anechoic wind tunnel facilities, which are expensive and complex to build and maintain. Sometimes they provide only very rudimentary diagnostic information, so engineers must rely upon intuition and experience in developing alternative solutions.

APPLYING SIMULATION TO WIND BUFFETING

Working with leading automotive original equipment manufacturers, Tata Technologies engineers are using numerical methods to simulate wind buffeting in the early stages of the product development process, long before prototypes are available. In the example shown here, engineers used CATIA® computer-aided design (CAD) to create five models:

- **Case 1:** Front window completely open, rear window closed
- **Case 2:** Front window closed, rear window completely open
- **Case 3:** Front window slightly open, rear window completely open
- **Case 4:** Front window halfway open, rear window completely open
- **Case 5:** Front window closed, rear window completely open, split pillar divided rear window

The objective of the study was to understand the influence of partially or fully open combinations of windows while cruising. A split-bar was installed in one of the cases to study the effect of altering window geometry. All cases were simulated at ambient conditions for vehicle speeds between 80 and 100 kilometers per hour. Virtual mannequins represent vehicle occupants, and other cabin details — like dashboard, seats, interior trim, etc. — were used to accurately model the cabin volume.

Tata Technologies engineers performed boundary meshing of each CAD model and then used ANSYS software to generate a volume mesh. Four stages of grid refinement were used with the finest mesh in the area around the window and rear-view mirror to resolve the turbulent structures and boundary layer. Buffeting is an inherently transient and complex turbulent phenomenon that involves generation and interaction of nonlinear turbulent eddies at the window corners. This poses a challenge in terms of computational resources.

A steady-state compressible Reynolds-averaged Navier–Stokes (RANS)/k-ε solution was used to initialize an unsteady large-eddy simulation (LES) solution. LES models resolve large turbulent structures in both time and space as well as simulate the influence of large-scale eddies that are responsible for generating acoustic sources on the window posts. The flow simulation calculated unsteady static pressure signals at four locations near

Tata Technologies engineers accurately simulate automotive wind buffeting, making it possible to evaluate many different designs without the time and cost involved in building a prototype.

THE RIGHT CFD SOLUTIONS FOR TODAY AND TOMORROW
ansys.com/92window2

each mannequin's right ear. This method is analogous to pressure extraction using microphones placed on a test dummy in wind tunnels.

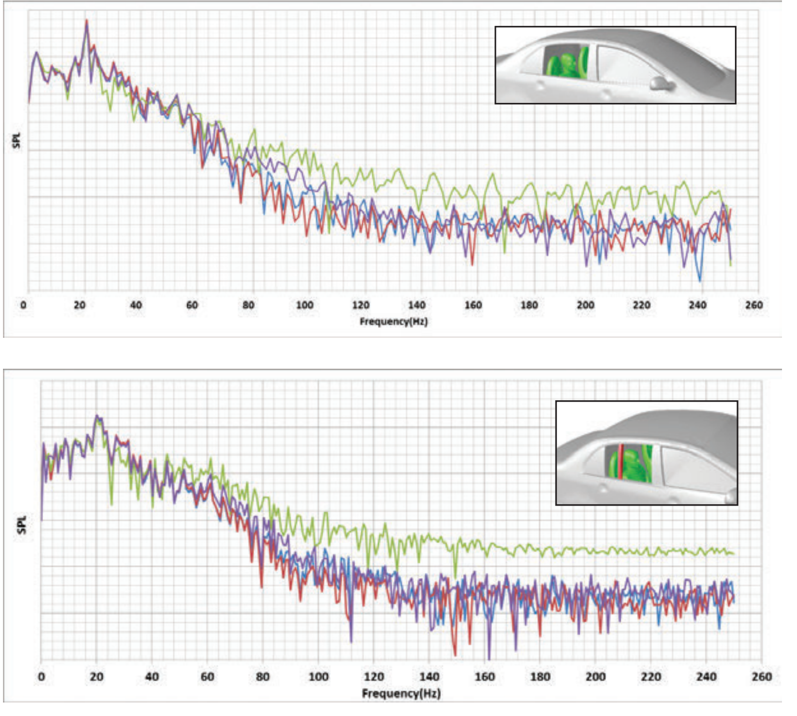
The Tata Technologies team post-processed each unsteady static pressure signal using the fast Fourier transform (FFT) tool. FFT converts the pressure signal from the time domain to the frequency domain. FFT enabled engineers to conduct spectral analysis that determined the peak sound pressure levels and frequencies at each virtual microphone location on the test dummy.

RESULTS USED TO IDENTIFY AND CORRECT WIND BUFFETING PROBLEMS

Buffeting studies on benchmark cases have shown that CFD results correlate well with wind-tunnel testing data. Similar modeling methodologies applied to the five case studies provided diagnostic information far beyond what could have been obtained with physical testing. For example, simulation results for case 2 showed that the side mirror and its pedestal wake excited the unstable shear layer at the B-pillar, reducing wind buffeting by about 6 dB with respect to a baseline case without a rear-view mirror. This demonstrates that protruding surfaces like the rear-view mirror and window visor offer opportunities to suppress buffeting. This information would have been difficult or impossible to determine with physical testing alone.

Power spectral density plots generated by the simulation showed that in case 2 much of the acoustic energy was concentrated at a frequency of about 20 Hz, which is close to the cabin's resonant frequency. Case 3 showed that this frequency slightly increased by 5 Hz when the front window was one-quarter open while the rear window was fully open, due to venting effects. Sound pressure level plots showed that the peak sound pressure level and the corresponding frequency was the same for all the four vehicle occupants although the overall sound pressure level varied from location to location.

Integrating simulation from the early stages of the design process helps the company to reduce wind buffeting and increase customer satisfaction.



▲ Sound-pressure level comparison between case 2 (top) and case 5 (bottom) shows the beneficial effects of the split pillar.

The results also showed that case 2 generated the highest sound pressure levels, between 120 dB and 130 dB, for vehicle velocities between 80 kph and 100 kph. Cases 3 and 4 showed a drop in sound pressure level of between 5 dB and 10 dB with respect to case 2, and case 1 produced the lowest sound pressure level. Reductions in sound pressure levels ranging from 8 dB to 12 dB for the different passengers in the vehicle were predicted with the split pillar in case 5 relative to the same configuration without the split pillar (case 2).

CFD makes it possible to quickly and inexpensively evaluate possible corrective measures; it provides diagnostic information that helps determine why a potential solution does or does not work without investing the time and cost of building a prototype. The ability to model acoustic generation/propagation and visualize the turbulent flow structures responsible for generating acoustic sources helps engineers to understand exactly why wind buffeting occurs in a particular case and assists them in identifying promising solutions. Tata Technologies engineers use CFD to investigate a wide range of possible design solutions to wind buffeting problems and to determine general guidelines for reducing wind buffeting. Integrating simulation from the early stages of the design process helps the company to reduce wind buffeting and increase customer satisfaction.

IT'S A SNAP

Audio system that uses snap-fit assembly

Valeo uses static nonlinear best practices to simulate snap-fits using ANSYS software.

By **K. Vaideeswarasubramanian**, Engineer, CAE; **Vinod Ryali Balaji**, Senior Engineer, CAE; and **Karthic Sethuraman**, Engineering Manager, Valeo India Private Limited, Chennai, India

Valeo produces many automotive components — such as smart antenna systems, smart keys, switches, mechanical control panels, thin film transistor (TFT) displays and electronic control unit (ECU) enclosures — that are secured and, in some cases, activated by snap-fits during the assembly process. In each case, the clipping and unclipping forces must be calculated, and risk of structural failure must be evaluated. This is achieved by performing static nonlinear simulation of snap-fits that includes multiple contacts with friction and thermoplastic materials. Valeo engineers have developed best practices for using ANSYS Mechanical in all stages of the simulation process, from geometry preparation to post-processing.

GEOMETRY PREPARATION

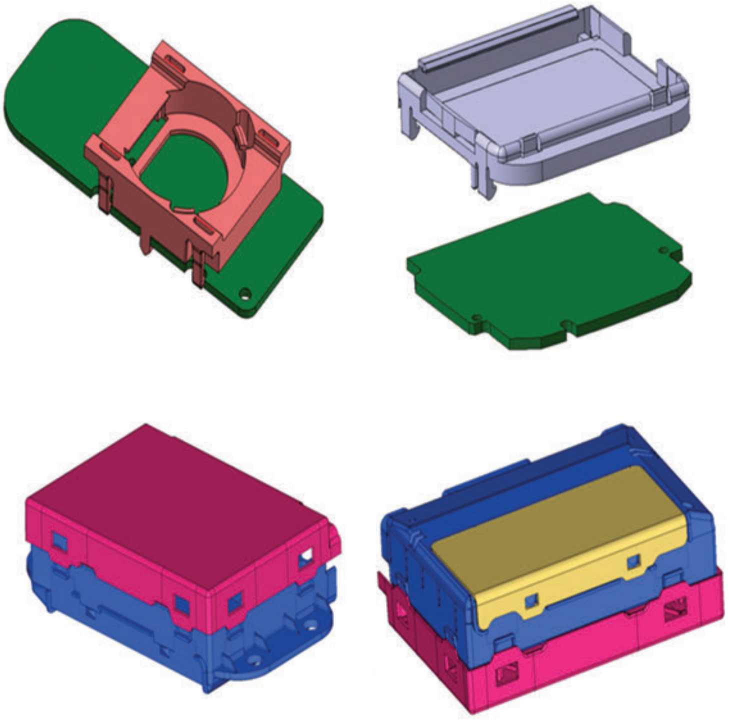
A sweepable volume has the same number of vertices per face and a smooth path from the source to the target face. One advantage of a sweepable volume is that it can be automatically meshed with hexahedron or brick elements that can fill a volume more efficiently. This leads to fewer elements and faster solution times. Another advantage is that brick meshes are more uniform, which provides greater accuracy. The option `Show - Sweepable Bodies` in ANSYS Meshing quickly identifies sweepable bodies within the assembly. Bodies that are not sweepable can be sliced into sweepable volumes and the `Form New Part` option can be used to ensure element connectivity between sliced parts.

Geometric features in the model with sharp edges close to the snapping region are common sources of nonconvergence. This problem can be addressed by adding small fillets to these specific contact regions in the simulation model.

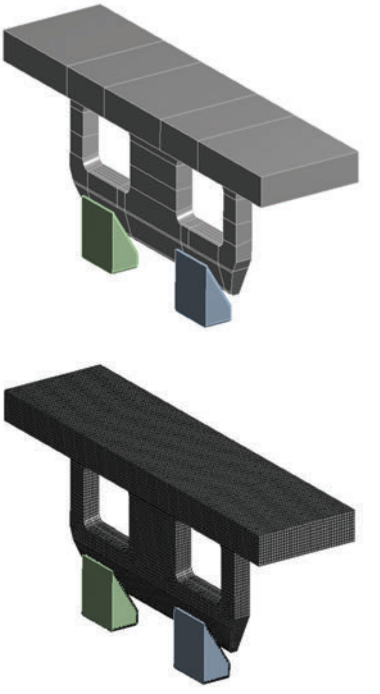
Engineers reduce computational time by defining the parts that are not of primary interest as rigid bodies, without having any significant effect on results accuracy.

MATERIAL MODELING

Thermoplastic material modeling is still much more of an art than a science, and each current method has limitations. One of the challenges is that the breaking point of many thermoplastic materials is not available in any number of commercial



▲ Typical snap-fit application



▲ Dividing geometry into sweepable volumes shown in top image. Meshed model shown below.

material databases. The absence of a breaking point can cause convergence difficulties. In some cases, Valeo engineers solved this problem by obtaining the breaking point of the material from the material supplier. However, when breaking point data is unavailable, extrapolation of the available stress–strain data is performed on a case-to-case basis to improve convergence.

A limitation of the finite element method is that when a small region of a model bears an excessive load, the elements in this region can become distorted, which has a negative impact on accuracy. The engineers avoided this problem by slicing the areas where high compressive stresses and strains occur, then assigning linear elastic properties to these slices to obtain better convergence. Generally, the results from a model with a small linear elastic region do not vary much from a nonlinear model. In addition, element distortion and resulting noise in the force displacement curve are usually eliminated.

CONTACT SETTINGS

When two or more clips are simultaneously activated in an assembly, convergence problems may occur due to contact chattering. Valeo engineers define the clips as a single contact-target pair to alleviate this problem.

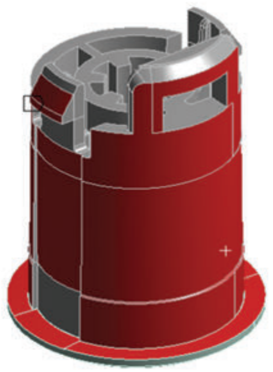
Co-efficient of friction values for interfaces in snap-fit assemblies are frequently not known. However, accurate friction values are often critical to achieve simulation results that correlate with physical tests. If the results do not correlate well with physical tests, the friction co-efficient is varied in the simulation until a good correlation is effectively achieved.

When multiple snap-fits are used in an assembly, the solution often does not converge beyond a particular point using frictional contacts. In this case, the team runs the solution until maximum force is obtained with a frictional contact. Accuracy is critical up to that point, because maximum force is often highly dependent on friction. The engineers then perform the complete simulation with a frictionless contact and use the results from frictionless contact only from that substep for which the solution with frictional contact did not converge.

MESHING

In some cases, problems such as generation of highly distorted elements may be experienced with a default surface mesh. These problems can be addressed by using the mapped face mesh option, in which the ANSYS software

Valeo engineers developed best practices for simulation of snap-fits using ANSYS Mechanical in all stages of the simulation process.



▲ Two clips defined as a single contact pair

maps a rectangular grid to a rectangular domain. The analyst can choose the number of divisions for each edge. The mapped face mesh option provides element shapes that are generally well within acceptable quality limits for the solver.

When converting geometry into sweepable volumes, it often turns out that there are some leftover areas that are not sweepable. In such cases, it is preferable to use a tetrahedral mesh. The hex-dominant mesh method should be used with great care, especially when high compressive strains on elements are expected.

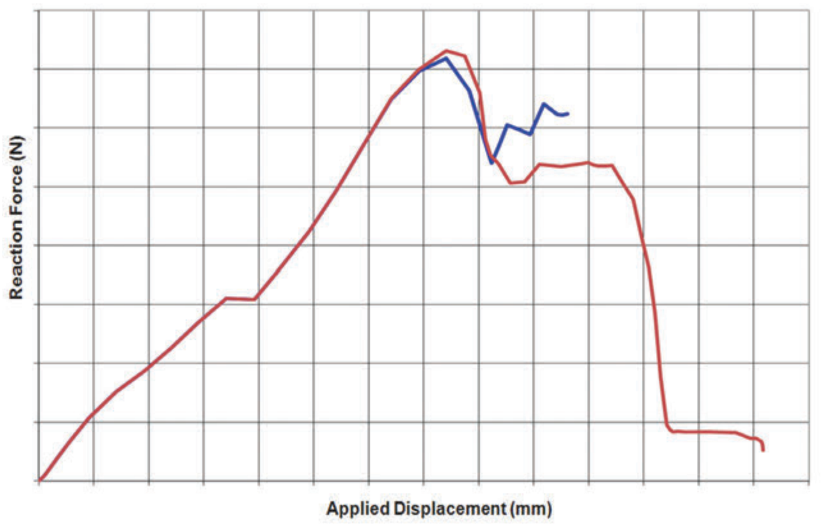
LOADING

Displacement control, rather than force control, usually provides better convergence in the snap-fit assembly.

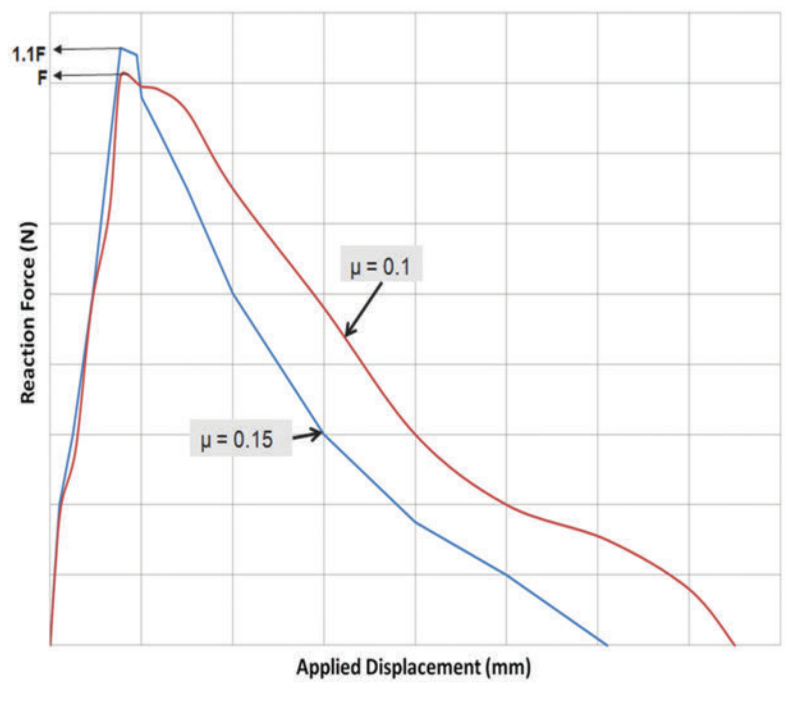
Many snap-fit assemblies experience large displacements for very small loads. Separating the loads into a number of small steps will aid in smooth convergence; it will also capture critical clipping points.

ANALYSIS SETTINGS

The distributed memory parallel solver for ANSYS Mechanical generally provides the fastest solution times. This solver decomposes the model into domains and sends each domain to a different core to be solved. A considerable amount of communications between the different cores is required. The results are automatically combined at the end of the solution. There are some cases, usually involving highly distorted elements and excessive strains, in which the distributed solver will terminate abruptly. In these cases, engineers use shared memory parallel solver.



▲ Blue line represents partially completed nonlinear solution. Red line is completed nonlinear solution with a local linear elastic region.

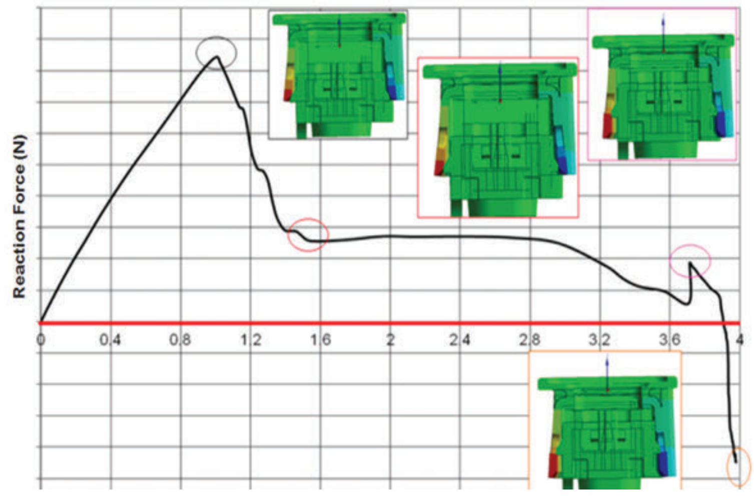


▲ Simulation results for different friction co-efficients. These results will later be compared to test results.

POST-PROCESSING

It is important to know the maximum force required for a snap-fit assembly; when multiple steps are involved, the force required in each step is also important. For better clarity, Valeo engineers overlay the corresponding deformed model alongside each peak in the reaction force curve.

Thermoplastics tend to be very strong in compression, so in most cases the results in tensile areas are most critical for the design process. However, if high stresses and strains occur in compression, there is the potential for plastic deformation to occur. In such cases, the compression results are treated on a



▲ Deformed shapes overlaid on reaction force curve showing multiple peaks

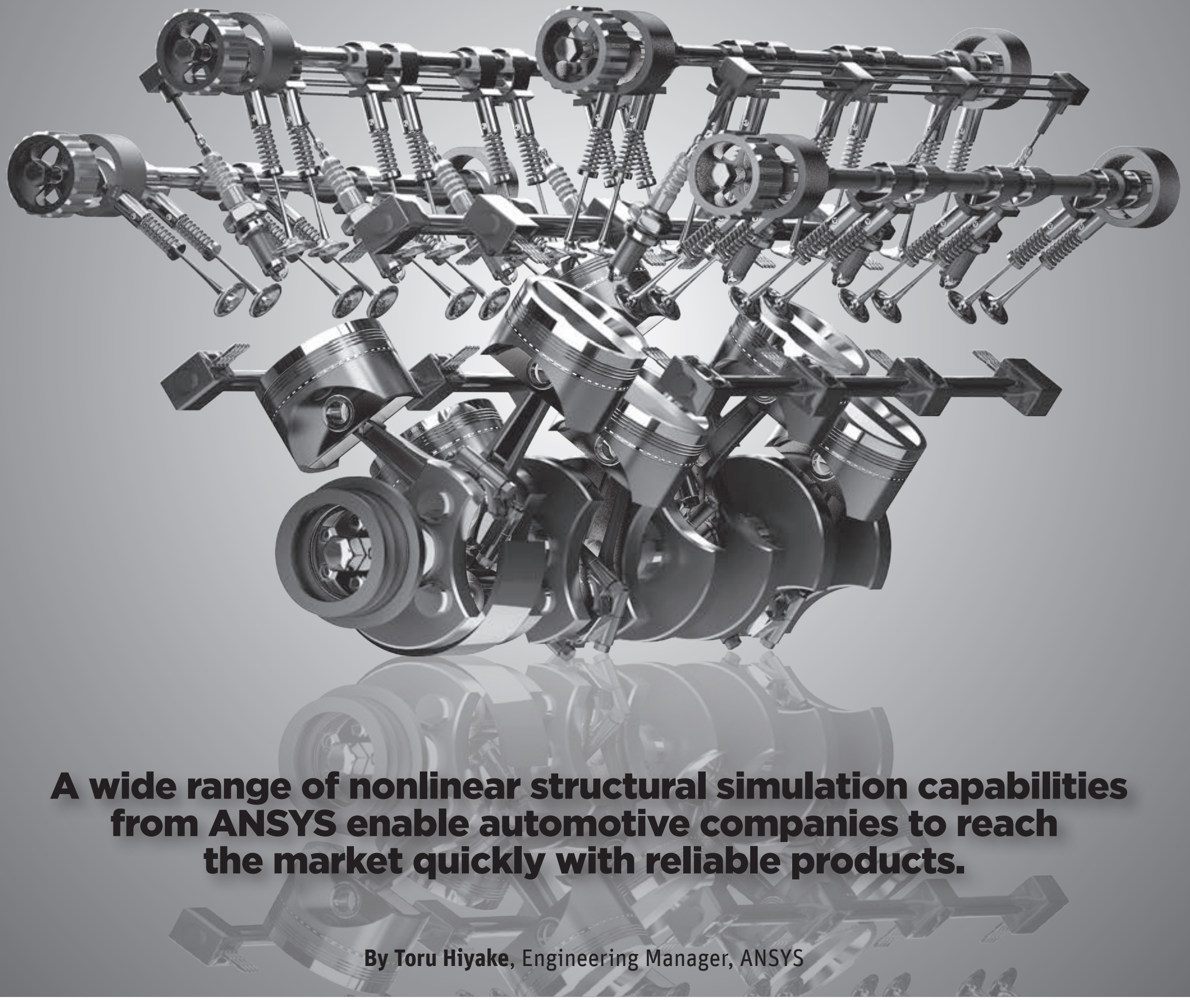
DIAGNOSING NONLINEAR STRUCTURAL SOLUTIONS IN ANSYS MECHANICAL
ansys.com/92snap

case-to-case basis depending upon the material and snap-fit design.

By using best practices and ANSYS Mechanical software for nonlinear simulation, Valeo engineers have confidence that their snap-fits will work reliably. Performing structural simulation very early in the design process helps to avoid costs associated with multiple prototypes, rework and changes to tooling. ANSYS high-performance computing has reduced simulation time by 50 percent, making it possible to complete the structural simulation for clipping and declipping processes in one week.

ANSYS HPC reduced simulation time by 50 percent, making it possible to complete structural simulation for clipping and declipping processes in one week.

AUTOMOTIVE DESIGN: ADVANCED NONLINEAR SIMULATION



A wide range of nonlinear structural simulation capabilities from ANSYS enable automotive companies to reach the market quickly with reliable products.

By Toru Hiyake, Engineering Manager, ANSYS

Automotive companies must remain competitive through innovation, so engineering teams are pressured to design and deliver quality parts quickly. To succeed, they must develop processes to finalize design from the drawing board to the assembly line in the shortest possible time. A key element in reducing development time is engineering simulation that allows companies to iterate designs to meet product requirements quickly by predicting real-life behavior in a virtual environment long before testing is required. Analysis using high-fidelity, proven capabilities allows engineers to cost-effectively determine performance

Automotive companies must develop processes to finalize design from the drawing board to the assembly line in the shortest possible time.

SMART STRATEGIES FOR STRUCTURAL SIMULATIONS
ansys.com/92strategies

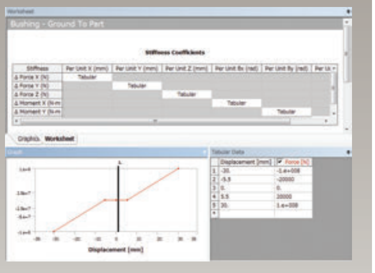
and reliability across a wide range of automotive applications.

The potential for simulation in the automotive industry is vast. The following represents only a fraction of the structural capabilities, applications and benefits of simulation for automotive engineers.

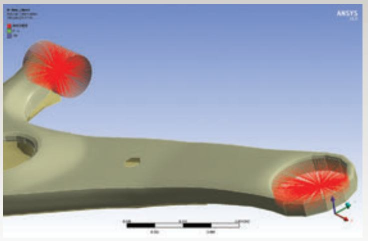
ANTI-VIBRATION RUBBER

Anti-vibration rubber has elastic and viscous characteristics. When applied to areas in which automotive parts are connected, it blocks vibration transmission. One method to model anti-vibration rubber is to employ the one-dimensional nonlinear spring-damper COMBIN14 element in ANSYS Mechanical. This element models the nonlinear behavior of rubber by directly using spring characteristics as well as data obtained through experiment. An example of how this technique can be used is a strength analysis of a suspension's lower control arm for which the nonlinear spring is defined at two bushing positions.

However, a detailed evaluation of the performance of an anti-vibration rubber product requires analysis of a three-dimensional model of the product. It is important to correctly identify characteristics of this nonlinear material to ensure an accurate and converged analysis. To characterize the rubber material, ANSYS Mechanical offers more than 10 constitutive hyperelastic models (including the Mooney–Rivlin and Ogden models). The parameters included in these material models can be automatically determined from experimental data using provided curve-fitting tools. To overcome possible convergence difficulties, a special lower-order tetrahedron element that incorporates a mixed u-P method (for handling both displacement and pressure as variables) is available in ANSYS Mechanical for use with nearly-incompressible rubber materials. Employing this element facilitates solving large rubber deformation and complicated contact conditions that would be time-consuming or more difficult using other element types.

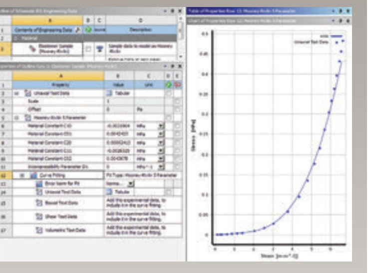


Defining nonlinear spring characteristics

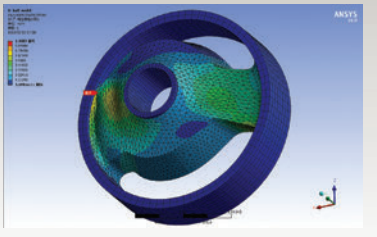


Lower arm bush mounting simulation

▲ Using one-dimensional nonlinear spring-damper model to perform strength analysis of a suspension's lower control arm



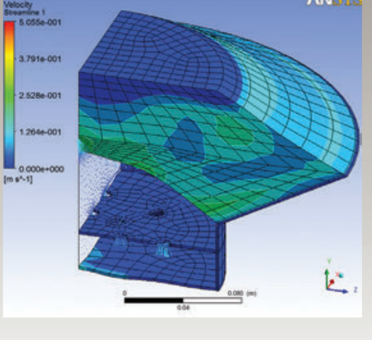
Curve fit window used to calculate parameters for material model



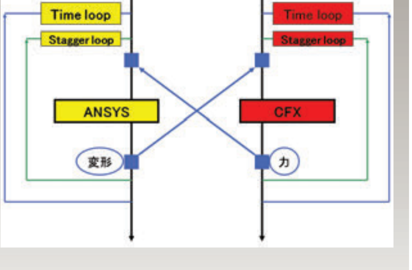
Strain contours for rubber bushing

▲ Detailed evaluation of anti-vibration rubber bushing

Some anti-vibration rubber products contain liquid that provides fluid resistance to optimize attenuation. An example is a liquid seal engine mount. Detailed analysis of such products must account for both structural and fluid characteristics. To help solve these problems, ANSYS offers powerful multiphysics technology that couples a structural solver (ANSYS Mechanical) and a fluid solver (ANSYS Fluent or ANSYS CFX). This technology enables a fluid–structure coupled solution (two-way fluid–structure interaction [FSI] analysis).



Flow velocity vector and strain contours for engine mount



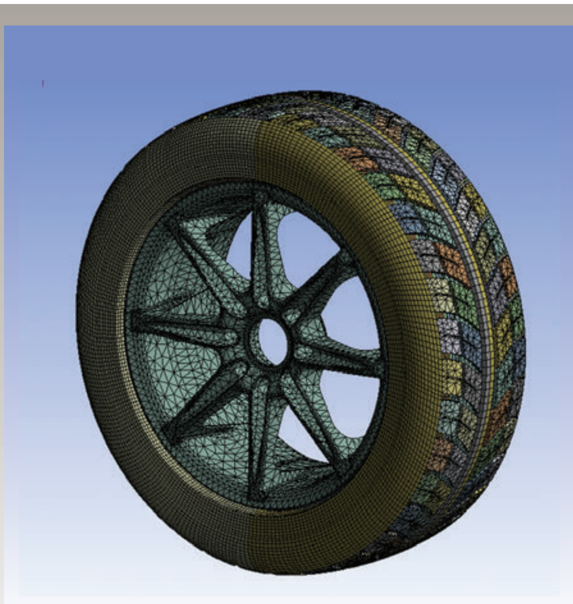
Two-way FSI coupling technology

▲ Multiphysics simulation of a liquid seal engine mount

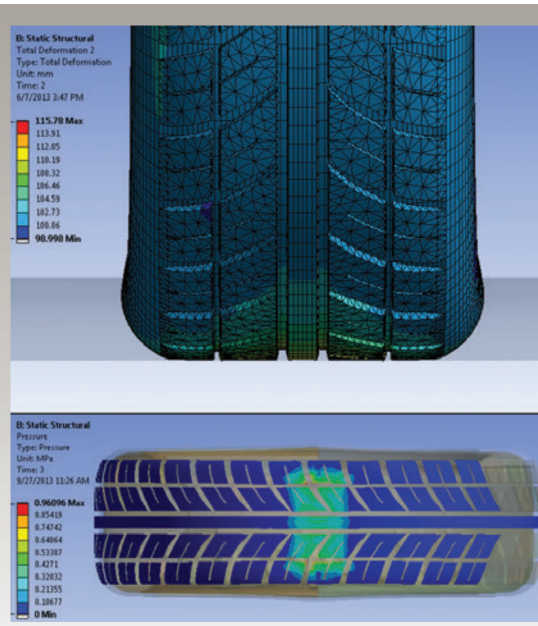
TIRES AND SUSPENSION

Tires are the only part of a vehicle that contact the ground, so they heavily influence an automobile's performance — from safety to the vibration/noise level that determines ride quality. Tire simulation requires modeling technology for complex internal structures. ANSYS offers special reinforcement elements (REINF263–265) to model the many reinforcement structures within a tire.

Pneumatic pressures at work within a tire are not always constant with



Wheel and tire model

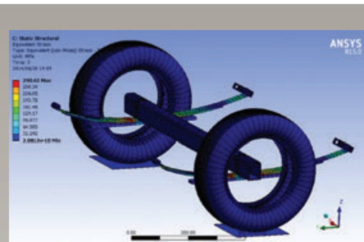


Deformation of contact patch (top) and pressure contours (bottom)

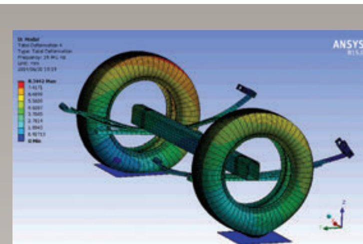
▲ Nonlinear analysis of tire's contact patch

respect to time because of the deformation caused by contact with the ground. These pressure variations due to geometric changes can be expressed by defining a hydrostatic fluid element (HSFLD241–242) for the tire.

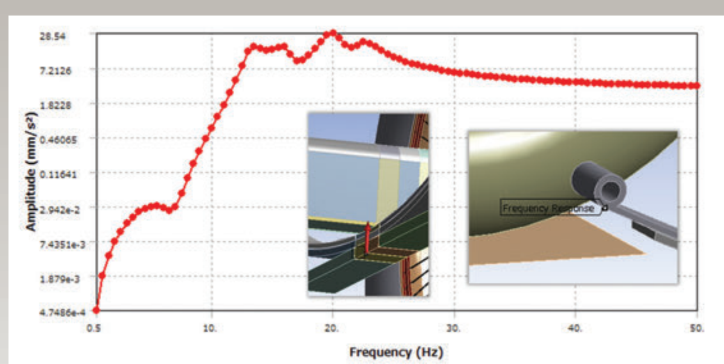
Suspension improves ride quality and steering stability. Together with the tires, the suspension is important in controlling the dynamic characteristics of a vehicle. The suspension fulfills its function only while the vehicle's weight is on it, so the characteristics in the initial state are important. For simulations like this, the ANSYS Workbench environment offers an analysis process called linear perturbation. To perform a static/dynamic analysis of a leaf spring suspension, first add the divided load of the vehicle weight to the suspension model and determine deformation due to the dead weight (static analysis). In this example, this stage consists of nonlinear analysis involving contact between leaf springs in addition to tire modeling as previously described. Next, perform eigenvalue analysis using the same model (modal analysis). All the initial conditions required for this stage, such as deformation due to dead weight and resulting initial stresses,



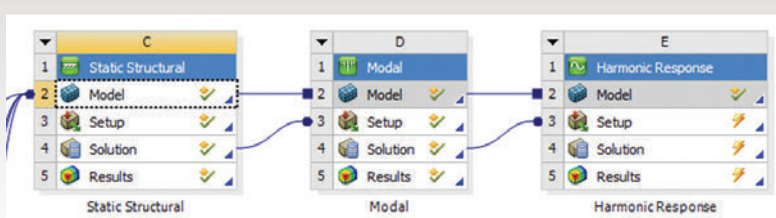
Nonlinear static analysis



Eigenvalue/modal analysis (including initial stress)

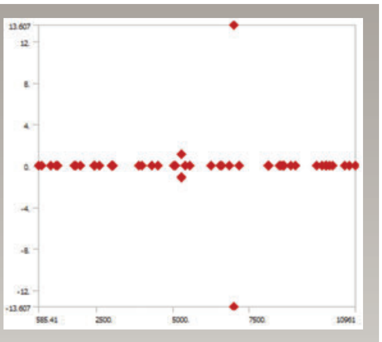


Frequency response analysis (using modes)

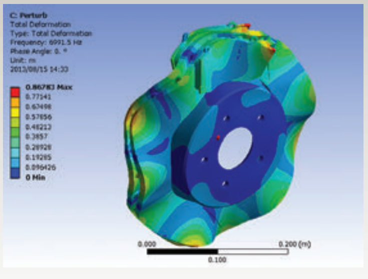


▲ ANSYS Workbench allows linking of the three types of analysis and passing data between them.

As simulations become higher fidelity and contain more physics, a solid HPC structure will facilitate solutions.

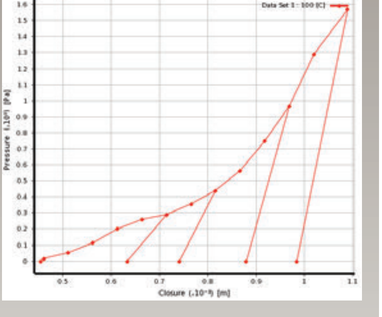


Root locus plot identifies whether motion is stable or unstable.

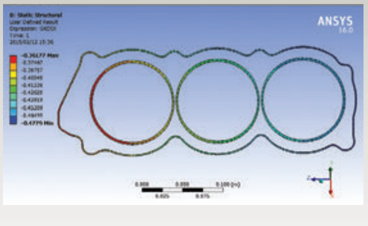


Unstable mode (complex eigenvalue)

▲ Brake squeal noise is related to frictional vibration, which can be predicted with simulation.



Gasket material model



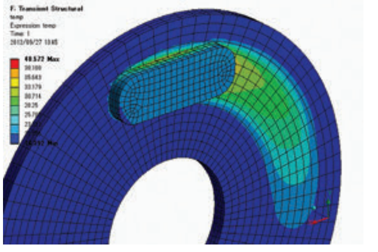
Gasket displacement contour

▲ Evaluation of behavior of gasket installed between cylinder block and head

$$q = TCC \times (T_t - T_c)$$

q: heat flux per area
TCC: thermal contact conductance
T_t: temperature of target surface
T_c: temperature of contact surface

▲ For frictional heating, a heating effect should be defined for a contact element.



▲ ANSYS Mechanical simulation of frictional heating of brake disc

$$w = \frac{K}{H} P^m V_{rel}^n$$

H: material hardness
K: wear coefficient
P: contact pressure
V_{rel}: the relative sliding velocity
m: pressure exponent
n: velocity exponent

▲ Rate of wear at contact node

▲ Simulation of abrasion loss due to friction of brake pads

are passed to the modal analysis simply by a drag-and-drop operation within Workbench. Finally, perform a modal frequency response analysis (harmonic response). Again, the setup information required for the frequency response calculation can be automatically passed from an eigenvalue modal analysis by dragging and dropping one system onto another.

BRAKES

Brakes are, of course, one of the most important safety features on a vehicle because they allow it to slow and stop. It is therefore vital that brakes perform as expected every time. A brake consists of a rotating brake disc and pads that grip it on both sides. Applying a brake generates heat, abrasion and squeal (noise), each of which are subject to design review by the manufacturer. ANSYS Mechanical enables one or more of these phenomena to be modeled using contact elements to ensure performance, safety and passenger comfort.

Frictional Heating

The use of coupled-field elements in ANSYS Mechanical (SOLID223, 226, 227) enables a single analysis that includes heating, heat transmission and structural deformation.

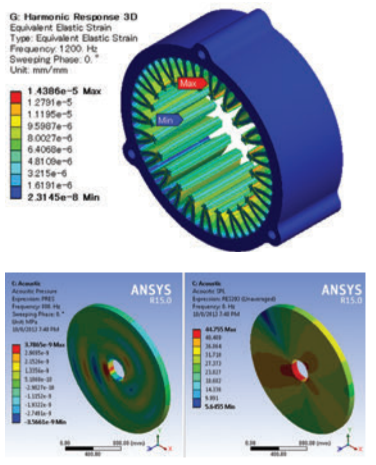
Frictional Wear

Frictional wear can be expressed by applying a wear model between contact elements. ANSYS offers the Archard wear model to calculate abrasion loss due to friction.

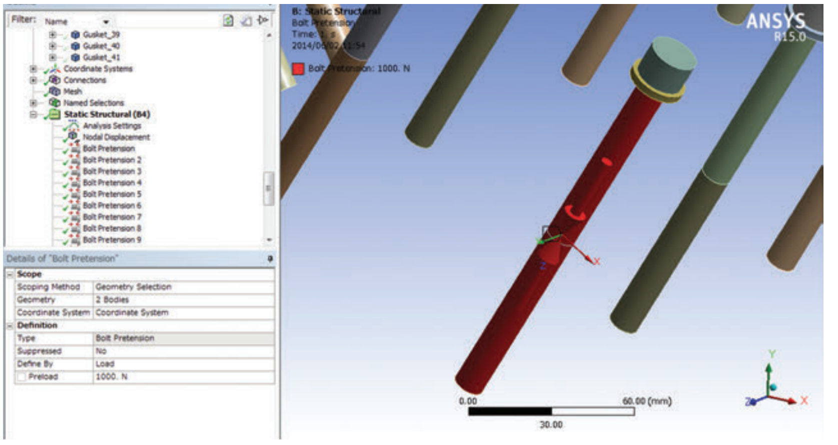
Brake Squeal/Noise

The generation of squeal noise is related to frictional vibration (self-sustained oscillation) that occurs between the disc and pads. One method to evaluate this vibration is via the stability criterion widely used in control theories. This criterion can identify whether motion is stable or unstable by solving a characteristic equation (complex eigenvalue problem). By setting a frictional co-efficient in the contact

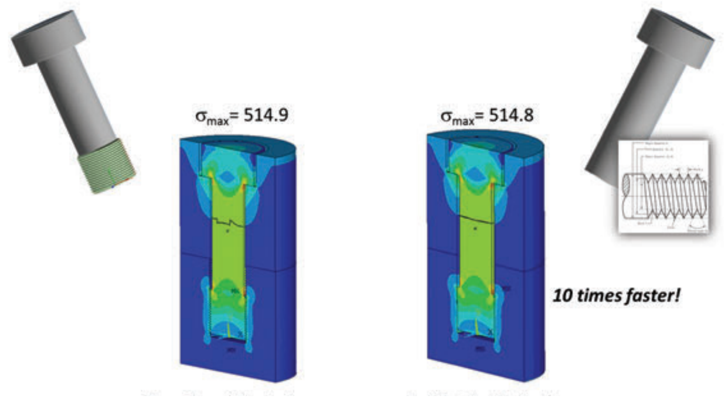
COMPONENT MODE SYNTHESIS IN ANSYS WORKBENCH SIMULATION
ansys.com/92nonlinear



▲ Motor vibration (top) and acoustic analysis (bottom)



▲ Bolt modeling in ANSYS Mechanical



▲ Using the bolt section method to model bolts can speed simulation time by a factor of 10.

region between the disc and pads, ANSYS Mechanical can automatically determine stable and unstable modes. Instability is often associated with noise, vibration and harshness.

ENGINE

An engine comprises many parts, and only a small fraction of the possible simulations can be covered in this article.

Gaskets

Gaskets make a structure airtight by acting in a compressive direction. They exhibit significant nonlinear characteristics from the compressive load. Special elements (INTER192–195) and a gasket material model within ANSYS Mechanical can be used to model a complex gasket assembly — for example, to evaluate behavior of a gasket installed between the cylinder block and the head.

Bolts

Bolts are commonly used to connect parts in a wide range of machinery and applications. For example, bolts connect the cylinder block and head (with a gasket between them). ANSYS Mechanical contains a bolt-modeling capability that can be easily configured to represent the tightening force and subsequent change in preload due to other external loads.

Although a bolt thread can be accurately modeled using contact, this method usually requires a significant increase in calculation cost due to the inclusion of minute details. ANSYS Mechanical offers technology to simulate the influence of the bolt thread simply by using thread pitch information.

OTHER PROCESS COMPRESSION CAPABILITIES

Component mode synthesis (CMS) is a substructure synthesis method in ANSYS Mechanical in which the flexibility of the model is retained, yet the number of degrees of freedom (DOF) is reduced. CMS analyzes large models as a set of interconnected components. Each one acts as a superelement that aggregates many individual elements from the initial model, thereby reducing degrees of freedom and solution time for large models. CMS can be used to accelerate analyses of large-scale models.

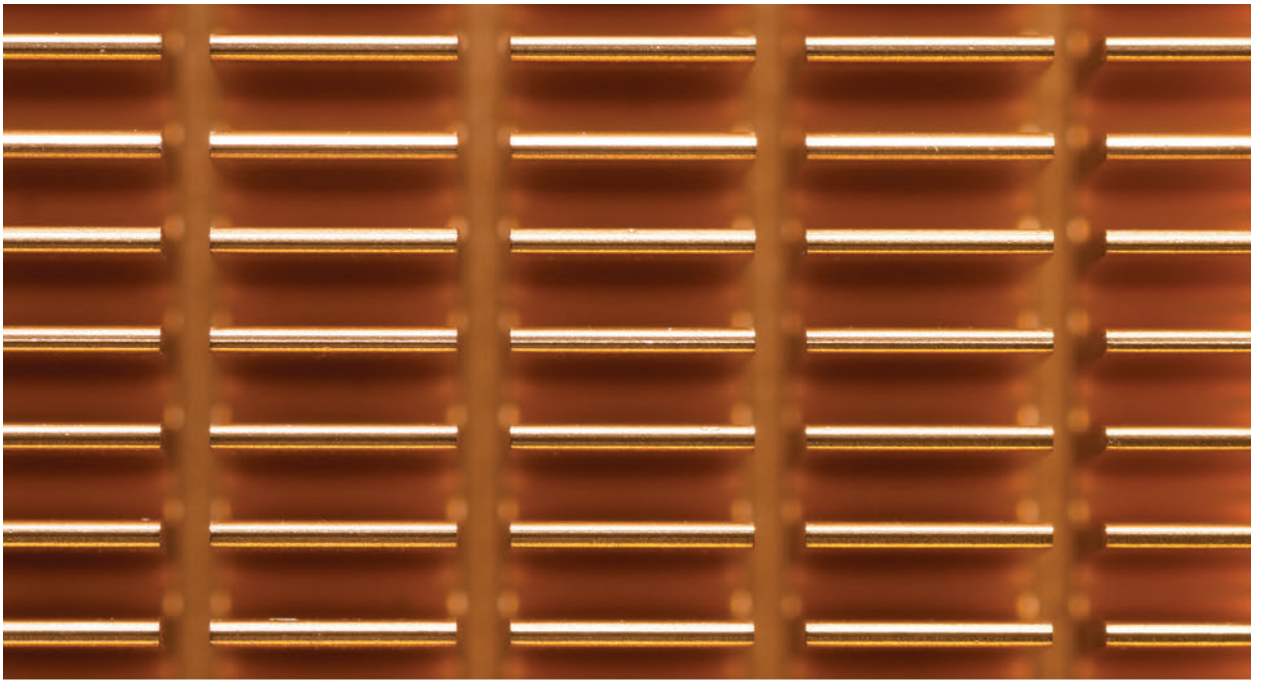
ANSYS Mechanical is also equipped with acoustic analysis capabilities that can be used for powerful electromagnetic field–structure–acoustic multiphysics coupled solutions when used in conjunction with ANSYS Maxwell electromagnetic analysis.

Structural simulation is only one way that companies can expedite the design process for automobiles. Fluids and electronic simulation are also required, and by using all these physics together on a single-platform multiphysics environment, full insight into real-world performance can be gained. As simulations become higher fidelity and contain more physics, a solid high-performance computing (HPC) structure will facilitate solutions. Also, automated parametric studies and multiple physics generate large amounts of data that must be managed and shared, so a specialized knowledge support system is a must. As the challenges to produce and deliver better and faster products in the auto industry accelerate, ANSYS delivers solutions no matter the scale or scope.

BEATING THE HEAT

Simulation helps an electronic product operate safely at a higher ambient temperature.

By Thierry Sin, VP of Sales & Marketing, Radian Thermal Products, Inc., Santa Clara, U.S.A.



Electronic assemblies must dissipate thermal energy from components to prevent premature failure due to overheating. Radian Thermal Products specializes in designing and building innovative custom heat sink designs that help to solve thermal problems. Recently, a manufacturer of telecommunications products asked Radian to help cool a pluggable card that was overheating. Radian engineers simulated the design with ANSYS Icepak electronics cooling simulation software. Using simulation, the team discovered that the fin density on the heat sinks in the original design prevented air from reaching downstream components and that variations in component height reduced the effective thermal conductivity. Engineers redesigned the cooling system using a larger number of smaller heat sinks attached to single components. They employed the ANSYS Workbench parameter manager to optimize the number of fins, fin thicknesses and other design parameters. The optimized design operates safely at ambient temperatures that are 20 C higher than the original design.

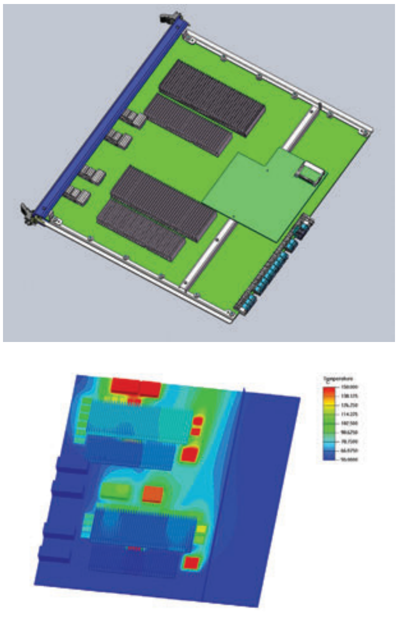
ORIGINAL CONCEPT DESIGN

A manufacturer of telecommunications products designed a pluggable card with several integrated circuits and heat sinks that fit into an Advanced Telecommunications Computing Architecture (ATCA) rack mount chassis. Simulation showed that the heat sinks cooled the chips they were attached to, but other components experienced excessive temperatures. Radian engineers began addressing this issue by obtaining information from the manufacturer such as the space available for heat sinks,

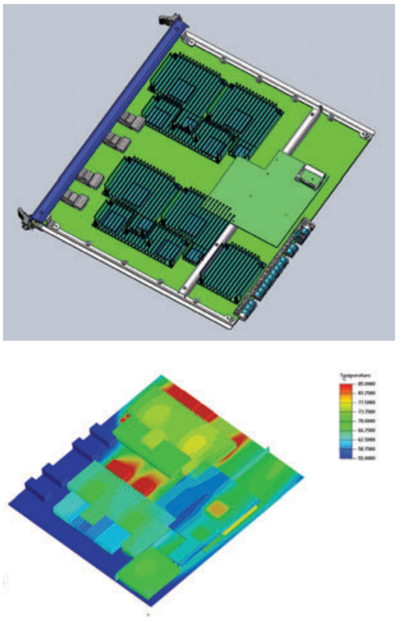
ANSYS Icepak helps Radian to quickly and efficiently diagnose thermal management issues and generate optimized designs.

dissipated power, power supply size, fan size and placement, target ambient temperature, attachment methods, and maximum allowable junction temperatures for the components. The maximum junction temperature is the highest internal temperature that a component is rated to withstand without damage. Traditionally, thermal management design is based on engineering experience and instinct. In the past, companies usually needed to build and test numerous prototype designs to understand and resolve thermal problems. Advanced simulation tools like ANSYS Icepak enable engineers to reduce costs and time to market by verifying a design's proof of concept with accurate thermal results prior to building a prototype. In this case, the manufacturer had already created a relatively simple heat-sink design and performed a thermal simulation with a non-ANSYS simulation tool. The simulation results showed that junction temperatures were too high but did not provide a clear path to resolve the thermal issues.

The optimized design operates safely at ambient temperatures that are 20 C higher than the original design.



▲ Original concept design and ANSYS Icepak simulation



▲ Radian's redesign and simulation

MODELING WITH ANSYS ICEPAK

Radian engineers are familiar with multiple thermal simulation tools but utilize ANSYS Icepak because of its powerful design optimization capabilities. The Workbench parameter manager makes it possible to run parametric studies and design of experiments. In addition, ANSYS DesignXplorer can be used with the parameter set to drive design of experiments, to carry out goal-driven optimization, and to perform Six Sigma analysis to investigate design robustness.

The manufacturer provided 3-D solid models of the original design to Radian engineers, who imported a mechanical computer-aided design (MCAD) model of the enclosure and an electronic computer-aided design (ECAD) model for the printed circuit boards into Icepak using CAD integration tools provided by the ANSYS Workbench environment. Engineers then created the simulation model geometry by dragging and dropping smart objects — such as fans, circuit boards, vents, openings, heat sinks and enclosures — into the imported geometry. They entered values to precisely match the geometric information, material properties and boundary conditions. Icepak generated a body-conformal mesh that represents the true shape of components and distributed the mesh appropriately to resolve fluid and thermal boundary layers. Radian customized the meshing parameters to refine the mesh and optimize the trade-off between computational cost and solution accuracy. ANSYS Icepak uses computational fluid dynamics (CFD) to determine the fluid flow and all modes of heat transfer — conduction, convection and radiation — in this case, for a steady-state thermal flow simulation. To reduce computational time, the team ran the solver in parallel on a multi-core machine. The solver yielded the fluid flow and heat transfer information for the entire simulation domain. The results were post-processed in ANSYS Icepak to visualize the airflow patterns and temperature distribution using velocity vector plots, temperature contours and fluid particle traces. Summary reports stated the calculated variable quantities. The post-processing capabilities available in ANSYS Icepak are critical to diagnose cooling performance.

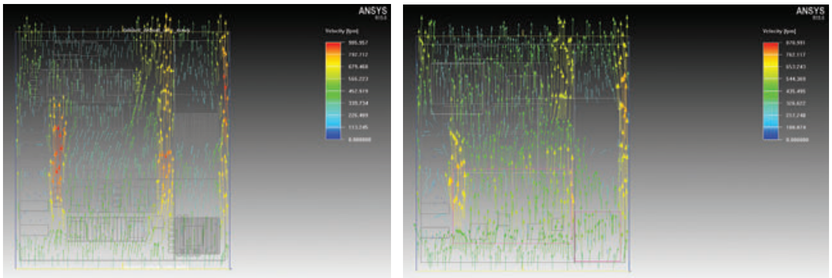
OPTIMIZING THE DESIGN

Radian's simulation results were similar to the manufacturer's simulation and showed junction temperatures up to 150 C

MULTIPHYSICS SIMULATION OF A PRINTED CIRCUIT BOARD
ansys.com/92heat

at an ambient temperature of 55 C. Based on these results, the board would need to be derated to an ambient temperature of 30 C. With this rating, the board could survive normal operating conditions in a telecommunications provider's central office but would likely fail if the ambient temperature rose due to a problem, such as an air conditioning outage. To enhance design robustness, the manufacturer wanted to ensure that the product could operate at an ambient temperature of 55 C.

The airflow patterns clearly showed that the original heat sinks were choking off the airflow from the fan and preventing air from reaching downstream components. Radian engineers addressed this problem by reducing the number of fins to promote airflow through the heat sink. To determine the optimum heat-sink design, Radian engineers used the Workbench parameter manager to run a parametric study of the



▲ Airflow for original (left) and redesigned card (right)

number of fins, fin thickness, fin height, heat-sink material and different base materials. The results showed, as expected, that the optimal design had fewer and thinner fins. The simulation results also showed that coplanarity issues resulted in poor heat transfer where heat sinks were connected to multiple components of varying heights. Radian engineers corrected this issue by designing separate heat sinks for each of the major heat dissipaters. The new design improved the bond between components and heat sinks, thereby promoting optimal heat transfer.

Radian engineers considered adding a heat pipe, which efficiently transfers heat from a hot spot to a heat sink located some distance away. A heat pipe contains a liquid that turns into vapor when it absorbs heat from a thermally conductive surface attached to a hot component. The vapor travels to the other end of the heat pipe, which is much cooler, and condenses back into a liquid. The liquid then returns to the hot interface through capillary action, and the cycle repeats. Simulation showed that the heat pipe provided further performance improvements. The Icepak model was updated, and

OPTIMIZATION IN ELECTRONICS THERMAL MANAGEMENT USING ANSYS ICEPAK AND ANSYS DESIGNXPLORER
ansys.com/92heat2

a new simulation showed that the final design achieved a 65 C reduction in temperature in the hottest area on the board.

Radian's U.S.-based manufacturing facility provided initial prototype heat sinks within three days. The manufacturer performed final physical tests that matched the simulation results and accepted the Radian design. The product is now on the market with a maximum junction temperature of 85 C at 55 C ambient. Radian is manufacturing the production heat sinks in its Asia facilities. The thermal performance of the product has been verified in the field.

ANSYS Icepak helps Radian to quickly and efficiently diagnose thermal management issues and generate optimal designs. The manufacturer's proof-of-design tests have confirmed the accuracy of Radian's Icepak simulations within a 7 percent margin. These results would not have been possible without simulation.

ELECTRIC POWER THROUGH THE AIR

Murata Manufacturing developed a more-efficient method for wireless power transfer using simulation.

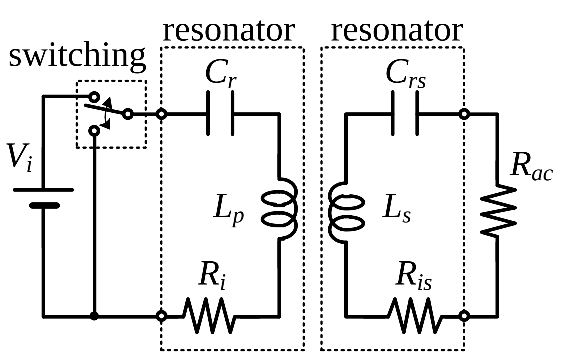
By Tatsuya Hosotani, Visiting Professor, Doshisha University, and Associate Chief Researcher, Murata Manufacturing Co., Ltd., Nagaokakyo, Japan

The dramatic increase in the number of devices in today's intelligent networks and the influence of the Internet of Things raise the question of how all of the devices will be powered. In many applications, the number and disperse locations of these devices rule out wiring all of them to a power supply. Using batteries for each device may create difficult maintenance challenges while also raising environmental issues regarding battery disposal. The ideal solution involves a wireless transfer system to power to these devices, but existing systems are not always up to this challenge. Murata Manufacturing used ANSYS simulation to test a wireless power transfer method called a direct-current-resonance power transfer system so that it can be commercialized.

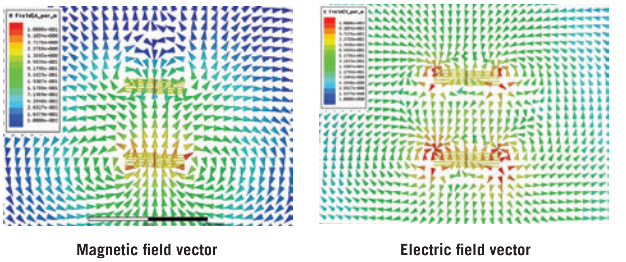
WIRELESS POWER TRANSFER ALTERNATIVES

The best known wireless power transfer method in current use is the Qi standard, which relies upon electromagnetic induction between planar coils. A base station connected to a power source incorporates a transmitting coil that generates an oscillating magnetic field. The magnetic field induces an alternating current in a receiving coil by Faraday's law of induction. However, because electromagnetic induction requires that the transmitting and receiving coils must be very close to each other, this approach does not provide a practical solution to the device proliferation problem. In the case of the Qi standard, the charger is normally physically connected to the device it is charging.

Another option, radio-frequency wireless power transfer, operates at much longer distances than electromagnetic induction but is not very efficient. One reason for the lack of efficiency is the many energy conversion steps that are required in these devices: from alternating current line power to insulated direct current power; to an intermediate radio-frequency power to operate the power amplifier; and then to radio-frequency power to drive the transmitting coil. Next, the radio frequency power is transferred over the air to the receiving device; this power is finally converted to direct current to operate the device on the receiving end. Each of these energy conversions consumes substantial amounts of power, resulting in very low efficiency for the overall system. The result is that radio-frequency power transfer systems generally are large and expensive, and consume relatively large amounts of power.



▲ Direct-current-resonance power transfer circuit



Magnetic field vector | Electric field vector

▲ Magnetic and electric field vectors generated by transmitter coil

NEW DIRECT-CURRENT-RESONANCE METHOD

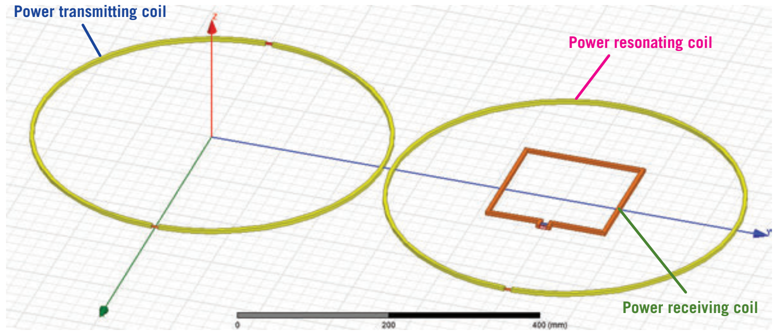
Murata Manufacturing creates innovative products and solutions to benefit society and the electronics industry. The company has developed and is in the process of commercializing a new technique for wireless power transfer called a direct-current-

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

resonance power transfer system. This method involves rapidly switching direct current on and off to drive a transmitting coil that resides on a circuit turned to the frequency at which the current is switched. The transmitting coil produces a resonance field that efficiently transmits direct current through a receiving coil to a circuit tuned to the same frequency.

This method is very efficient because the transmitting coil operates on direct current, the form used by nearly all electronic devices, so no additional energy conversion step is required. And, because direct current is generated on the receiving circuit, no energy conversion step is required. The transfer system distance provided by the direct-current-resonance method is much higher than electromagnetic inductance, and it can be further extended through the use of unpowered resonator coils that relay power to distant receivers. The direct-current-resonance method can use a single power transmitter to drive multiple power receivers as well as resonator coils. The result is an excellent mix of distance and efficiency. This combination makes the direct power resonance method a good fit for powering a wide range of devices that cannot be wired together and for which batteries do not provide a good solution because the device is difficult to reach. Some examples include monitors, radio-controlled equipment, computers and electronics, lighting, and robots.

WIRELESS POWER TRANSFER
ansys.com/92wireless



▲ ANSYS HFSS model with one transmitter, one receiver, one resonator

[redacted]

COMMERCIALIZATION CHALLENGE

In developing its direct-current-resonance power transfer method, Murata faces the challenge of optimizing the power efficiency, size, cost and other features of its devices to produce a competitive commercial product. Optimizing the design of any wireless product requires evaluating many different design iterations. The number of iterations that need to be considered is larger than normal with direct-current power transfer systems because the concept is new, so there is very little experience to draw upon.

It would be very expensive and time-consuming to build and test a prototype for each of these iterations. The build-and-test method would progress relatively slowly toward an optimized design because the amount of diagnostic information it produces is limited. For example, while the amount of power transmitted to the receiving coil is easy to measure, it is usually possible to measure electric and magnetic field parameters only in a few locations.

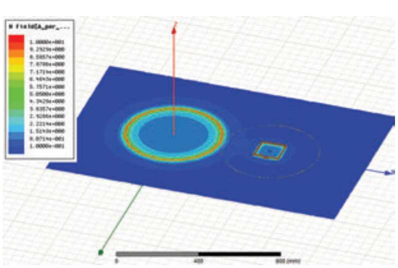
SIMULATION SPEEDS DESIGN OPTIMIZATION

Murata electronic engineers address these design challenges by using ANSYS HFSS to simulate a wide range of design alternatives that take into account the actual geometry and location of the power transmitter, receiver and resonator coils, as well as the effects of other components in the area. Engineers can generate a variety of concept designs to achieve the required levels of power transfer to the target device while addressing other objectives such as minimizing device weight, size and cost.

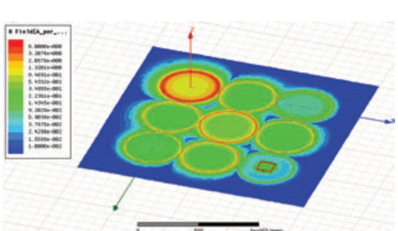
The simulation sequence normally begins with modeling the transmitting coil alone as a subsystem and viewing the resulting electromagnetic field. Next, a single receiving coil is added to the simulation to evaluate power transfer performance. Most real-world applications involve many receiving coils and resonator coils, so these elements are incrementally added to the simulation. Finally, other electronic and structural components are added to evaluate their impact on power transfer performance. Simulation lets engineers explore the power transfer device's sensitivity to parameters such as coil topology, coil diameter, number of windings, resonant frequency, number of receiving coils, number of resonator coils, etc. Engineers can easily modify these and many other design parameters to understand their impact on power transfer performance.

The growing proliferation of connected devices makes it essential to improve on today's wireless power transfer technology. Simulation makes it possible to consider the impact of various design alternatives and deployment strategies in the early stages of the design process. Simulation saves months of testing time and tens of thousands of dollars in resources for each design project by enabling engineers to refine options through virtual prototypes rather than physical prototypes. Murata will continue its research and development with the goal of contributing to the development of wireless power transfer system science and technology.

[redacted]



▲ Magnetic field analysis of model with one transmitter, one receiver, one resonator



▲ Magnetic field analysis of model with one transmitter, one receiver, eight resonators

FUELING RESEARCH REACTORS

Simulations help to design a new low-enriched nuclear fuel for materials testing, isotope production and neutron radiography for research that reduces proliferation threat.

By Markus Piro, Research Scientist, and **Anthony Williams**, Research Scientist,
Canadian Nuclear Laboratories, Chalk River, Canada

Research reactors are used for a variety of applications, including materials testing, neutron radiography and the production of radioisotopes for medicinal and industrial purposes. Canadian Nuclear Laboratories (CNL, formerly Atomic Energy of Canada Ltd.) — Canada's premier nuclear science and technology laboratory — is developing a new low-enriched uranium (LEU) fuel designed for use in research reactors around the world. LEU fuel is favored for research reactors because it reduces proliferation risks in comparison to highly enriched uranium (HEU) fuels.

Photos courtesy Canadian Nuclear Laboratories.

DEVELOPING A NEW REACTOR FUEL

Nuclear research reactors generate a neutron source for a wide variety of research and industrial purposes. HEU fuel is used in many research reactors because it allows a compact core and reasonably long times between refueling. As security concerns with the use of HEU fuel have grown within the international community, many research reactors have converted to LEU fuel.

CNL is designing and qualifying a new LEU fuel that can be used in research reactors to replace HEU and currently used LEU uranium-silicide fuels. Uranium–molybdenum (U-Mo) dispersion fuel is an attractive option for this next generation of fuels because of the ease of spent-fuel reprocessing and the potential for substantial density improvements over uranium silicide–based fuels. With any fuel design, the modeling of thermal–mechanical properties is key to concept evaluation. ANSYS software provides an effective toolset to perform these evaluations and guide more-detailed fuel design. Furthermore, numerical simulations help to guide the safety analyses of experiments by making it possible to study the thermal and fluid behaviors of candidate fuel designs before they are placed in the reactor for complete testing.

Microstructural modeling with ANSYS Mechanical finite element analysis (FEA) software helped CNL scientists to estimate

ANSYS Mechanical helped CNL scientists to estimate the thermal conductivity of potential LEU fuels.

ANSYS Fluent simulated all three modes of heat transfer: convection, conduction and radiation.

the thermal conductivity of candidate LEU fuels, which is the most important material property in predicting fuel behavior. ANSYS Fluent computational fluid dynamics (CFD) software was used to perform solid–fluid conjugate heat transfer simulations of two candidate LEU fuels to establish how several design variables affected their operating temperature, pressure loss and power output. The simulation results helped to optimize the design of fuel elements that will be used for in-reactor testing scheduled for the near future.

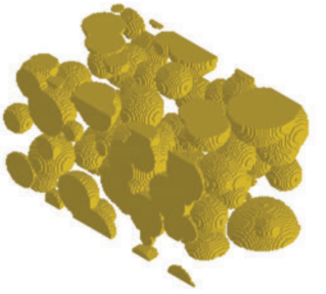
FEA PREDICTS THERMAL CONDUCTIVITY

Determining thermal conductivity of fuel during irradiation is a critical first step in simulation, since both fuel performance and safety depend on its thermal behavior. Measuring the thermal conductivity of irradiated fuel is both difficult and expensive. Of particular importance in U-Mo fuels dispersed in an aluminum or magnesium matrix is capturing the impact of the low thermal conductivity

NUCLEAR ENERGY
ansys.com/92nuclear

interaction layer formed by a chemical reaction between U-Mo fuel particles and the matrix. CNL researchers used ANSYS Mechanical to model a representative region of the fuel rod as a prism-shaped unit cell made of brick elements. The elements within the unit cell were assigned the material properties of either the fuel or matrix, depending on their position, to represent randomly distributed fuel particles with a size distribution similar to the manufactured fuel.

The team simulated the presence of a fuel–matrix interaction layer by adding a third set of material properties assigned to the finite elements that surround each fuel particle. Researchers estimated the thermal conductivity of the interaction layer by choosing a value that best matched the observed degradation in thermal conductivity of the fuel sample as a function of the volume fraction of the interaction layer. Applying an appropriate heat flux across the unit cell made it possible to determine the effective thermal conductivity of the unit cell as a function of the volume fraction of the fuel particles. The effective thermal conductivity of the material was determined as a function of the volume fraction of the interaction layer.

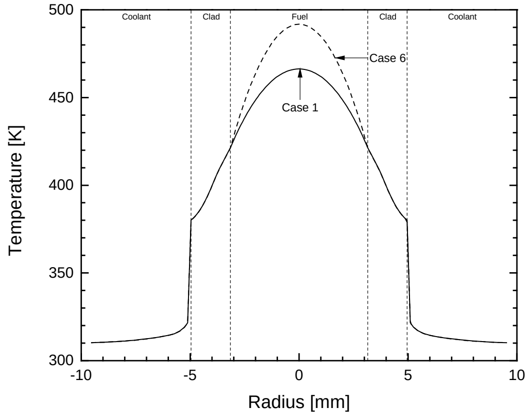


▲ Typical particle distribution in ANSYS Mechanical model

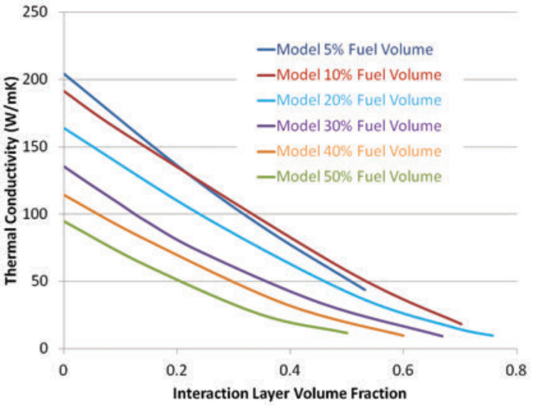
CFD PREDICTS OVERALL FUEL PERFORMANCE

CNL researchers then used the simulated thermal conductivity values as input to CFD simulations of the proposed U-Mo dispersion fuels to optimize fuel design and ensure its safety in preparation for physical testing. Fluid flow and heat transfer of a fuel assembly were simulated in a 3-D geometry. Heat is generated by nuclear fission within the fuel, which is cladded entirely in aluminum. Together, the fuel and cladding are referred to as a fuel element. Eight concentric fins are attached to the cladding of each fuel element to enhance heat removal from the fuel to the surrounding coolant. The CFD software simulated all three modes of heat transfer: convection, conduction and radiation.

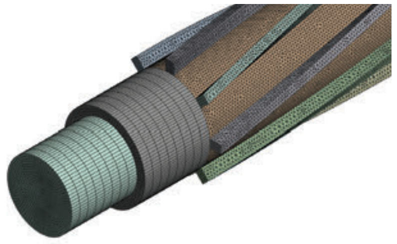
Simulations were performed in which inlet fluid velocity, fuel type and linear power were varied to calculate the relative sensitivity of the fuel, cladding and coolant temperatures to each design variable. In addition, CNL considered an alternate fuel geometry that replaced the straight



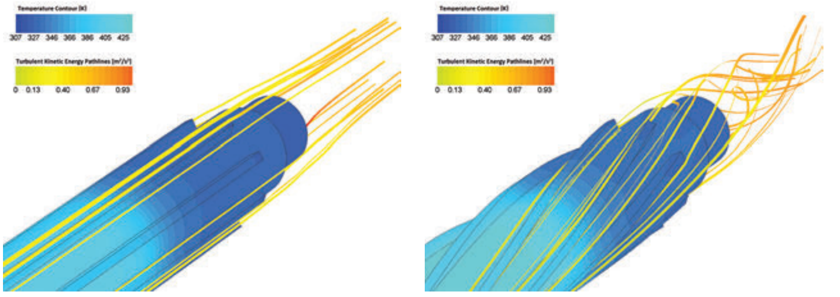
▲ Predicted temperatures are compared for U-Mo/Al (case 1) and U-Mo/Mg (case 6) fuels.



▲ Simulation shows that thermal conductivity falls as the interaction layer volume fraction grows.



▲ Meshed mini-element with straight fins. The mesh representing the fluid region is not shown to allow the solid structure to be visible.



▲ Fluid-flow pathlines for clad surface with straight fins (left) and helical fins (right)

fins in the original design with helical fins — which does not greatly complicate the manufacturing process — to investigate the effect on heat transfer. Replacing straight fins with helical fins reduced the predicted cladding and fuel temperatures. Heat transfer between the fluid and solid regions was improved by the increased level of turbulence generated by the nonlinear geometry, which promotes mixing of coolant near the hot cladding surface with the relatively cooler bulk fluid. Thus, the enhanced heat transfer with this design suggests that higher operating powers may be permitted.

The numerical simulation predictions support performance and safety analyses of candidate fuel designs and guide physical experiments of U-Mo dispersion fuels. The trends seen in varying power and inlet flow conditions will be useful in optimizing performance and safety of the physical experiments. The temperature predictions provided by the model will help to determine the linear power for possible future use in a reactor. The benefits of the helical-finned cladding provided justification for manufacturing this unique design. In the end, this new fuel design may enable efficient and safe use in research and test reactors while reducing proliferation risks. ■

Reference

Canadian Nuclear Laboratories
cnl.ca

ACCELERATING MECHANICAL SOLUTIONS USING THE LATEST INTEL TECHNOLOGIES

Advances in ANSYS 16.0 and Xeon technology address the high-performance computing needs of Windows users.

By Wim Slagter, Lead Product Manager, and **Jeff Beisheim**, Lead Software Developer, ANSYS

Organizations using ANSYS structural mechanics simulation software expect the accuracy, efficiency and throughput required to generate reliable designs as quickly as possible. ANSYS has worked with Intel to make sure that these companies can leverage the latest Intel® Xeon® E5 v3 processors and Xeon Phi™ coprocessors for their simulation workloads.

Structural mechanics simulations often require a large amount of computing resources including memory, disk space and I/O. This impacts time spent computing. Because CPU clock rates are not increasing as quickly as they were a decade ago, faster CPUs cannot be counted on to maintain the computing pace. The new performance paradigm is parallel computing that leverages the swelling number of CPU cores that continues to grow every couple of years to deliver increased computations at each clock cycle. This has resulted in significant performance gains for structural simulation software. But engineers are always trying to minimize simulation times so that they can increase the complexity of their models (for example, by increased mesh density or nonlinear behavior) or simply run more simulations in a given period of time.

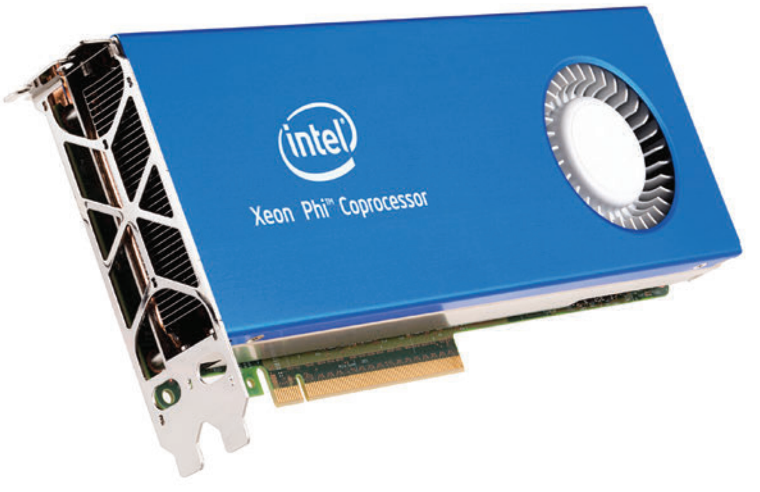
One way to speed up structural mechanics simulations is to make full use of the latest available hardware. The computer industry has delivered enormous increases in computing performance with continued platform advancements, including more compute cores per CPU, integrated I/O processor (yielding higher memory bandwidth), additional and faster memory (channels), larger L3 cache size, faster disk storage (like solid-state drives for ANSYS Mechanical), faster interconnects, and Intel Advanced Vector Extensions 2 (AVX2) support. Intel and ANSYS continue to work together so that ANSYS solutions can take advantage of these hardware advances.

Intel Processor Family	Iterative Solver Benchmarks	Direct Solver Benchmarks
E5 (Sandy Bridge)	1,408 sec	1,067 sec
E5 v3 (Haswell)	1,117 sec	535 sec

▲ The benchmark suite was run using ANSYS 16.0 on two very similar systems: one containing two Intel Xeon E5-2670 (Sandy Bridge, 2.6 GHz, 16 total cores) processors, and one with two Intel Xeon E5-2697 v3 (Haswell, 2.6 GHz, 28 total cores) processors. The geometric mean of the total elapsed times for each benchmark run with 1, 2, 4, 8 and 16 cores was used to generate the times shown in the table. The Haswell system is on average 20 percent faster than E5 v2 for the iterative solver benchmarks and 40 percent faster than E5 v2 for the direct solver benchmarks.

Engineers always try to minimize simulation times so that they can increase the complexity of their models or simply run more simulations in a given period of time.

INTEL SOLID-STATE DRIVES INCREASE PRODUCTIVITY OF PRODUCT DESIGN AND SIMULATION
ansys.com/92coprocessor



LEVERAGING INTEL XEON E5 V3 PROCESSORS

ANSYS structural mechanics products have supported parallel processing over two decades, allowing engineers to effectively use multi-core processors and/or clusters to speed up their simulations. With the launch of release 16.0, ANSYS continues its sustained investment by adding capabilities to exploit the latest Intel processor technologies.

With Intel's latest Xeon E5 v3 processors, ANSYS users will see significant reduction in simulation runtimes, mainly due to the additional cores (up to 18), Intel AVX2 support, larger L3 cache (up to 35 MB), and higher memory speed (up to 2,133 MHz). ANSYS Mechanical 16.0 shows improved performance for the E5 v3 generation of processors from Intel, code-named Haswell. The E5 v3 system is on average 20 percent faster than E5 v2 for iterative solver benchmarks (usually good measures of memory bandwidth speed) and on average 40 percent faster than E5 v2 for direct solver benchmarks (usually good measures of raw compute speed).

LEVERAGING INTEL XEON PHI COPROCESSORS

To leverage cutting-edge hardware advancements to deliver faster engineering simulation technology, ANSYS has worked with NVIDIA since the release of ANSYS 13.0 to develop and release parallel solver execution on general-purpose graphics processing units (GPUs). GPUs can now speed up fluids, structural and electromagnetic simulations to increase the value of ANSYS high-performance computing (HPC) capabilities.

Recently, Intel released the Xeon Phi series of coprocessors that are similar in design to high-end GPUs. They are full-height cards that plug into a PCI Express slot and require at least 200 watts of additional power. However, the coprocessors are not meant for graphics and have no connections for graphical display output (for example, HDMI or a display port). Each Xeon Phi coprocessor contains roughly 60 cores that can perform computations at just over 1 teraflop and has 8 GB to 16 GB of GDDR5 memory to provide significant amounts of memory bandwidth. This new hardware accelerator can potentially speed up structural mechanics simulations.

IMPLEMENTATION

Before starting the implementation to support Xeon Phi coprocessors in structural mechanics products, ANSYS required that:

- The user experience would be straightforward and simple.
- Xeon Phi hardware must never slow down the simulation and, when applicable, should accelerate it.
- Xeon Phi would not compromise the accuracy of the solution.

ANSYS 16.0 — Efficiency and Robustness

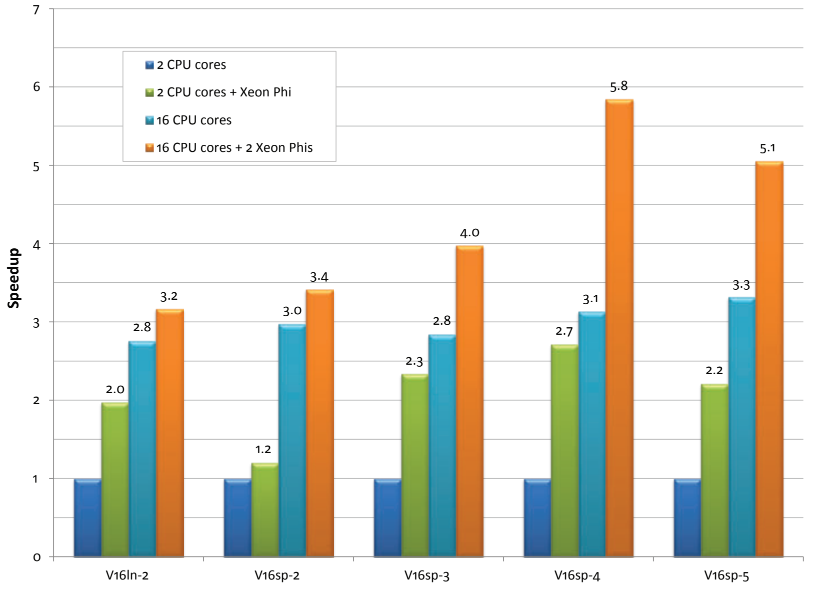
With the launch of release 16.0, ANSYS continues its sustained investment to improve efficiency and robustness for structural mechanics simulations.

Key improvements in solver numerics allow faster and more robust simulations.

- Numerous enhancements improve the convergence of nonlinear analyses.
- Sparse solver improvements allow more jobs to run in-core, leading to better solver performance.

Numerous improvements were made in the area of distributed memory parallel computing.

- Domain decomposition has been further improved, leading to faster performance and better scaling, particularly at higher core counts.
- Newly added capabilities include support for inertia relief, QRDAMP eigenvalue extraction method (in modal analysis) and mode-superposition method (in harmonic and transient analysis).



▲ Overall simulation speedup factors using Intel Xeon Phi coprocessors with ANSYS Mechanical 16.0

Benchmarks	V16in-2	V16sp-2	V16sp-3	V16sp-4	V16sp-5
Simulation Type	modal analysis for the 50 lowest frequencies	nonlinear, transient, structural analysis	harmonic structural analysis	nonlinear, static, structural analysis	nonlinear, transient, structural analysis
Number of Equations	2 million	4.7 million	1.7 million	3.2 million	6 million

With Intel's latest Xeon E5 v3 processors, ANSYS users will see significant reduction in simulation runtimes.

DEBUNKING SIX MYTHS OF HIGH-PERFORMANCE COMPUTING
ansys.com/92myth

To utilize the Xeon Phi coprocessor to speed up ANSYS structural mechanics simulations, the software uses the GPU accelerator capability. Although Xeon Phi allows for other execution models, the GPU accelerator was a natural fit to introduce this coprocessor. Because the sparse direct solver is the default solver and is commonly used for all types of analyses, this linear equation solver was the best place to start.

ANSYS Mechanical 15.0 supported Xeon Phi coprocessors with shared-memory parallelism on Linux® platforms only. However, distributed memory parallelism typically provides more significant speedup than shared memory parallelism, and ANSYS structural mechanics software is often run on the Windows® platform. ANSYS Mechanical 16.0 supports shared memory and distributed memory parallelism for both the Linux and Windows platforms. Virtually all ANSYS users — including those who have access to clusters in which each compute node contains one or more coprocessors — can accelerate structural mechanics simulations using Xeon Phi coprocessors.

THE VALUE OF HIGH-PERFORMANCE COMPUTING FOR SIMULATION
ansys.com/92value

USING XEON PHI ACCELERATION

To enable the use of Xeon Phi hardware within ANSYS Mechanical, activate the GPU accelerator capability upon launching the software by adding the `– acc intel` option to the list of command line arguments. You can also select how many Xeon Phi coprocessors to use with `–na N`, where N is an integer number greater than 0. (The software defaults to 1 for a single coprocessor.)

ANSYS Workbench users can easily enable this feature during solution by modifying the GPU acceleration option on the Advanced Properties page of the Solve Process Settings. Select INTEL in the associated drop-down box and then choose the number of Xeon Phi coprocessors to enable during the simulation. Activating this capability requires one additional HPC license for each coprocessor.

Once activated, this capability will accelerate the solution, when possible, by automatically using the Xeon Phi hardware. No user input is required. In cases in which acceleration is not possible, the CPU core(s) will continue to be used, and the Xeon Phi feature will have no effect on the progress of the solution.

PERFORMANCE

ANSYS conducted a series of standard benchmarks for ANSYS Mechanical to obtain performance data. The benchmarking used a workstation running Windows 7 x 64 SP1 with 128 GB of RAM and two Intel E5-2670 (2.6 GHz) processors with a total of 16 CPU cores. Two Xeon Phi 7120A coprocessors were utilized in the workstation.

The results showed that using a Xeon Phi always provides some level of acceleration. However, the amount of acceleration achieved varies greatly from benchmark to benchmark, and it also depends on the number of CPU cores involved. With two CPU cores and a single Xeon Phi coprocessor, an average speedup of 2.1 times is achieved for the entire simulation, compared to using only two CPU cores. With 16 CPU cores, the addition of the two Xeon Phis provides on average 1.4 times speedup for the overall simulation. Because the performance varies for each benchmark, some guidelines are required to understand which structural mechanics models are expected to achieve the most acceleration when using a Xeon Phi coprocessor.

USAGE GUIDELINES

The amount of acceleration gained from using the Xeon Phi coprocessor varied greatly with the hardware used and the model simulated. These guidelines can help to determine whether the coprocessor will provide a performance boost.

Using newer, faster CPU hardware typically decreased the amount of speedup achieved when using a Xeon Phi card. Using more CPU cores per Xeon Phi coprocessor will also decrease the amount of speedup achieved. If one or more coprocessors is requested, all available coprocessors are used. However, for performance reasons, the number of processes per Xeon Phi coprocessor is limited to a maximum of eight.

Certain classes of simulation are expected to achieve more acceleration when using a Xeon Phi. For ANSYS Mechanical simulations, more acceleration is achieved when:

- The sparse solver is running in the in-core memory mode.
- The assembled matrix size is greater than 2 million equations.
- Models are three-dimensional, have bulkier or thicker geometry, contain higher-order element types or include certain types of boundary conditions (for example, constraint equations).

INCREASING VALUE THROUGH ONGOING COLLABORATION

As the computing power provided by hardware vendors increases, ANSYS will continue to harness the full potential of this new technology. As vendors provide more parallel hardware, ANSYS developers continue to parallelize more algorithms in the software. For structural mechanics simulations, these efforts are critical to ensure that companies can meet competitive demands to deliver innovative and robust products to market by performing increasingly complex simulations in a short time.

Intel and ANSYS will continue to work together to deliver optimized and tested solutions that deliver value. For these new types of hardware accelerators, like Xeon Phi coprocessors, the main limitation is the amount of computations that can be offloaded onto the accelerator device. Future Xeon Phi products aim to offer the ability to accelerate more computations as well as to remove the limitation of transferring data to the device (via the PCI Express channel).

ANSYS MECHANICAL 16.0 ON INTEL PLATFORMS
ansys.com/92accelerate

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

Send address corrections to
AdvantageAddressChange@ansys.com