ANSYS

Excellence in Engineering Simulation

ADVANTAGE

SPECIAL ISSUE 2016

Best of
Automotive

Getting Around in Style

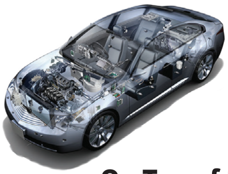
On the Fast Track

Safe Automobile Controls

Table of Contents

Best of Automotive

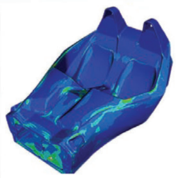
2 Driving Automotive Innovation



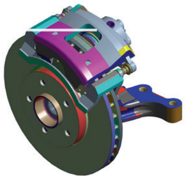
3 On Top of the World

6 Leveraging Upfront Simulation in a Global Enterprise

BODY AND CHASSIS

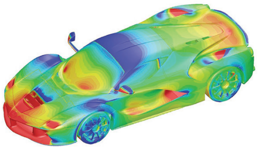


8 Getting Around in Style



12 Breakthrough for Brake Design

15 It's a Snap



19 On the Fast Track

TRADITIONAL POWERTRAIN

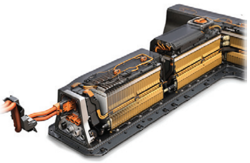
23 Cleaning Up

26 Cleaner, Greener Engine Design

29 Playing It Cool

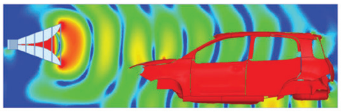
AUTONOMOUS AND ELECTRONIC SYSTEMS

33 Safe Automobile Controls



38 Automating Battery Pack Design

42 In the Loop



45 Test Drive for EMI

50 Hot Flash

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

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

Join the simulation conversation
ansys.com/Social@ANSYS

Contact Us
Web: www.ansys.com
Phone: 844.Go.ANSYS (844.462.6797)
Email: info@ansys.com

DRIVING AUTOMOTIVE INNOVATION

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

Innovation is not just a buzzword in the automotive industry — it is a critical competency needed to transform vehicles into smart machines that incorporate electronics for infotainment (phone, multimedia), guidance (GPS) and control of a variety of systems, such as adaptive cruise control and automatic parallel parking. Innovation is also indispensable to meet new government standards that regulate fuel efficiency/emissions and drive the need for hybrid/electric vehicles. While accelerating these advancements, OEMs and suppliers must also control increasing product complexities and multiplying failure modes to keep vehicles robust, reliable and safe.

Auto companies address fuel economy and emissions by reducing aerodynamic drag, vehicle weight and rolling resistance. Hybrid/electric vehicle (HEV) designers address these same issues through innovations in batteries, traction motors, power electronics and fuel cells. R&D efforts in electronics and embedded software focus on antenna design, EMI–EMC, electronics reliability and ISO 26262-qualified code generation. Engineers are also making revolutionary advances in autonomous vehicles and advanced driver-assistance systems (ADAS), developing radical new sensors, machine perception algorithms and control techniques.

This ultra-competitive industry turns to engineering simulation to cut through complexity, virtually testing thousands of operating scenarios and uncovering hard-to-find, potentially disastrous problems early on. Tier 1-supplier DENSO, for example, embeds CAE into all phases of its product development process, improving quality and reducing time to market along the way. Advanced virtual analysis enables such pacesetters to create category-changing innovation.

Body and Chassis

Simulation is key to solving issues upfront in the design phase. Companies that fine-tune auto body and chassis can reduce fuel consumption and build in reliability upfront in the design process with simulation. KTM Technologies incorporated radical composites into a sports car, which called for new design, analysis and optimization technologies. Created using simulation, the product struck a fine balance between requirements, performance and costs while exceeding customers' requirements.

Friction-induced brake squeal grows important as other vehicle noise sources are mitigated. ZF-TRW engineers accurately simulated squeal and automated the simulation process while reducing time and money spent on validation testing. Performing simulation early in the design process helps to avoid costs associated with multiple prototypes, rework and tooling changes. Valeo used nonlinear best practices to simulate thermoplastic snap-fits, leveraging HPC that shrunk simulation time by 50 percent.

Traditional Powertrain

An early adopter of simulation technology, the auto industry regularly applies simulation to complex real-world physics interactions and makes value-added design trade-offs. To reduce engine emissions and improve fuel efficiency, Magneti Marelli models the complete ICE cycle virtually, reducing the time required to develop innovative components. Cummins applies simulation to workhorse diesel engines for trucks, reducing weight, improving fuel economy and reducing emission. Toyota's simulation approach enables it to evaluate more design alternatives for transmission cooling performance in the early stages of the product development process.

Electric Powertrain

Companies that bring more fuel efficient, less expensive HEVs to market sooner will dominate future automotive business. Engineering simulation drives time and costs out of the development cycle.

In today's connected cars, electronic control units (ECUs) that manage various systems are governed by complex software, which is susceptible to glitches. Subaru uses virtual modeling to develop safe, reliable, electronically controlled systems for HEVs. The process also reduces software development and testing time.

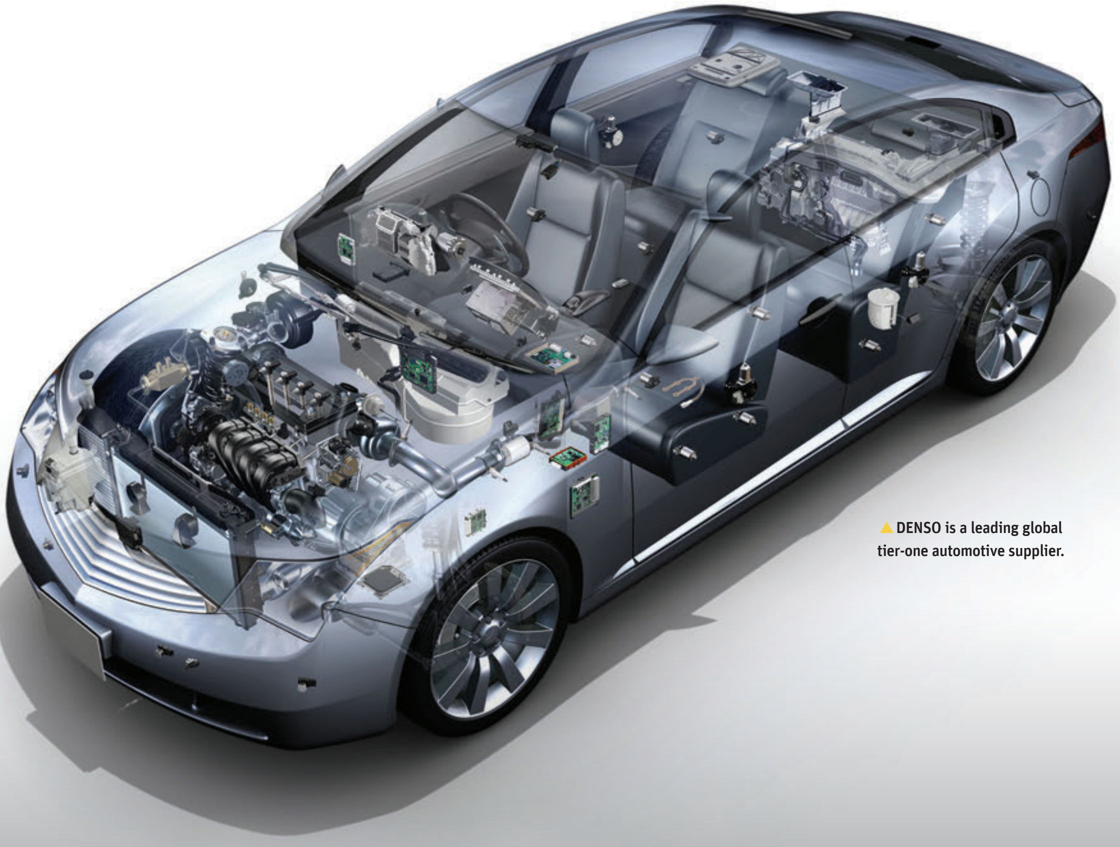
General Motors is leading a cross-industry team in developing an efficient cooling system for HEV battery packs. Using systems-level simulation tools to design lithium-ion systems and accurately predict their performance is a vital component of the R&D strategy.

Autonomous and electronic systems advanced driver assistance systems (ADAS) involve a complicated control-loop that must function flawlessly over the millions of scenarios that autonomous vehicles encounter. Simulating ADAS involves drive-scenario modeling, sensor physics modeling, sensor data fusion, human–machine interfaces and more. The German Aerospace Center is applying virtual modeling to correct defects and gain insight early in the design process, substantially reducing the time required to produce vehicle automation systems.

Simulation allows for accurate what-if analysis to determine potential EMI issues caused by electronic communications devices. Simulation also leads to better understanding of transient noise issues caused by the myriad motors included in every vehicle. Many standards, directives and regulations are designed with vehicle safety in mind.

In the future, the automotive industry will see many radical changes. As autonomous cars become more prevalent over the next 25 years, the market will transform from B-to-C to B-to-B. "Carline" and "robo-taxi" fleets will appear; individuals will no longer own vehicles. But one thing that will not change is the need for engineering simulation to design the vehicles of the future: To physically test an autonomous vehicle to current standards would require more than 1 billion road-testing miles over 100 years. Consumers and the auto industry cannot wait that long!

ON TOP OF THE WORLD



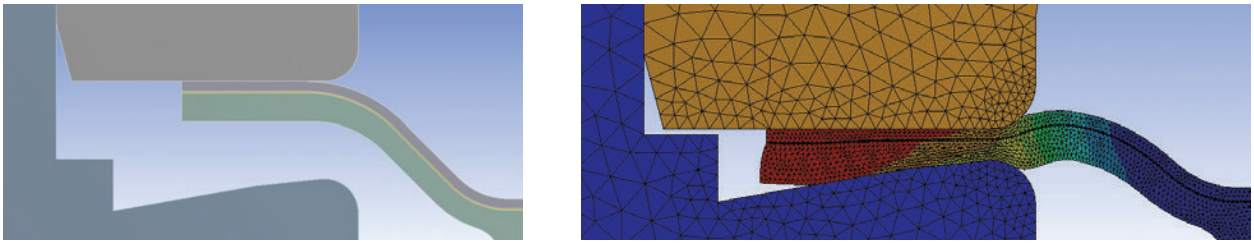
DENSO is a leading global tier-one automotive supplier.

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

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

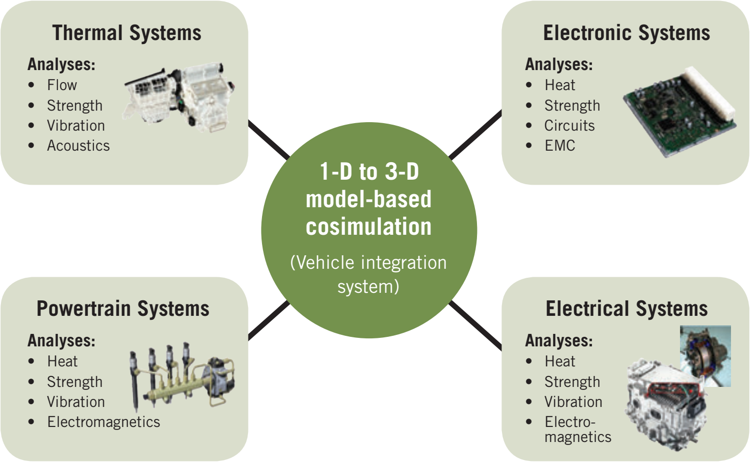
Competition is intense in the automotive systems and components business. Best-in-class simulation capabilities are necessary to thrive amid global competition. DENSO Corporation — a leading supplier of advanced automotive technology, systems and components for all the world's major automakers — faced 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.

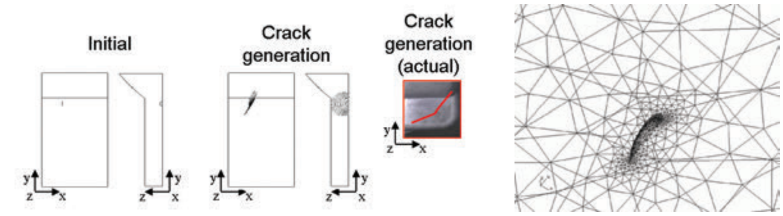


Using large deformation analysis allows the DENSO team to predict strength and the shape profile. With some compressions at more than 50 percent of the original thickness, structural simulation helps the company to ensure product reliability.

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



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.

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

DENSO BACKGROUND

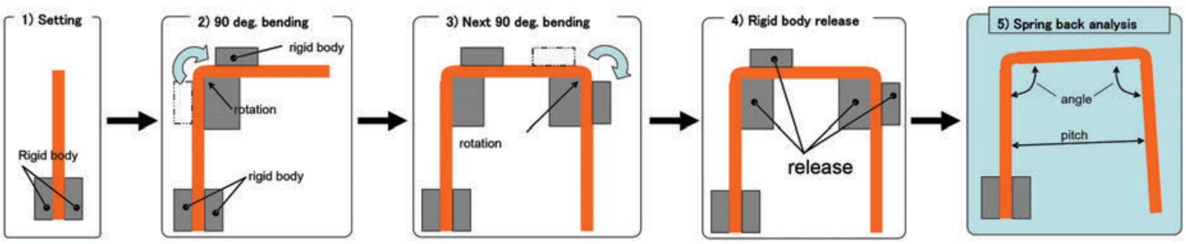
With sales of $39.8 billion U.S. in the latest fiscal year, DENSO is organized into business groups focusing on powertrain controls, electronic systems, thermal systems, and information and safety systems. A small group of DENSO analysts began using computer-aided engineering (CAE) about 30 years ago 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 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.

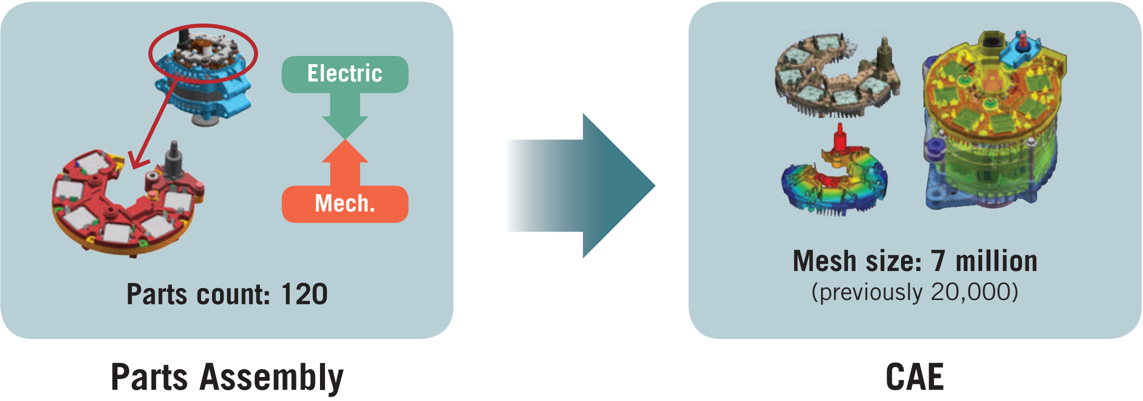
BENCHMARK STUDY IDENTIFIES ANSYS AS TECHNOLOGY LEADER

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.

Leveraging Upfront Simulation in a Global Enterprise

Corporate initiative at Delphi focuses on the benefits of simulation as an integral part of early product design at sites around the world.

By Fereydoon Dadkhah, Senior Engineer, Mechanical Analysis and Simulation,
Delphi Electronics & Safety Systems, Indiana, U.S.A.

Fereydoon Dadkhah, Delphi Electronics & Safety Systems

Most high-technology companies now realize the potential benefits of simulating the performance of their products using tools such as finite element analysis (FEA). They also clearly know that performing analysis early in the design cycle has the potential to identify and solve design problems much more efficiently and cost effectively compared to handling them later. One of the leading companies in employing upfront analysis throughout the product engineering organization is Delphi Electronics & Safety Systems — a major division of Delphi Corporation specializing in mobile electronics and transportation systems for the automotive and consumer product industries.

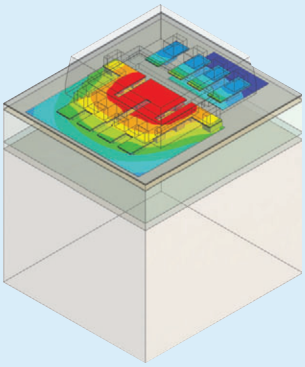
Beginning in the late 1990s, Delphi Electronics & Safety embarked on a program to take full advantage of FEA in the product development process. Along with other companies, Delphi Electronics & Safety had been using FEA in a more limited way as a troubleshooting tool often later in development. The new initiative intended to employ finite element analysis as an integral part of the product development process — especially focusing on the use of simulation up front in the design cycle.

To achieve this goal, Delphi put into place a comprehensive program to train design engineers in the use of FEA in the early stages of the design process. This program began by classifying engineers according to their skill levels in use of FEA and interpretation of analysis results. Gradually,

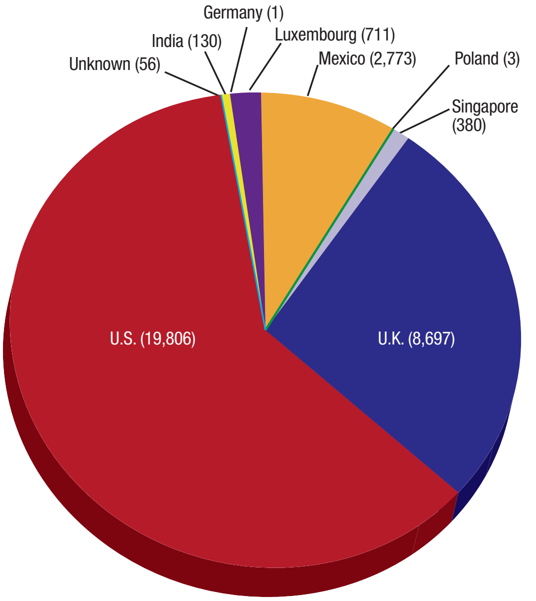
Steady-State Thermal Analysis

The most common use of the ANSYS Workbench tool at Delphi is by design engineers engaged in product development. Analysis types include steady-state thermal, free vibration and linear static stress analysis. More advanced types of analysis, including those involving material or geometric nonlinearity, transient loading, fluid flow and multiphysics, are performed by full-time analysts using products from ANSYS or other commercially available analysis tools.

Delphi produces a number of products including those for use in the automotive and consumer product sectors that must meet stringent thermal requirements. A steady-state thermal analysis is, in many cases, the first step in ensuring that the final product will meet the thermal requirements of the customer. Based on usage data collected annually, the ANSYS DesignSpace tool is widely used to perform this type of analysis. Shown in the accompanying figure is an example of the results of a steady-state thermal analysis of an integrated circuit package used in a transmission controller unit. Once the steady-state performance is established, transient and system-level analyses are performed to completely characterize the system.



Steady-state thermal analysis of an IC package used in a transmission controller unit



Number of hours of ANSYS DesignSpace usage at various Delphi sites internationally

the program incorporated use of FEA into the Delphi Electronics & Safety product development plan. Safeguards such as peer reviews, engineering fundamentals training and mentoring were implemented to ensure proper use of FEA. Furthermore, Delphi Electronics & Safety has restricted use of this technology to engineers and scientists with a minimum of a bachelor's degree. Training in the use of the structural mechanics simulation software — in this case, ANSYS DesignSpace that uses the ANSYS Workbench platform — is a prerequisite at Delphi Electronics & Safety. Occasional users such as product engineers utilize the software to perform linear and static analyses. More advanced analyses involving nonlinearity or transient loading are referred to full-time analysts.

Today, the company has incorporated the use of structural mechanics simulation into the Delphi Product Development Process (PDP) as a requirement. The PDP begins with the concept stage and proceeds to the validation stage when prototypes are built and tested, and finally the program is handed off to manufacturing. This has led to developing much more robust and reliable products as well as greatly reducing or eliminating validation failures. This process is enforced by a Design Failure Modes and Effects Analysis (DFMEA) plan represented by a spreadsheet of possible failure modes for a product and the required analyses to show that the product is immune to the specific failures.

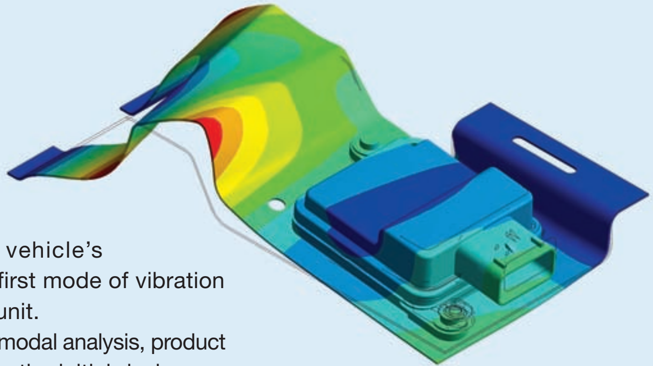
A large number of engineers around the world in the overall Delphi organization use these tools, including the full suite of software from ANSYS within the ANSYS Workbench interface, to perform thermal, stress, vibration and other general analysis in the course of product development. In 2007, the number of ANSYS DesignSpace users exceeded 200, and approximately 30 percent were Delphi Electronics & Safety engineers. Delphi Electronics & Safety users globally logged 11,151 hours of usage on the software, or 34 percent of the total for all sites internationally. The licenses for many CAE tools — including ANSYS products such as ANSYS DesignSpace — are supported from servers in Michigan, in the United States, allowing engineering management to stay up to date on the use of these tools.

By adopting a comprehensive approach for implementing FEA across the worldwide organization, Delphi has effectively incorporated an extremely powerful technology into the product development process. The initiative to focus on upfront analysis in particular has resulted in outstanding business value for Delphi in terms of improved designs developed very efficiently. The use of the ANSYS Workbench platform has certainly facilitated this process by providing the ability to perform a variety of analysis types of different complexities in the same familiar environment. Perhaps the best indicator of the effectiveness of this software in a business context is management support for its widespread use by such large numbers of Delphi engineers around the world. ■

Finding Natural Vibration Modes

ANSYS DesignSpace software is often used for determination of the natural modes of vibration of a system. Many of Delphi's products are used by the automotive industry, and the first step in establishing that a product can be used in a vehicle is to ensure that the first few modes of vibration of the product are beyond the minimum values that can be excited by the vehicle's operation. The accompanying figure shows the first mode of vibration for a bracket used to support an airbag control unit.

Using the ANSYS DesignSpace tool to perform modal analysis, product engineers are able to determine if any changes to the initial design are needed to improve the vibration characteristics of the system. The design then proceeds to the next stage, in which harmonic and power spectral density (PSD) analyses are performed and any required changes are made.



First mode of vibration for bracket that supports an airbag control unit

Getting Around in Style

Engineers quickly and reliably design a composites sports car and an electric bicycle using ANSYS technology.

By Martin Perterer, Head of Research and Simulation, KTM Technologies GmbH, Salzburg, Austria

DESIGN INTEGRATION OF ADVANCED LIGHTWEIGHT COMPOSITE MATERIALS AND THE CHALLENGES OF MULTI-FUNCTIONALITY

ANSYS.COM/82style

> **Industry-leading research-based ANSYS software delivers the high-quality results the company depends upon.**

The use of composites is rapidly growing across many industries, fostering the need for new design, analysis and optimization technologies. Every industry feels increasing pressure to launch breakthrough products that outperform competitors and meet market needs. For many design applications that require strong yet lightweight materials, layered composites are ideal. Even so, faster, more frequent product introductions and new technologies cannot compromise ultimate product quality, reliability and speed to market.

For KTM Technologies, fiber-composites engineering, technology and consulting comprise the core business. Founded in 2008, the Austrian-based company is part of the KTM Group, focused on people-moving applications — automobiles, motorcycles and bicycles — using high-performance composites. KTM Technologies is a leader in selling solutions and supporting customers in economical, composite engineering via a holistic approach. All departments work together — from design through development and simulation to manufacturing — to benefit customers.

Composites design — in particular composites with carbon fibers — evolves continually as new fibers are developed, existing materials are re-purposed into composite layers, and new applications are explored. KTM Technologies needs to find that fine balance between requirements, performance and costs while exceeding customers' requirements. The company uses ANSYS software to accomplish such advanced materials design. KTM also used ANSYS computational fluid dynamics for shape optimization of the KTM X-Bow to ensure passenger comfort by reducing the cockpit's highly turbulent, high-velocity flow.

ANSYS COMPOSITE PREPPOST

In composites design, two or more materials with very different properties provide light weight and high strength along with highly flexible components ideal for manufacturing complex-shaped products. KTM leverages ANSYS Composite PrepPost to run simulations in the product development/concept phase. This allows engineers to make quick, responsive decisions early before committing major time and resources. The user-friendly software interface provides a fast learning curve for new hires, and results can be calculated quickly. Industry-leading research-based ANSYS software delivers the high-quality results that the company depends upon.

▲ KTM X-BOW, from sketch to prototype in just 18 months, thanks to a cross-functional team approach

KTM X-Bow Sports Car

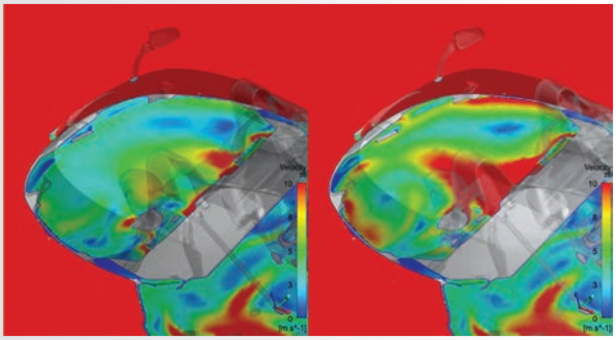
The KTM X-BOW (pronounced “cross-bow”) is a radical, lightweight production sports car that demonstrates what optimizing design and function using composite structures can deliver. Approved for road traffic, this mid-engine sports car builds on race track technology. The body incorporates an innovative monocoque of composite carbon fibers, a pioneering technology previously reserved exclusively for racing vehicles, which provides weight and safety advantages. With a monocoque design, the external skin provides the main structural support — like an egg shell — as opposed to an internal frame. This approach provides the required structural loads using a composite layer design — up to 300 layers in some parts.

Using ANSYS Composite PrepPost during product development allowed the KTM design team to investigate the directional dependencies of the various layers, physical properties and possible layups, fiber orientations, and other variables. All the details could be precisely analyzed and simulated to ensure that design requirements were met consistently. The team fully leveraged the design flexibility of the software. Engineers were able to quickly run three different variations at the concept stage and, within a half day, they could determine which one best fulfilled design requirements.

The team analyzed the failure behavior of the composite design under different load scenarios before committing to the final design. Because all the major components (including front/rear suspension and seats) are interconnected in a monocoque design, many different load cases were run to analyze static and dynamic loads. For example, the rear-engine mounting points on the aluminum rear frame must handle extreme forces, including torsional forces that occur when accelerating around curves. The unique design for the X-BOW used a torque arm directly connected to the carbon monocoque form.

▲ The composite monocoque exterior shell has more than 300 pre-cut layers.

▲ Evaluating failure criteria for all layers in the monocoque saved time and money.



▲ Using ANSYS fluid dynamics software, KTM designed a wind blocker to ensure passenger comfort. Simulation was performed at driving speed of 140 kilometers per hour, with red indicating higher velocity: air velocity with wind blocker (left) and without (right).

The first designs of the KTM X-BOW were engineered without using Composite PrepPost. However, once the team applied the software, they reduced the monocoque’s weight — a very important aspect of sports car design — by 20 percent.

In addition, ANSYS fluid dynamics software was used to help reduce high-velocity flow in the car’s cockpit. By changing the design to add a wind blocker, engineers enhanced passenger comfort.

MODELING COMPLEX COMPOSITE STRUCTURES USING ANSYS
ansys.com/82style2

Designers must predict how well the finished product will perform in the real world — such as on a race track or road. Predicting failure, delamination, ultimate strength and other development variables is critical before prototype and manufacturing stages. For composites, only a reliable simulation software can provide insight into how and why the layers work. Running simulations with Composite PrepPost helps to avoid costly problems late in design and manufacturing stages that could compromise the entire project.

Using the unique draping capabilities of ANSYS software, the design team can define the exact orientation of every layer. Initially, engineers optimize using modifications to the geometry — such as placing ribs and reinforcement in problematic areas. Later, when they select the layer structure, they can fine-tune the composites design.

The design team is particularly interested in flexibly running different simulations with complex geometries. With Composite PrepPost, design engineers create variations quickly, as the parameters are already defined. The engineering team swiftly builds, runs and modifies simulation models and even obtains cost information, which is particularly useful when working with initial designs and determining how sensitive certain variables are at a specific layer. By identifying this type of information early, savings (time and materials) can be realized downstream. After many investigations and iterations, the result is a ply-book that can be used in the manufacturing stage. ANSYS Composite PrepPost provides the company with a high level of confidence in the ultimate product's integrity and behavior.

Building on its racing heritage, KTM has successfully used composites in the design of both an electric bike and a sports car. (See sidebars.) These are just two examples of how the company has achieved success in the field of high-performance composites.

Support for KTM Technologies GmbH is provided by ANSYS channel partner CADFEM. Parts of this article originally appeared in CADFEM Journal.

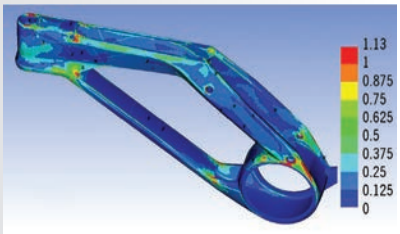
Audi e-Bike Wörthersee

▲ The ultralight carbon frame weighs only 1,600 grams — without sacrificing performance or aesthetic design.

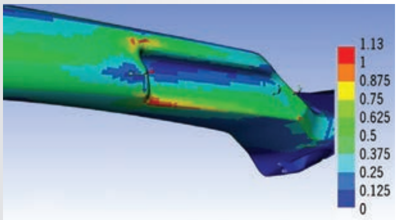
Unveiled in 2012, the Audi e-Bike Wörthersee is a concept electric bicycle, blending lifestyle, action and sport into a rugged e-bike that easily handles even the toughest tricks. This e-bike pushes the limits of design, lightweight construction, e-connectivity and function.

The frame, wheels, handlebars and rear-wheel swing arm are all made of ultra-light carbon-fiber–reinforced plastic (CFRP) to meet structural requirements while still delivering a sleek, forward-looking design. The frame weighs only 1,600 grams: The innovative composite design, though lightweight, delivers optimal placement of reinforcement only to where it is needed to manage static and dynamic loads.

In just 16 weeks, KTM Technologies took the e-bike concept and transformed it to the prototype stage. This included 3-D data, mold–making, production, assembly and even marketing strategies. By embracing a holistic approach, with extensive cooperation among all the different disciplines, KTM designers used ANSYS Composite PrepPost simulation to optimize the design. For example, they optimized the frame by simplifying the design as well as restricting layer overlaps and draping.



▲ Failure simulation of carbon-fiber frame



▲ Failure analysis of handlebars

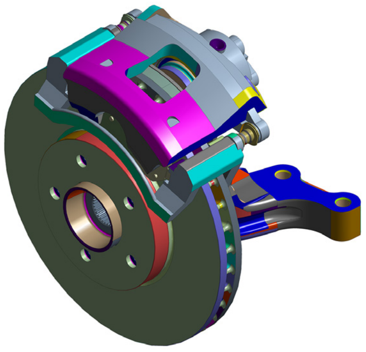
The design of the frame included German Institute for Standardization (DIN) load cases (pedal load, jump, brake load) along with additional load cases from steering stiffness and the adjustable seat. This resulted in modifications of the composites layers' layup by thinning the design at low-stress areas, which contributed to the lightweight design. Small design changes early in the design cycle proactively addressed potential local stresses.

An output of ANSYS Composite PrepPost was a detailed ply-book to support production. Unique draping capabilities showed the exact orientation of every layer. Without ANSYS Composite PrepPost, it would not have been possible to finish this project within the short timeline.

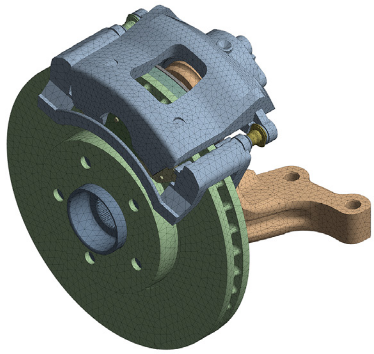
BREAKTHROUGH FOR BRAKE DESIGN

New method simulates brake-squeal problems early in the design process.

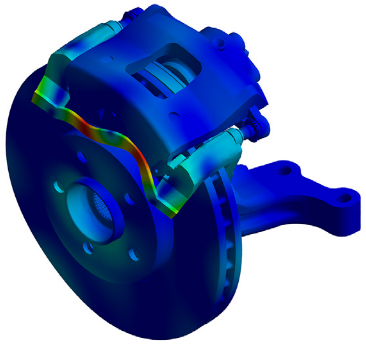
By Greg Roth, Chief Engineer – Engineering Technologies NA, TRW Automotive North America, Livonia, U.S.A., and **Mike Hebbes**, Regional Technical Manager, ANSYS, Inc.



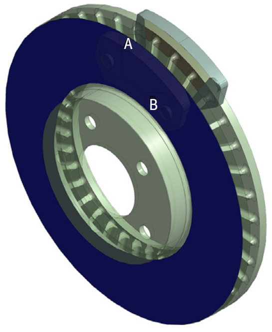
Typical TRW brake system, with disc rotor (green) and ventilation slots around perimeter. The caliper (adjacent pink and grey sections) houses the brake pads, which are visible through the inspection hole. The caliper mount (teal and grey) and steering knuckle (blue) also are shown.



Tetrahedral mesh on disc rotor (green), caliper and caliper mount (both grey), and steering knuckle assembly (tan)



Visualization of complex eigenvalue for unstable mode. Contours of relative deformation on the caliper mount (yellow, red) have been scaled up for clarity.



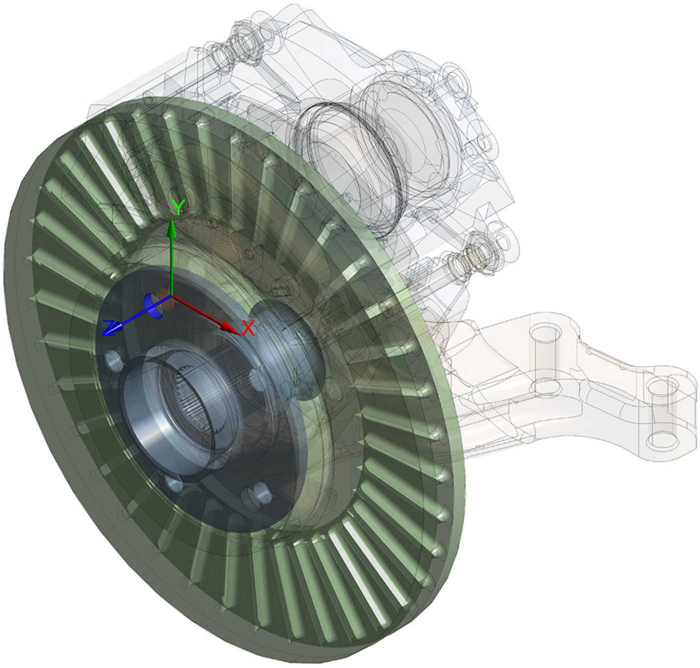
Cutaway of brake pads and disc rotor highlighting one of the frictional contact pairs (marked A and B) between the rotor and one of the brake pads

Friction-induced brake squeal has been a challenging issue for the automotive industry for decades; it has become particularly important today as other sources of noise have been reduced or eliminated. TRW and other brake producers previously relied on a brake-squeal simulation method in which interfaces between brake pads, rotors and other components were manually modeled prior to performing structural analysis. The weakness of this approach is that it requires assumptions of how the components contact each other that then must be validated by physical testing. This takes considerable time and money, and delays the product development process.

More recently, TRW has validated a new method that uses ANSYS Mechanical software to establish the initial contact and compute the sliding contact between the pads and the disc. Simulation studies have determined that this approach accounts for system contact conditions, enabling brake noise to be simulated and reducing the need for physical testing to tune the models. The entire simulation process is contained within a single environment, which saves time by automating many aspects of the process and setting up batch runs for design optimization or manufacturing variation analysis. This method has made it possible to design and build quieter brakes in less time than is possible with traditional methods.

CHALLENGE OF SIMULATING BRAKE SQUEAL

Researchers have estimated that noise, vibration and harshness, including disc brake squeal, generate warranty costs of about $1 billion a year to the automotive industry in North America alone. At least $100 million of those expenses can be attributed to brake squeal. Automobile suppliers and original equipment manufacturers stand to benefit if they can identify the potential for a proposed design to squeal early in the process, before millions of dollars have been invested in detailed design, prototyping and manufacturing tooling. Since it is not feasible to produce every single component to exact dimensions and material



One of the joint interfaces between rotor and hub

TRW engineers wanted to accurately simulate brake squeal without having to spend extra time and money on validation testing.

specifications, it's important to determine whether the small variations that are inevitable in the manufacturing process will cause a percentage of production builds to squeal.

Although all brake squeal causes are not fully understood, it is commonly accepted that such noise is initiated by instability due to the friction forces, leading to self-excited vibrations. This process is inherently more complex than a typical simulation problem that consists of applying a measurable load to a structure. Simulating brake squeal has typically required a tedious mesh generation process of manually creating couplings between the brake pads and discs. Dynamic analysis is used to analyze the eigenfrequencies of the system to determine whether or not squeal will occur.

A big weakness of this method is that factors such as deflection may change the way that the pad and disc surfaces come together. They may contact each other at an angle or with greater or less force than is expected. With the traditional method, the uncertainty in the contact conditions is addressed by building and testing a physical prototype and comparing the measurements against simulation predictions to tune the model. The first try is often a poor match, necessitating that the simulation be run over and over again, adjusting the contacts each time until the predicted results accurately match the physical tests. This approach is both expensive and time-consuming.

DEVELOPMENT OF NEW SIMULATION METHOD

TRW's engineers wanted the ability to accurately simulate brake squeal without having to spend extra time and money on validation testing. The TRW team worked with ANSYS technical services staff to accurately define contact conditions prior to physical testing by using a nonlinear static solution to establish the initial contact and compute the sliding contact between pads and disc. ANSYS software enabled the entire brake-squeal simulation process to be incorporated within the ANSYS Workbench environment, which allowed automating the ability to simulate expected manufacturing variation and determine if the design met robustness requirements.

The new simulation process — jointly developed by TRW and ANSYS — begins with importing the CAD model into Workbench. The production-intent parametric CAD model of the brake assembly incorporates component-level models such as the pad assembly, caliper, rotor and knuckle. Additionally, the component models are created to incorporate manufacturing variability. After the initial import, the software automatically detects and performs setup for the contacts or joints between parts of an assembly. ANSYS meshing technology then provides multiple methods to generate a hex-dominant mesh or a tet mesh, depending on analysts' requirements.

Successfully simulating brake squeal then requires capturing the linear behavior of the structure based on its prior linear or nonlinear pre-loaded status. The TRW team uses linear perturbation analysis to solve a linear problem from this pre-loaded stage — a process that is essentially automated in ANSYS Mechanical. Next, engineers employ a nonlinear static solution to establish the initial contact and compute the sliding contact between pads and disc. The applied stresses and rotation of the disc create the pre-loaded effect, and friction contact generates an asymmetric stiffness matrix during static structural analysis.

In the second phase of the linear perturbation analysis, TRW engineers perform a QR-damped or unsymmetric modal analysis. The eigensolver uses the unsymmetric stiffness matrix generated in the contact elements and may produce complex eigenfrequencies. The results of a perturbation analysis then show the damped frequencies for each mode number along with the stability or real part of the eigenvalues. When the coupled mode shows a positive real value, it indicates instability in the system that may be a source of brake noise or squeal. The analysis results also include the mode shapes, which often provide useful diagnostic information that helps in changing the design to eliminate instability.

The company is confident that it can substantially improve brake quality while reducing engineering costs and lead time.

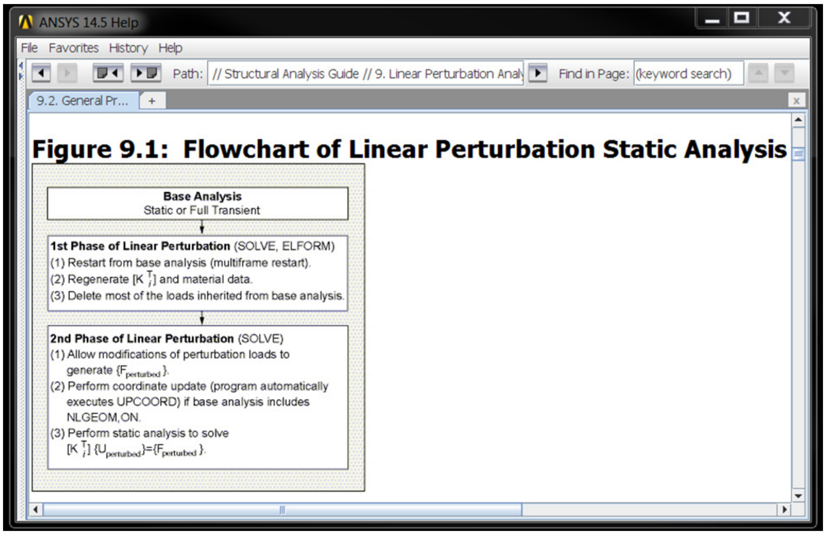


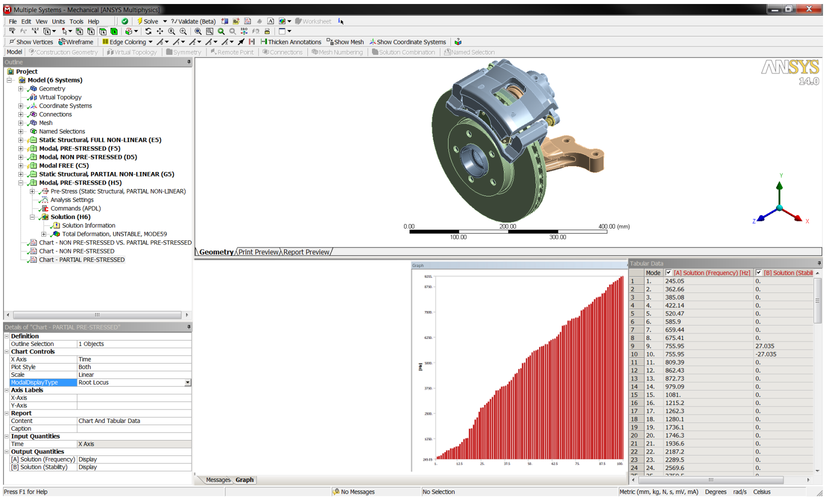
Figure 9.1: Flowchart of Linear Perturbation Static Analysis

Base Analysis
Static or Full Transient

1st Phase of Linear Perturbation (SOLVE, ELFORM)
(1) Restart from base analysis (multiframe restart).
(2) Regenerate $[K_i^T]$ and material data.
(3) Delete most of the loads inherited from base analysis.

2nd Phase of Linear Perturbation (SOLVE)
(1) Allow modifications of perturbation loads to generate $\{F_{perturbed}\}$.
(2) Perform coordinate update (program automatically executes UPCOORD) if base analysis includes NLGEOM,ON.
(3) Perform static analysis to solve $[K_i^T]\{U_{perturbed}\}=\{F_{perturbed}\}$.

Flowchart of linear perturbation analysis



Project
- Model (6 Systems)
  - Geometry
  - Virtual Topology
  - Coordinate Systems
  - Connections
  - Mesh
  - Named Selections
  - Static Structural, FULL NON-LINEAR (E5)
  - Modal, PRE-STRESSED (F5)
  - Modal, NON PRE-STRESSED (D5)
  - Modal FREE (C5)
  - Static Structural, PARTIAL NON-LINEAR (G5)
  - Modal, PRE-STRESSED (H5)
    - Pre-Stress (Static Structural, PARTIAL NON-LINEAR)
    - Analysis Settings
    - Commands (APDL)
    - Solution (H6)
      - Solution Information
      - Total Deformation, UNSTABLE, MODE59
  - Chart - NON PRE-STRESSED VS. PARTIAL PRE-STRESSED
  - Chart - NON PRE-STRESSED
  - Chart - PARTIAL PRE-STRESSED

Details of "Chart - PARTIAL PRE-STRESSED"

Definition	
Outline Selection	1 Objects
Chart Controls	
X Axis	Time
Plot Style	Both
Scale	Linear
ModalDisplayType	Root Locus
Axis Labels	
X-Axis	
Y-Axis	
Report	
Content	Chart And Tabular Data
Caption	
Input Quantities	
Time	X Axis
Output Quantities	
[A] Solution (Frequency)	Display
[B] Solution (Stability)	Display

Tabular Data

	Mode	☑ [A] Solution (Frequency) [Hz]	☑ [B] Solution (Stabil
1	1.	245.05	0.
2	2.	362.66	0.
3	3.	385.08	0.
4	4.	422.14	0.
5	5.	520.47	0.
6	6.	585.9	0.
7	7.	659.44	0.
8	8.	675.41	0.
9	9.	755.95	27.035
10	10.	755.95	-27.035
11	11.	809.39	0.
12	12.	862.43	0.
13	13.	872.73	0.
14	14.	979.09	0.
15	15.	1081.	0.
16	16.	1215.2	0.
17	17.	1262.3	0.
18	18.	1280.1	0.
19	19.	1736.1	0.
20	20.	1746.3	0.
21	21.	1936.6	0.
22	22.	2187.2	0.
23	23.	2289.5	0.
24	24.	2569.6	0.

Damped frequencies generated by partial pre-stressed complex eigenvalue analysis. Mode 9 has a positive real component and is a potential source of brake squeal.

DETERMINING ROBUSTNESS OF A PROPOSED DESIGN

Beyond accurately modeling the physics of brake squeal, another attraction of the new simulation method is its ability to perform studies that determine the impact of small variations in variables such as dimensions, material tolerances and loads. These variables change from build to build, and, even though they remain within tolerance limits, there is a chance that they might increase or reduce the amount of noise produced by the brakes. With previous simulation methods, there was no way to know what these effects would be. The new method makes it possible to evaluate the robustness of a proposed design by simulating whether any squeal would occur if hundreds of thousands of brake units were built.

The TRW team used ANSYS Workbench to determine the robustness of a proposed design as an extension of a single simulation. Using ANSYS DesignXplorer, the team performed a design of experiments (DOE) to create a set of simulations that explore the design space with a minimum number of simulation iterations. When TRW executes an "Update All Design Points" in DesignXplorer, the first set of parameter values is sent to the parameter manager in Workbench. This action drives changes to the model from the CAD system to post-processing. TRW engineers then simulate the new design point, and output results are passed to the design point table to be stored.

This process continues until TRW solves all of its design points. DesignXplorer presents the expected output variation so the engineering team can determine whether or not the design meets robustness requirements. If not, the team looks at the sensitivity plot and other charts to determine which parameters need to be adjusted or tightened to obtain the required robustness. This information helps to reveal which tolerances can be relaxed without compromising the design.

VALIDATION

The new simulation approach proved its capability in a series of simulations that the TRW team validated with physical testing. These confirmation studies demonstrated that it is possible to simulate brake noise and other output parameters, such as mode shapes and frequencies, without using physical testing to calibrate the results. Clearly, the new approach can more accurately model the physics behind brake noise than the traditional simulation method. The ability to incorporate manufacturing variation into simulation and predict what proportion of builds will squeal is another major advantage.

TRW is moving to implement the new method into its design process for future brake programs. The company is confident that it will be able to substantially improve brake quality while reducing engineering costs and lead time.

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

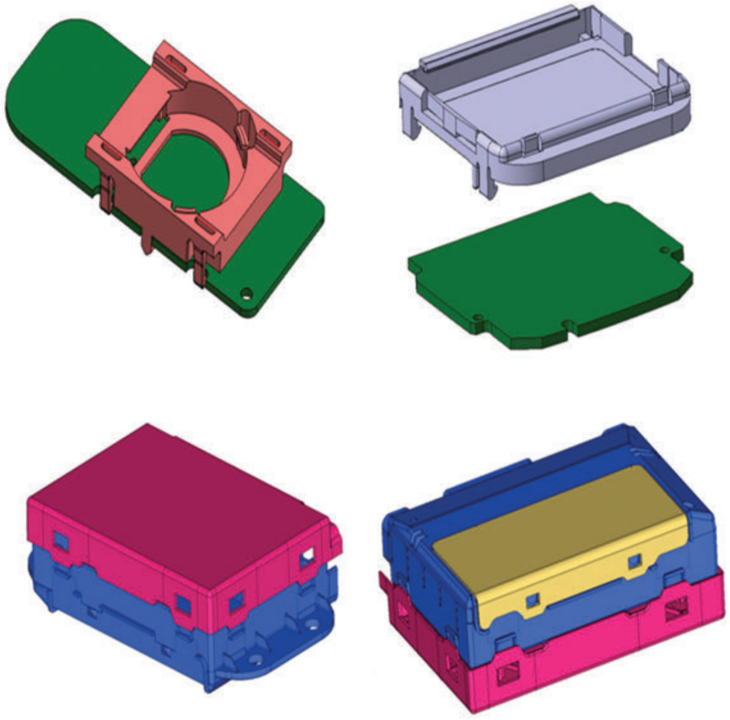
A sweepable volume has the same number of vertices per face and a smooth path from the source to the target face. One advantage of a sweepable volume is that it can be automatically meshed with hexahedron or brick elements that can fill a volume more efficiently. This leads to fewer elements and faster 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.

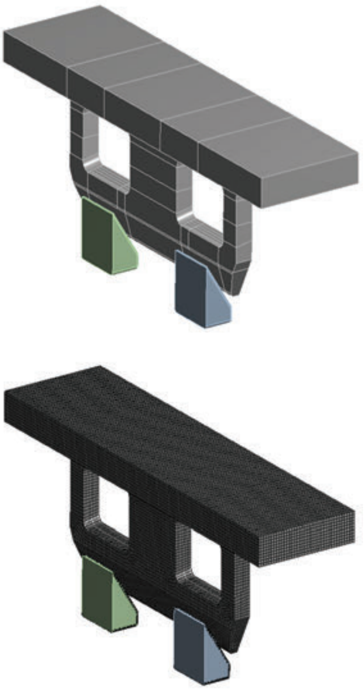
Engineers reduce computational time by defining the parts that are not of primary interest as rigid bodies, without having any significant effect on results accuracy.

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.

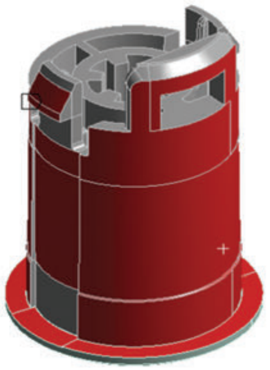
Co-efficient of friction values for interfaces in snap-fit assemblies are frequently not known. However, accurate friction values are often critical to achieve simulation results that correlate with physical tests. If the results do not correlate well with physical tests, the friction co-efficient is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

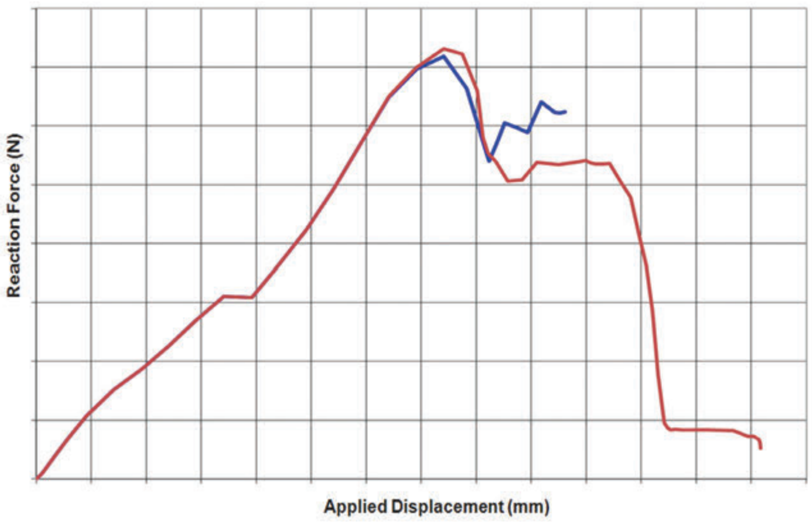
LOADING

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

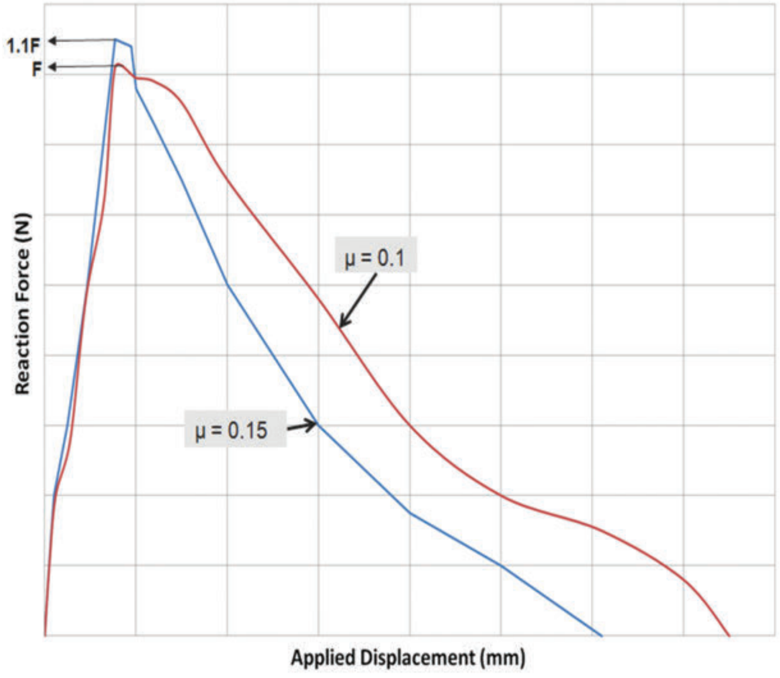
Many snap-fit assemblies experience large displacements for very small loads. Separating the loads into a number of small steps will aid in smooth convergence; it will also capture critical clipping points.

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.

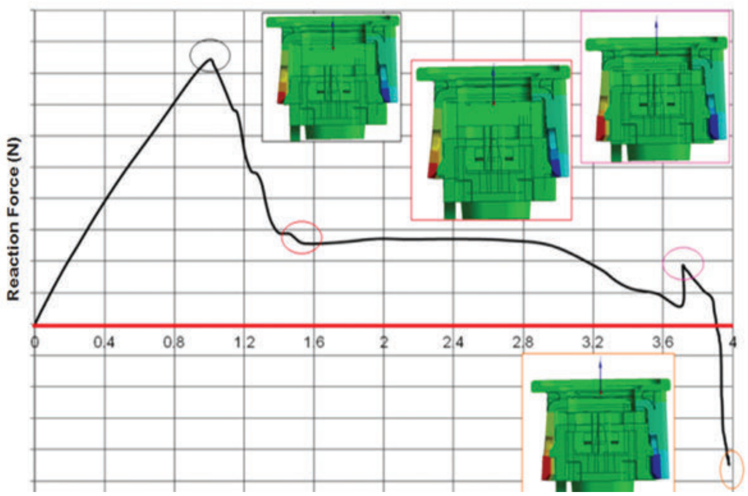


▲ Simulation results for different friction co-efficients. These results will later be compared to test results.

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



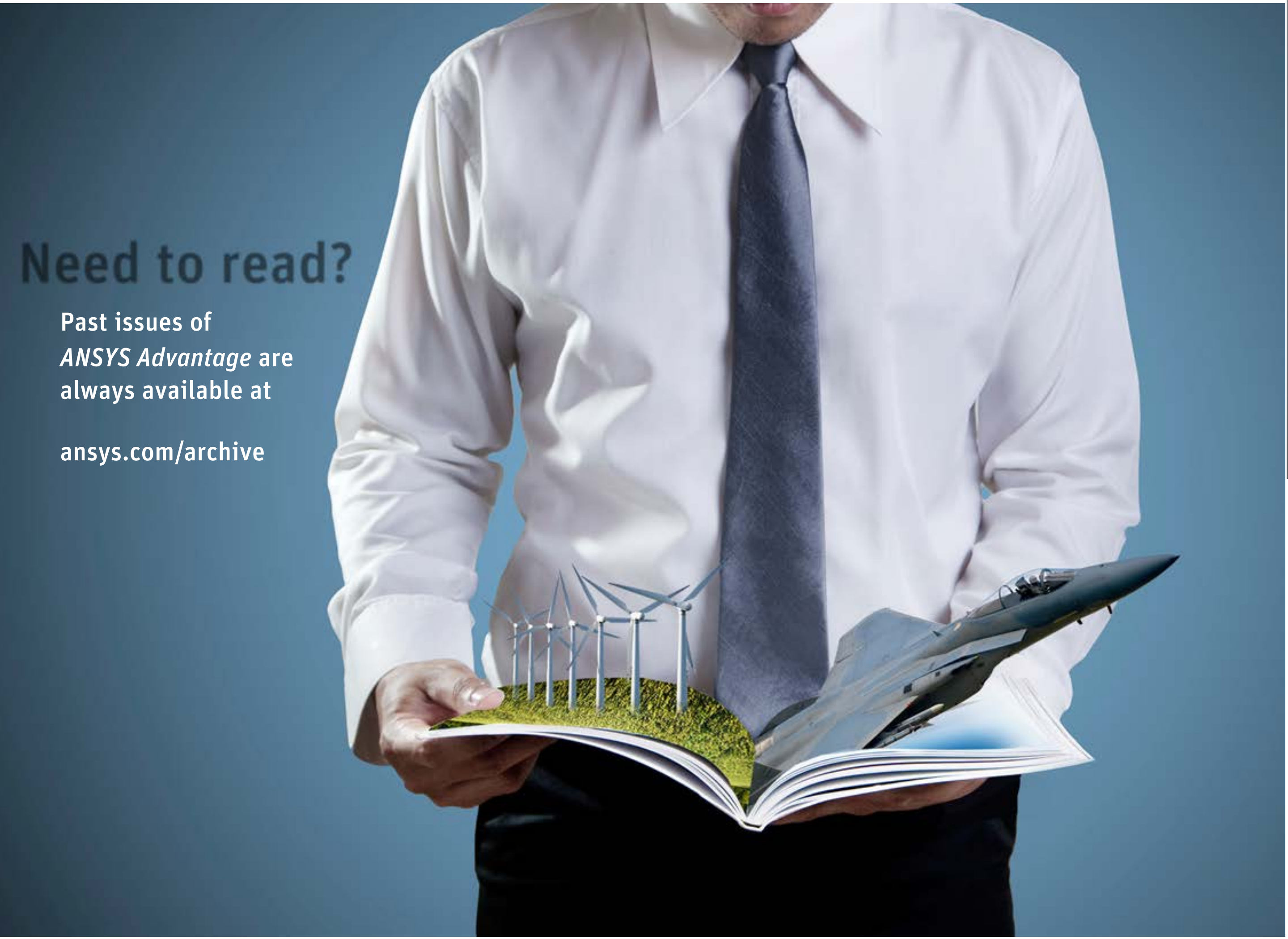
Deformed shapes overlaid on reaction force curve showing multiple peaks

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.



ON THE FAST TRACK

Ferrari pushes the limits of simulation in improving aerodynamic performance of racing cars.

By Enrico Cardile, Aerodynamics and Thermal Management Manager, Ferrari S.p.A, Maranello, Italy

> **The contribution of simulation is huge.**
>
> *– GT Driver Gianmaria Bruni*

Aerodynamics plays a key role in motorsports. Ferrari S.p.A. has made dramatic improvements in its racing cars' aerodynamic performance by combining computational fluid dynamics (CFD) simulation and wind-tunnel testing. Ferrari engineers have extensively automated the simulation process and run many design iterations to explore the design space and improve speed, reliability and safety. It takes about three to four weeks to arrange a session in the wind tunnel, while company engineers can perform more than 100 CFD simulations in the same time period. Simulation dramatically increases the number of different

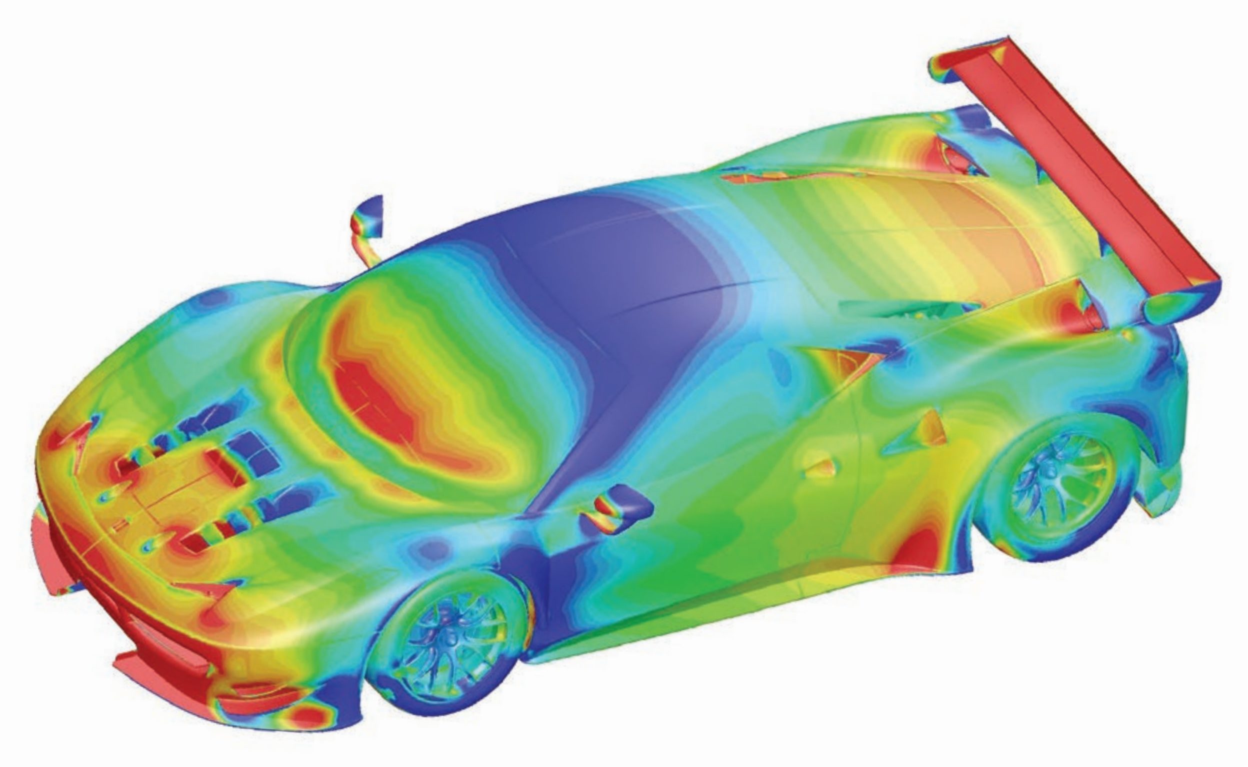
Simulation enables substantial performance improvements that have played a key role in Ferrari's many track victories.

aerodynamic alternatives that can be evaluated, enabling substantial performance improvements that have played a key role in Ferrari's many track victories. "The contribution of simulation is huge," said GT Driver Gianmaria Bruni, winner of the 2012 24 Hours of Le Mans in a Ferrari 458 Italia.

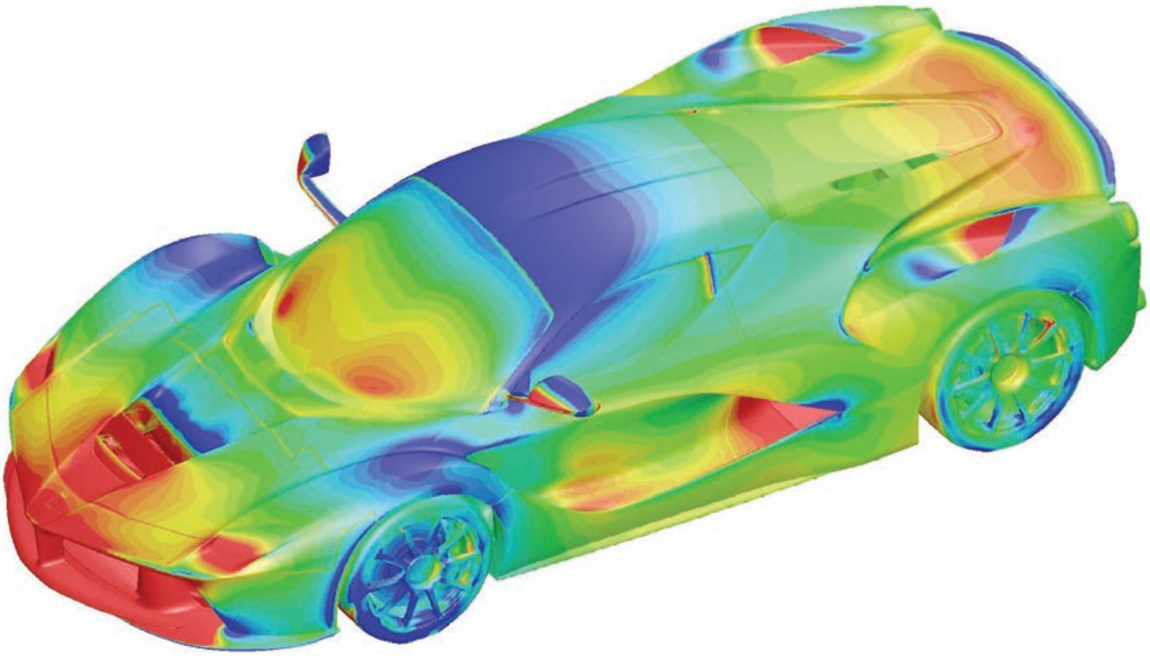
GT2 RACING

Ferrari has been involved in racing since the company began, competing in a wide range of categories. Ferrari's current GT2 entry is the 458 Italia GT2, which is based on the 458 Italia production model. Aerodynamics plays a major role in the design of these cars because the shape of the upper part of a GT car body generates lift, so the underbody must be designed to create down-force. This increases the tires' gripping capabilities during braking and cornering, without increasing drag. Ferrari improves down-force in GT2 cars by smoothing out the underbody and adding diffusers at the rear to intensify air speed and mass flow under the car. A diffuser ejects air from the underside of the car, causing an increase in velocity and a reduction in pressure of air below the car. The slower-moving air above the car generates a higher pressure, and the resulting pressure differential pushes the car onto the ground.

On the latest 458 Italia GT2 model, Ferrari engineers performed hundreds of CFD simulations with ANSYS Fluent to optimize the aerodynamic performance of the car. In general, the process begins when the design team provides a proposed design in the form of a computer-aided design (CAD) file. An analyst then manually generates the surface mesh, the only part of the process that is done manually. Then an automation script takes over and executes the entire simulation process, starting with generating a volume mesh based on the surface mesh, specifying boundary conditions, and running the CFD solver. In early stages of the design process, analysts typically evaluate one proposed design at a time and closely examine flow speed and direction as well as pressure around the body to understand the performance of the design and how it might be improved.



▲ **High down-force configuration of F458 GT2**



Low down-force configuration of LaFerrari

Ferrari engineers performed more than 1,000 CFD simulations on LaFerrari, saving between 40 and 50 hours of wind tunnel testing.

OPTIMIZING THE DESIGN

Once analysts gain a general understanding of flow patterns and which design parameters have the most impact, they set up a design of experiments (DOE) using the ModeFRONTIER® optimization tool, which runs tens to hundreds of simulations without user intervention to evaluate the design space. ModeFRONTIER provides several different optimization algorithms, including the response-surface method (RSM) that is fitted to the data points revealed by the DOE. This technology allows Ferrari to explore the design space with minimal computational effort. ModeFRONTIER also provides genetic algorithms to evolve a group of candidate designs toward better solutions. The simulation is run on a high-performance computing (HPC) compute cluster. ANSYS Fluent splits up the mesh and data into multiple partitions, then assigns each mesh partition to a different compute node.

The position of the car's components often constrains the aerodynamic design. For example, if engineers are working on the rear diffuser, that part's maximum expansion is limited by the presence of the frame and the muffler. So they configure the optimization tool to examine only expansion angles and curvatures of expansion for the diffuser that can be accommodated without interference. Sometimes the aerodynamics team removes constraints to determine if a large improvement might be achieved in the absence of a constraint. In that case, the aerodynamics team meets with the design department to see if the design can be altered to remove the constraint.

Ferrari engineers have used these methods to optimize the down-force on the 458 in a number of areas. They have applied CFD to evaluate vortices under the body of the GT2 cars and evolve the body design to minimize the vortice's impact. In addition, they have optimized brake cooling inlet and outlet ducts with simulation. The geometry of the brake cooling ducts has a critical impact on brake performance as well as on the down-force on the car's front axle. The complexity of the brake cooling ducts' geometry makes the ducts very difficult to evaluate in a wind tunnel, so these critical areas are designed nearly entirely with simulation. CFD analysis showed that even the side mirror's design is closely related to the shape of the engine air intake. By examining the streamlines around the side mirror on the 2014

▲ Ferrari GT2 racecar

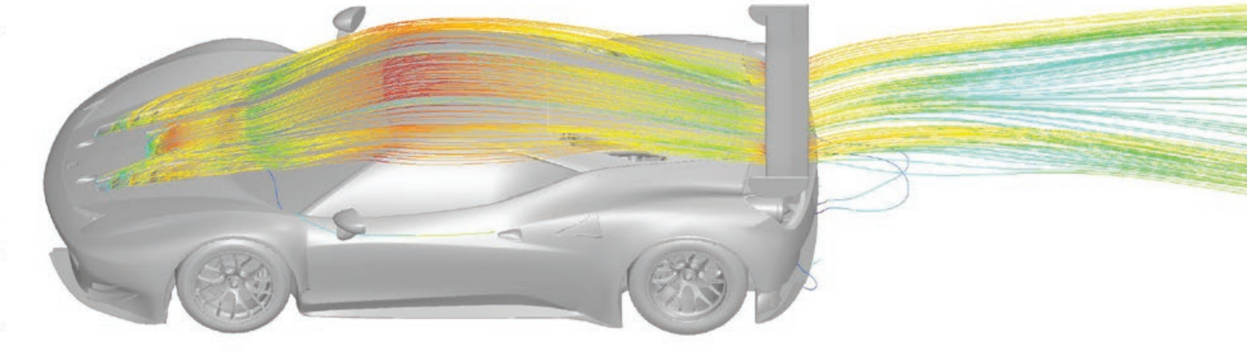
GT2 model, engineers modified the shape of the mirror to get the best performance without having a negative impact on the engine air intake.

LaFERRARI

Ferrari engineers used ANSYS Fluent to optimize the aerodynamic performance of LaFerrari — a limited production hybrid sports car that was officially unveiled at the 2013 Geneva Auto Show. Ferrari engineers performed more than 1,000 CFD simulations on LaFerrari, saving between 40 and 50 hours of wind tunnel testing. LaFerrari has active aerodynamics devices in the car's front and rear, which help to increase down-force and reduce drag. The front of the car has three flaps: two lateral flaps in the front diffusers and a variable flap in the radiator. The rear of the car has a wing and an air-active rear spoiler. Ferrari engineers used CFD to optimize the aerodynamic performance of each of these devices. They ran a design of experiments that evaluated the aerodynamic performance with the car at different speeds; in different pitch, roll and yaw positions; and with the variable flap in different positions.

Time is a critical success factor, so development schedules are very tight. For this reason, Ferrari engineers need to perform simulations and tests as quickly, reliably and efficiently as possible to better drive style and design, ensure accuracy, and achieve performance targets. HPC solutions from ANSYS, including recent software advances — such as improved parallel scaling performance for very large simulations, hybrid parallelism for multicore processors within clusters and support for parallel file systems — enable the Ferrari team to improve car performance while adhering to development schedules.

Every new racing car developed by Ferrari must rise to a new level of aerodynamic performance to match the successful results it has achieved on the track over the past 80-plus years. The time required for wind-tunnel testing makes it impossible to achieve performance targets within the allotted timeframe. By combining CFD to understand the application, evaluate the design space, and iterate to an optimized design with wind tunnel testing for verification and validation, Ferrari is able to stay at the forefront of aerodynamic performance. The results are victory after victory in prestigious races. For example, the 458 Italia GT2 won the 2011 Petit Le Mans, the 2011 Intercontinental Le Mans, the 2011 Le Mans Series and the 2012 24 Hours of Le Mans. The aerodynamic technology developed for race cars is quickly transferred to road cars. "Simulation has been vital to our victories," Bruni concluded.



▲ Streamlines from radiator outlet of F458 GT2

CLEANING UP

Magneti Marelli reduces engine emissions and improves fuel efficiency by modeling the complete engine cycle.

By Nazario Bellato, Simulation Manager
Magneti Marelli Powertrain S.p.A.
Bologna, Italy

In 2009, the European Union ratified the Renewable Energy Directive (RED), which requires that 10 percent of motor vehicles run on renewable energy by 2020. Biofuels consisting of ethanol or an ethanol blend are one of the most practical renewal fuels for motor vehicles. In Brazil, the use of biofuel is already common; but most current vehicle engines do not deliver optimal performance, fuel economy or exhaust emissions with biofuels.

Magneti Marelli, a leading supplier of automotive components and systems around the world, is working to develop fuel injection systems that will improve the performance of existing engines running on biofuels. An Italian–Brazilian Magneti Marelli team is working with the support of ANSYS to use 3-D computational fluid dynamics (CFD) to simulate the complex operation of an internal combustion engine (ICE) to evaluate many virtual prototypes in the time that would be required to build a single physical prototype.

ADVANCED CFD TECHNOLOGY FOR PORT FUEL INJECTION ENGINES USING BIOFUELS

Simulation of an internal combustion engine analysis is time consuming and complex, and obtaining accurate results requires an appropriate simulation tool and engineering expertise. Increasingly complex control systems make it more difficult than ever to predict in-cylinder mixture formation, combustion and emission in these engines. Acceptable resolution of the engine flow and combustion requires large hybrid meshes for each configuration with associated computing overheads. Once the analysis has been set up, it takes many hours or days of computing to solve the model and evaluate the results. The results include large data sets that require considerable time and effort to evaluate and then generate useful information that can be fed back to the design process. In existing engines, the goal is to improve engine performance by optimizing the geometry and operating parameters of the air intake manifold and injectors. This requires comparing the analysis results of many different parametric configurations.

FUEL INJECTION SPRAY MODELING

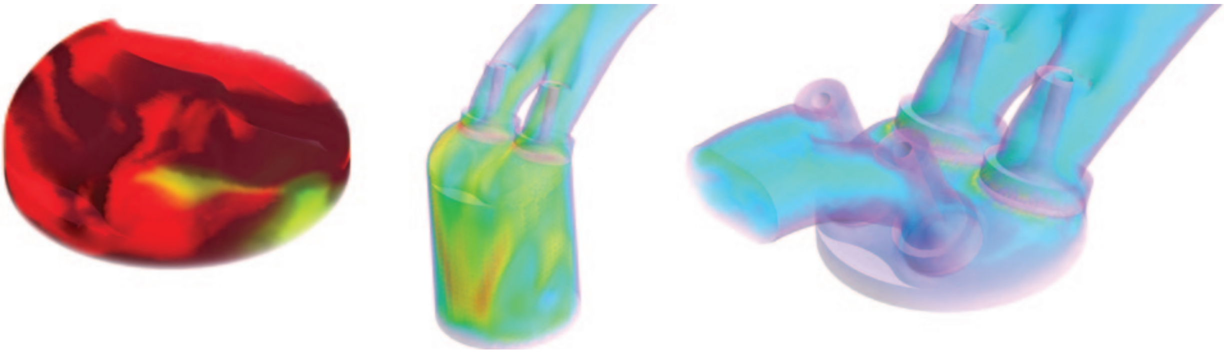
Magneti Marelli engineers have overcome these challenges by combining automatic remeshing and parametric analysis methods using ANSYS CFX with a moving mesh to simulate the different states of the engine cycle, including fuel injection, spray formation and particle breakup. The most important part of engine simulation is modeling the fuel injection process. This requires accurately modeling the flow in the nozzle, including cavitation, spray motion, breakup and evaporation, and film formation and evaporation. The spray algorithm parameters are tuned with experimental data to obtain satisfactory quantitative prediction, especially when defining a new design or different operating conditions. In spray simulation, the most important characteristics are spray penetration depth and spray angle. The engineering team defines spray penetration depth as penetration depth in a specified direction. The spray cone angle is the radial expansion of the spray, measured at the end of the injection cycle. Predefined post-processing routines allow engineers to make decisions while the simulation is running.

Engineers use fluid dynamics to tune several aspects of fuel injection — the design of the injector, the spray targeting and the spray–wall interaction. This process is important during combustion mixture formation because wall spray growth and wall film dynamics influence combustion efficiency and pollutant formation, especially in port fuel injection engines using biofuel. Fuel deposited on the intake wall creates engine control difficulties because not all the injected fuel moves immediately into the combustion chamber. Fuel sediment is progressively transported to the combustion chamber, making it difficult to control the amount of fuel that is actually injected into the chamber, resulting in reduced engine performance and increased fuel consumption and emissions. Leading-edge fuel injectors spray fuel directly on a specific zone on the intake valve and its valve rod to reduce liquid film on the intake walls.

Simulation using software from ANSYS can accurately predict mixture formation, breakup phenomena, evaporation, and droplet–droplet and wall–droplet interactions, as well as enable the comparison of alternative engine designs with respect to all these different factors. The workflow within the ANSYS Workbench environment allows Magneti Marelli engineers to automatically investigate multiple parametric design variations. The company uses design of experiments (DOE) to reduce the number of simulation runs needed to explore the complete design space. DOE examines first-order, second-order and multple-factor effects simultaneously with relatively few simulation runs. It is possible to optimize the design with far fewer simulation runs — and with a higher level of certainty and in less time than the traditional one-factor-at-a-time approach.

Simulation using software from ANSYS can accurately predict mixture formation, breakup phenomena, evaporation, and droplet–droplet and wall–droplet interactions.

High turbulence levels facilitate fine mixing and atomization of fuel. One of the most valuable methods for evaluating the turbulence levels is the tumble



Fluid dynamics can be used to model fuel injection — the most important element of engine modeling. Images illustrate simulation at different stages of the engine evolution: exhaust and intake valves both open (left); intake valve open (center); both exhaust and intake valves closed (right) with combustion.

index approach. The index quantifies the relative amount of tumbling or swirling flow in the engine. The tumble index is most important at low engine rpm speed, at which it can be difficult to ensure complete combustion and subsequent rapid flame propagation. The tumble index can also be used to confirm that turbulence levels at the end of the compression stage are adequate for combustion. Finally, the index provides an indirect indication of the injector positioning.

INTEGRATED ENGINE MODELING

To understand the impact of flow on engine performance, engineers couple CFX with GT-Power engine simulation software from Gamma Technologies. GT-Power predicts engine variables such as volumetric efficiency, torque and power based on combustion chamber architecture and various processes such as ignition, mixture formation and combustion. Fluid flow–driven predictions of the engine's maximum power correlate very closely to experimental measurements.

In addition, engineering staff use Workbench fluid–structure interaction (FSI) capabilities to automate the process of transferring temperature information from the fluids simulation into thermal analysis to determine the temperature distribution on a structure. The temperatures are, in turn, used in thermal analysis to identify thermal loads. These loads are applied to static structural analysis to ascertain stresses and deformations. Finally, prestressed modal analysis calculates mode shapes and frequencies of the fuel injector system components. Magneti Marelli also uses ANSYS electromagnetic simulation software to define the magnetic injector circuit; the company is beginning to use the tool for hybrid engine components.

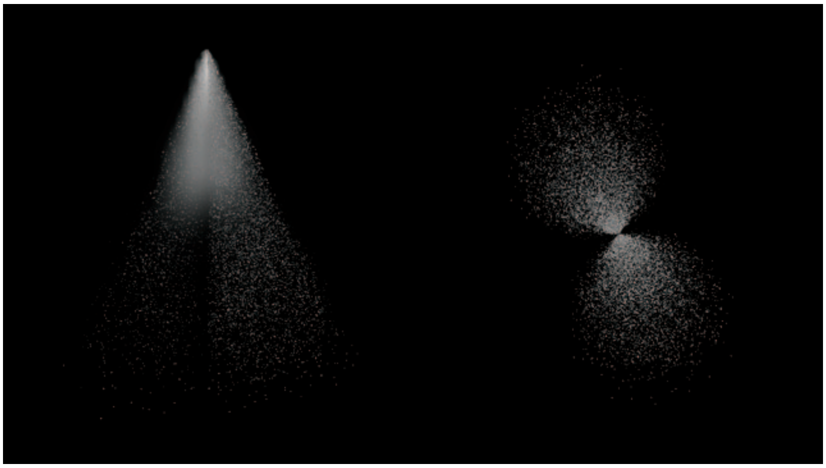
Reliable simulation from ANSYS is helping Magneti Marelli to reduce the time required to develop innovative engine components that improve fuel efficiency and reduce emissions. Λ

Reference

www.magnetimarelli.com

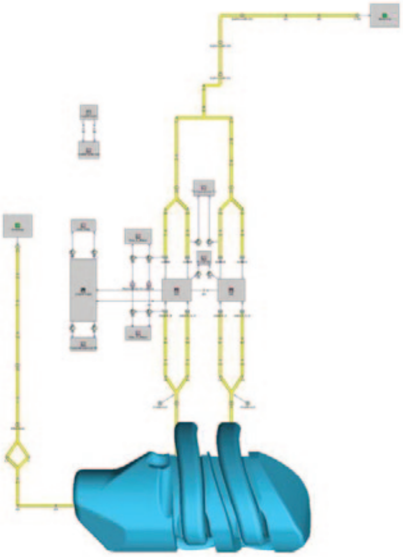
The workflow within the ANSYS Workbench environment allows Magneti Marelli engineers to automatically investigate multiple parametric design variations.

ANSYS is helping Magneti Marelli to reduce the time required to develop innovative engine components that improve fuel efficiency and reduce emissions.



ANSYS CFX spray pattern evolution: front view (left) and top view (right)

Coupling fluid dynamics to GT-Power engine simulation provides accurate maximum power predictions.



Cummins ISX15 heavy-duty engine

Cummins ISF3.8 light commercial vehicle engine

Image © iStockphoto.com/PeskyMonkey

Cleaner, Greener Engine Design

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

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

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

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

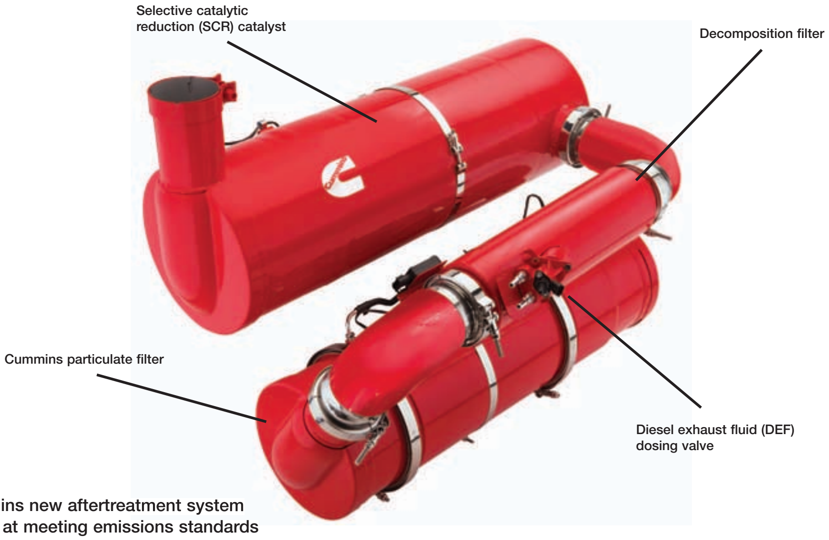
Cummins' recent product development efforts target a great deal of attention on fuel economy and emissions — and with good reason. Government environmental standards grow more challenging every year. Because commercial trucking is a low-margin business, every improvement that Cummins makes in fuel economy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

Building the Truck of the Future

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

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

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



Cummins new aftertreatment system aimed at meeting emissions standards

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

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

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

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

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

Using Simulation to Rev Up Ongoing Engine Improvements

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

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

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

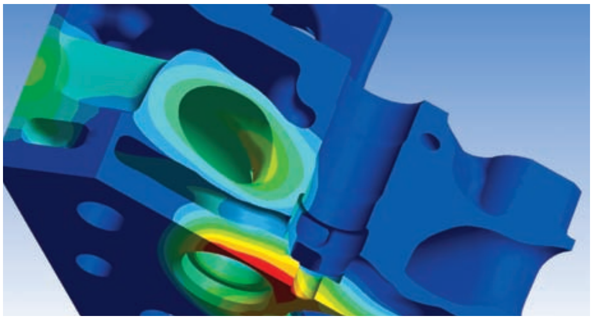
By using software from ANSYS to predict maximum temperatures and temperature distribution within its new diesel particulate filter under diverse conditions, Cummins engineers can ensure that the manufactured component will have acceptable durability.

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

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

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

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



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

Shifting Gears: ANSYS Creates a Cultural Change

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

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

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

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

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

PLAYING IT Cool

CFD SIMULATION OF DRIVE UNIT COOLING HELPS TO IMPROVE RELIABILITY.

By Tadashi Yamada, Drivetrain Unit Engineering Design Division
Toyota Motor Corporation, Toyota, Japan

The cooling performance of oil is critical to the functionality and durability of vehicle drive units, such as transmissions and differentials. Traditionally, automotive R&D teams evaluate cooling performance by building prototypes, installing them in a vehicle, and conducting tests in wind tunnels. The use of free-surface multiphase flow modeling with high-performance computing (HPC) has made it possible to accurately predict oil cooling performance via readily available computing resources. Automotive leader Toyota uses this approach to evaluate more design alternatives in the early stages of the product development process.

A typical drive unit consists of a case containing rotating internal parts, such as gears and shafts supported by bearings that transmit power. These are surrounded by oil and air. The oil serves various functions, including lubrication, power transmission and cooling. The major heat sources within the drive unit include meshing of the gears, sliding friction between bearings and shafts, and stirring oil as a result of gear movement. The heat generated is conveyed to and through the oil to the internal surface of the case; from there, it goes to the case's external surface and surrounding air. Oil flow patterns within the transmission are critical to efficient lubrication, power transmission and cooling performance, and to avoid negative effects, such as churning loss, in which friction between the oil and gears revolving at high rpm can consume several horsepower.

Simulating the cooling capacity of a drive unit requires predicting internal oil flow patterns involving free surfaces, external air flows, and the complex three-dimensional flows of heat from the oil to the air. A key difficulty is that external air flows can be resolved with sufficient accuracy only by modeling the entire vehicle,

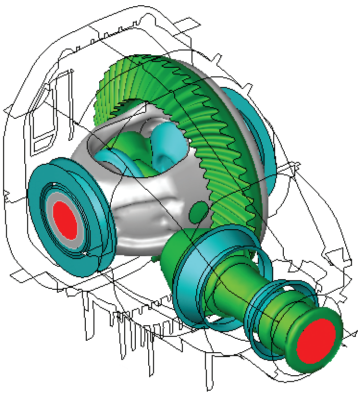
This approach allows Toyota to evaluate more design alternatives in the early stages of the product development process.

whereas the internal temperature distributions must be analyzed for about an hour until a saturation temperature is reached. Until recently, this combination of complex physics, large spatial scale and long duration required such extensive computing resources that simulation could not be completed within a time frame that would positively affect the design process.

Advancements in physical models and HPC performance in the latest generation of CFD software spurred Toyota engineers to initiate new efforts to simulate oil cooling performance in drive units. A strong coupling approach in which the entire computational domain is formulated is computationally very expensive. For this reason, the internal oil flows and vehicle airflows were simulated separately but coupled to exchange data to determine the entire system's behavior.

The internal oil flows were solved to obtain the heat transfer coefficients between the internal components and fluids, and between the fluids and the case. The whole-vehicle air flow simulations produced heat transfer coefficients between the case and the external air. Toyota engineers used the results of the internal and whole-vehicle simulations along with the heat generation rates of various components as boundary conditions for a heat calculation model consisting of only solid parts of the unit, with oil temperature as an unknown variable. The model was iterated until it converged on a solution that achieved energy balance to calculate the temperature distribution of the oil and internal components. Toyota used ANSYS Fluent to solve the internal fluid flow.

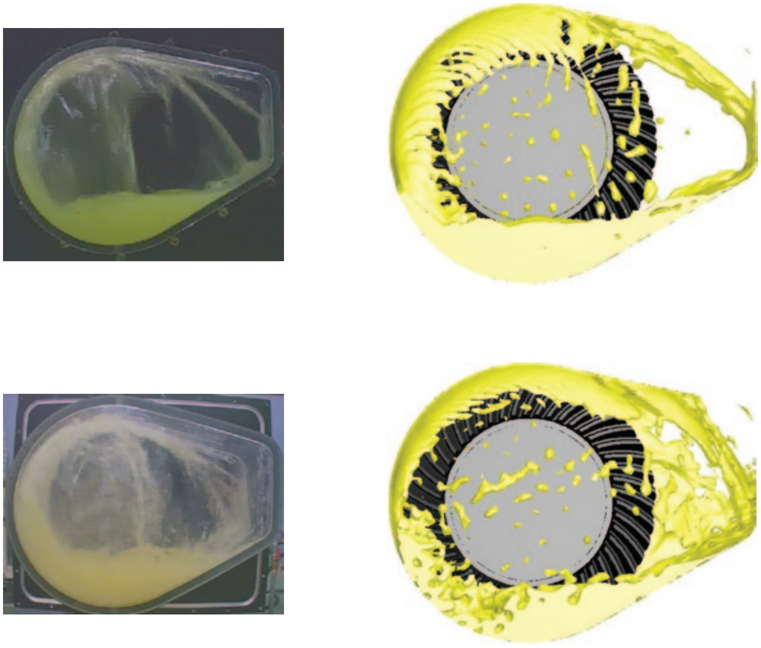
Within the drive unit, as the rotating parts stir the oil, free surfaces are formed at the interface between the oil and air. Engineers used the volume of fluid (VOF) technique within Fluent to track the surface as an interface through a grid and apply boundary conditions at the interface. They employed an explicit geo-reconstruction scheme to solve the interface behavior, as it provides the most realistic interface between phases. Because of the importance of the gear tooth geometry on the free-surface formation, they modeled the tooth geometry



Simplified differential unit used for correlation with CFD simulation

Oil distribution calculated with CFD using VOF model to identify free surfaces as accurately as possible, and the computational grids surrounding the gears were rotated using a moving mesh capability.

Using the von Karman analogy between induced drag on wings and wave drag on bodies, engineers calculated local heat-transfer coefficients from the transient internal oil flow simulation. Then they applied a user-defined function (UDF) to time-average the results. This method was validated by physical testing on a differential drive unit equipped with a single hypoid ring gear. The behavior and flow velocity of the oil stirred by the ring gear were measured and compared to CFD predictions. The behavior of the main flow of oil stirred up by the gear, the amount of oil accumulated on the pan, and the geometry of the free surface correlated well between testing and CFD through multiple revolutions. The team noted relatively minor differences in how droplets and thin films were scattered. Laser Doppler velocimetry (LDV) measurements agreed with the CFD flow velocity predictions for both the magnitude and the direction of flow in the gear stir-up periphery where rapid flow occurred.



Comparison between physical measurements (left) and CFD visualizations (right) at 550 rpm (top) and 1,000 rpm (bottom) shows good agreement.

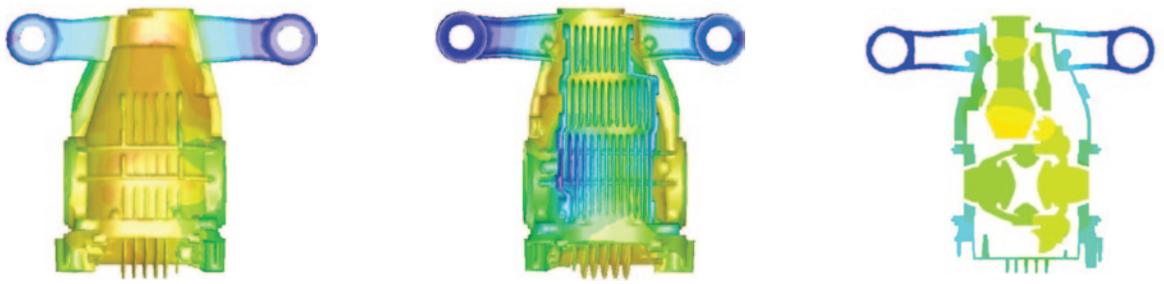
Time-averaged local heat-transfer coefficients between internal parts and oil, and between oil and case

The results demonstrated good agreement in heat flux distribution and temperature distribution between the actual measurements and simulation, illustrating the value of this simulation technology in the product development process.

Based on successful correlation of the simplified drive train, engineers developed a model of a real rear independent-suspension differential unit with about 1.3 million cells. The model was solved on an HPC cluster consisting of a network of about 80 personal computers; it was used to determine the oil distribution and local heat-transfer coefficients between the internal parts and the oil, and between the oil and the case. The results again demonstrated good agreement in heat flux distribution and temperature distribution between the actual measurements and simulation, illustrating the value of this simulation technology in the product development process.

Visualization of the results showed that there was sufficient cooling in the bottom of the unit, where the external airflow velocity is high, but insufficient cooling in the upper front section. Based on these results, Toyota modified the simulation model to evaluate the effect of adding cooling fins to areas where oil temperature was high and removing fins from areas where the simulation showed that they were not needed. Rerunning the simulation showed that this change significantly improved the surface temperatures in the differential drive unit while also reducing case weight.

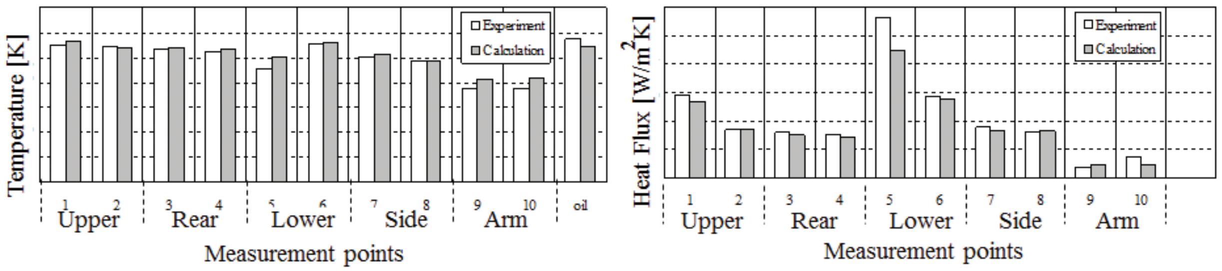
This example demonstrates the ability of simulation to enable engineers to



Top

Bottom

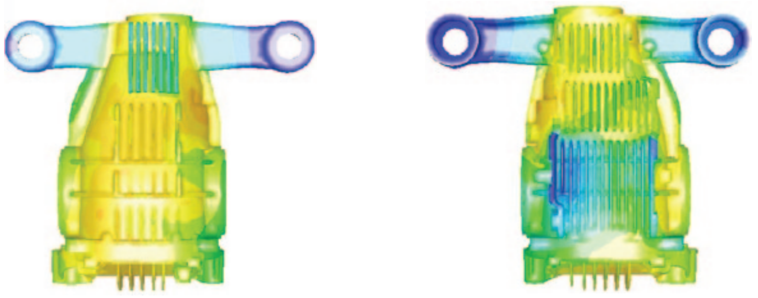
Cross section



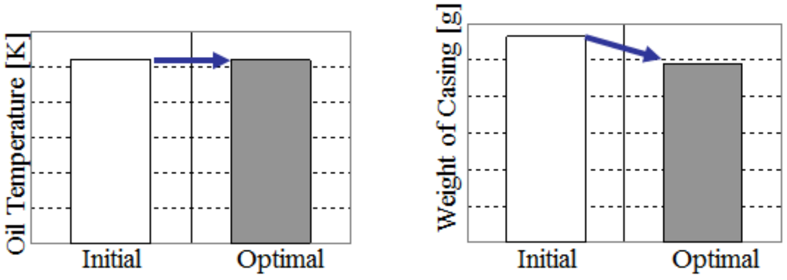
Heat calculation model generated predictions of heat flux distribution and temperatures, which agreed well with measurements.

The team reduced the weight of the drive unit while improving its cooling capacity.

evaluate design alternatives in a fraction of the time and cost required for physical prototypes. Simulation also provides more diagnostic information than can be obtained from physical testing, such as the ability to determine flow velocity, pressure and temperature at any point in the computational domain. As a result, the team reduced the weight of the drive unit while improving its cooling capacity. In the near future, Toyota plans to extend its simulation capabilities to model other types of drive units and to perform transient simulation, which should offer improved accuracy.



Surface temperatures were reduced in the model on the right after redistributing cooling fins based on earlier simulation results.



Design changes made with input from CFD simulation maintained the existing optimal oil temperature while significantly reducing case weight.

SAFE AUTOMOBILE CONTROLS

Subaru uses SCADE software to develop safe and reliable electronically controlled circuits and systems for hybrid-electric vehicles.

By Masaru Kurihara, Deputy General Manager, Electronics Engineering Department,
Fuji Heavy Industries Ltd., Tokyo, Japan

Market pressures to increase fuel economy, maintain safety and provide entertainment are forcing the auto industry to develop automobiles that are increasingly complex and adaptable. Vehicles are now largely computerized, and the electronic control unit (ECU) that manages the systems in each model is governed by complex software. New hybrid-engine vehicle (HEV) technologies rely on extensive circuitry and software. In an HEV, a central computer manages both a traditional combustion engine system and an electric motor via ECU.

Because of the need to continually juggle costs and design requirements, the automotive industry employs AUTOSAR (a development partnership of electronics, semiconductor and software organizations that provides standards to manage growing electronics complexity in this industry) standards and a methodology called model-based development or design (MBD). MBD requires design engineers to use a common design environment that supports model integration and virtual real-time testing of the entire system.

Subaru®, the automotive brand of Fuji Heavy Industries (FHI) Ltd. — a comprehensive, multifaceted transport equipment manufacturer — recently started its own HEV and electric vehicle (EV) programs and adopted the MBD approach for all upcoming development projects, like the Subaru XV. In the company's search for a development environment tool that meets MBD requirements, Subaru engineers evaluated the SCADE software modeling tool from Esterel Technologies, a wholly owned ANSYS company. SCADE provides an intuitive graphical interface so system and software engineers can easily integrate and verify their models. C source code is generated automatically from the models produced in SCADE. This minimizes the chance of programmer error and automatically incorporates SCADE's strict standards and safety requirements.

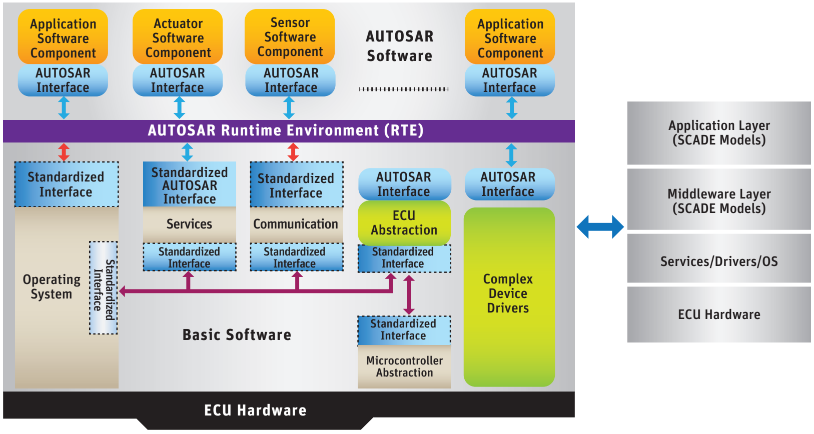
Vehicles are now increasingly complex, adaptable and largely computerized.

To continually juggle costs and design requirements, the automotive industry employs AUTOSAR standards and model-based design.

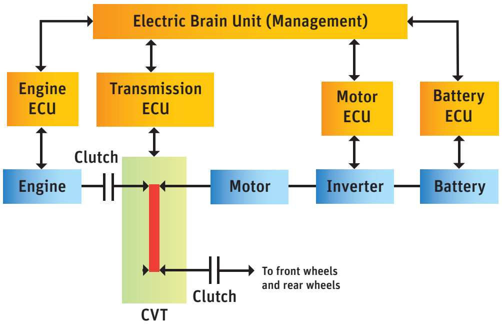
REORGANIZATION OF SOFTWARE ARCHITECTURE

Since HEV development was quite new to Subaru, engineers had the opportunity to create the software architecture from the ground up. Applying AUTOSAR standards to the new ECU reduced basic software development cost and time. However, an even more challenging and important goal was to ensure that the software components could be reused for future projects. All requirements for the ECU are marked as either application software or basic software to configure three layers of software, as per AUTOSAR architecture. Services, drivers and the operating system (OS) are components of basic software; they are created by hand-coding or are available as commercial off-the-shelf (COTS) products. Subaru focuses on pure application design according to the requirements, and this facilitates the deployment of SCADE since the software's models are independent from the OS and hardware-dependent implementations.

When SCADE is deployed, interface nodes for the middleware layer are created automatically by Subaru's own Java® utilities based on Eclipse APIs that analyze the definition files written for basic software. Data types and data structures are extracted from the definition files and defined as SCADE types in the SCADE project. Bridging the gap between application and services with a middleware model layer minimizes human error. When the interface between the middleware layer and the basic software layer is changed, the definition of SCADE types is automatically updated in a consistent way. SCADE's semantic checker continually verifies the SCADE type definition with the models to eradicate modeling error.



Software architecture in EBU control software is layered to comply with the AUTOSAR standard.



SCADE is deployed for the hybrid vehicle management system named electric brain unit (EBU) for a production vehicle released to the market in 2013. In this schematic, the electric brain unit manages several ECUs.

SOFTWARE DESIGN PROCESS WITH SCADE

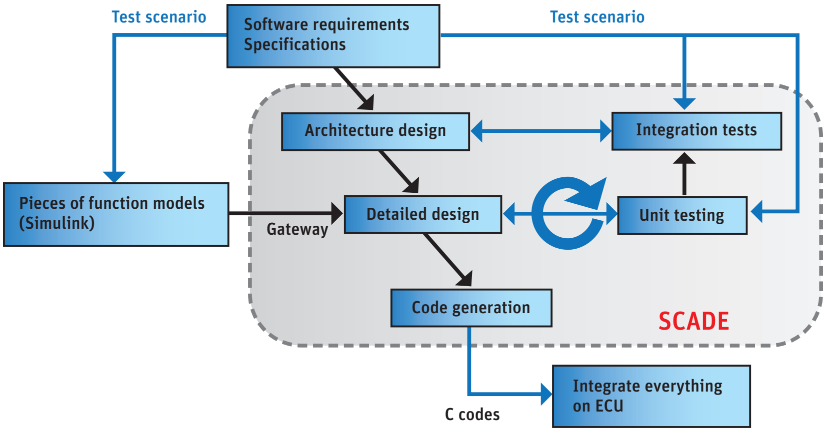
Once the SCADE types are defined, Subaru can execute a set of small cycles iteratively until a detailed design and verification process is completed. The software development process with SCADE at Subaru starts with development of test scenarios based on software requirements and conversion of many pieces of functional models (Simulink®) designed by system engineers into SCADE models via a Simulink gateway. Once the models are converted successfully, they are integrated into a safe SCADE architecture model. To maintain consistency of the conversion from Simulink into SCADE, engineers feed the same test scenarios developed for functional model design simulation into SCADE for unit testing. When the detailed design is completed with unit tests, C code is generated from the integrated model to pass to the EBU test.

SAFE ARCHITECTURE DESIGN WITH SCADE

Due to the modularity of each SCADE node, functional part models are verified by unit testing. The most critical issue for safe software architecture is to ensure that all data are safe when the application operates under multi-task execution. Introducing a safe partitioning architecture from a COTS OS into the automotive software is not easy because of the trade-off between cost and functionality;

multi-task execution is mandatory for optimized execution. To satisfy the trade-off issue, a safe SCADE architecture model is designed.

A root node is commonly deployed to a safe architecture independent from projects or type of applications. It consists of a safe state machine containing a decision tree. The state machine has only two states: init and run. The init state is active only at initialization, while the run state is active for the rest of the cycle. The run state contains a decision tree with three selections that can be executed, depending on the value of a variable thread coming from the basic software. The difference in the values comes from the timing of execution configured in OS. For example, each subnode can be executed each 0.1 ms, 1 ms or 10 ms, as shown in the figure. However, subnodes communicate with each other via an interface and, therefore, need protection from interruption. A faster task could be interrupted during execution of a slower task. What if the slower task uses the data intended for a faster one? During the execution of the slower task, the output data from the faster one is updated by the interruption, and this may overwrite the values being used for the current execution. These types of data errors can produce random results in the model. For slower tasks to execute safely, the inputs should be stored explicitly before being used in the calculation.



▲ HEV software development process with SCADE

COURTESY SUBARU.

COMBINING APPLICATION MODELS WITH A SAFE SCADE ARCHITECTURE MODEL

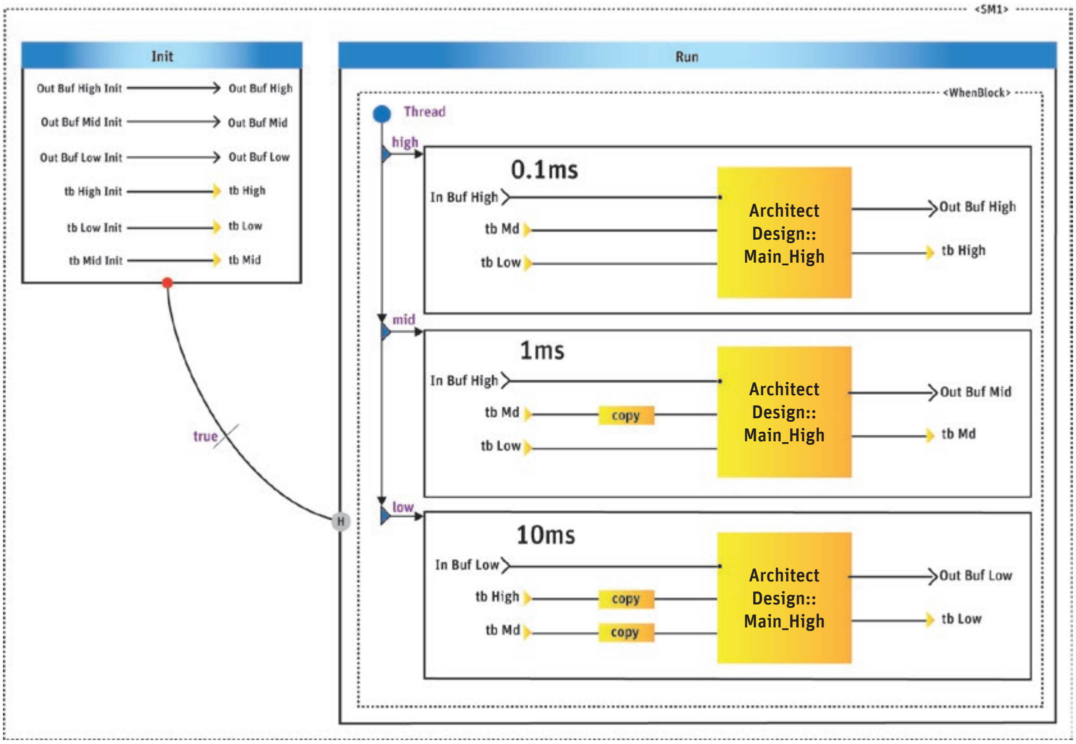
Once the root node describes concrete architecture, each subnode can be designed using a modular approach. Functional models for EBU applications are designed by a system engineering team with Simulink. These Simulink models are also imported into SCADE using the Simulink gateway. Before being imported, they are verified in the Simulink environment based on the software requirements. The test scenarios are described in Excel® sheets and converted into *.in format for the SCADE simulator. When both simulation results are identical, it means that the simulation is correct. Once all pieces of the functional Simulink models are converted into SCADE correctly, they are integrated into the safe SCADE architecture node.

REFINEMENT OF SCADE MODELS

To reduce verification time, the SCADE Suite Design Verifier is used to check if the SCADE model prior to refinement is identical to the one after refinement. In general, formal verification techniques are used to test properties like safety, but verification algorithms may face numerical difficulties if the nodes contain more arithmetic calculations than decision diagrams and state machines. Subaru engineers refine the SCADE models daily, and Design Verifier helps to validate the models.

SAFE AND RELIABLE AUTOMATIC CODE GENERATION FOR INTEGRATION

The final stage of the SCADE process is to generate C source codes from the verified SCADE models using the IEC 61508 certified SCADE Suite KCG code generator. The generated code usually meets safety objectives. Because the KCG tool has been qualified and certified for this purpose, Subaru verifies that the safe properties obtained from the safe SCADE architecture model are retained in the generated codes. Two SCADE Suite KCG features are essential to ensure that safe properties

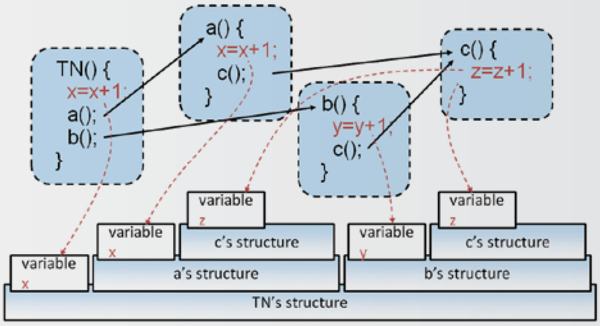


▲ A combination of safe state machine and decision diagram is the basis of the safe SCADE architecture model.

Comparing the Architecture Model and Generated Codes

SCADE assures data access with a unique path when a node contains variables that memorize the value. This is not applied to a "function" in SCADE because the function does not contain any variables that memorize the values. If node TN() calls node a() and node b(), and a() and b() commonly call node c(), and all of them contain variables that memorize values, the data are structured by the node in the generated codes. Although node c() is commonly called from two nodes a() and b(), its corresponding data are structured separately. Because KCG guarantees it, another data structure is created safely when the root node TN() is called by interruption while it is being executed.

The generated code is also correct thanks to SCADE Suite's KCG IEC 61508 certified code generator. C code is a part of the generated codes from a safe SCADE Suite architecture model using KCG. To compare the safe SCADE architecture model with the codes, all copies of inputs are coded by the kcg_copy macro function using the default memcpy function. KCG makes it possible to replace the macro with a user-defined macro after code generation without any impact on the other codes. Subaru replaced the kcg_copy macro with the one inhibiting interruption during the copy.



▲ Safe access to variables that memorize the values exists as a unique path.

are met in the codes: The variables are protected from overwriting when interruption occurs, and input data is appropriately copied before being used as described in the safe SCADE architecture model.

DEVELOPING CONTROL SOFTWARE FOR HEVS USING SCADE

Using SCADE software, Subaru was able to describe consistent readable models ranging from safe architecture design to detailed designs. Thanks to SCADE Suite's KCG IEC 61508 certified code generator, the verification time at code level was significantly reduced, as most of the verification was completed upfront at the SCADE model level. A small group of Subaru engineers completed a large and very complex application while significantly reducing software development and testing time. Subaru engineers continue to use SCADE Suite as an important part of their HEV development process.

> **Subaru engineers completed a large and very complex application while significantly reducing software development and testing time.**

COURTESY SUBARU.

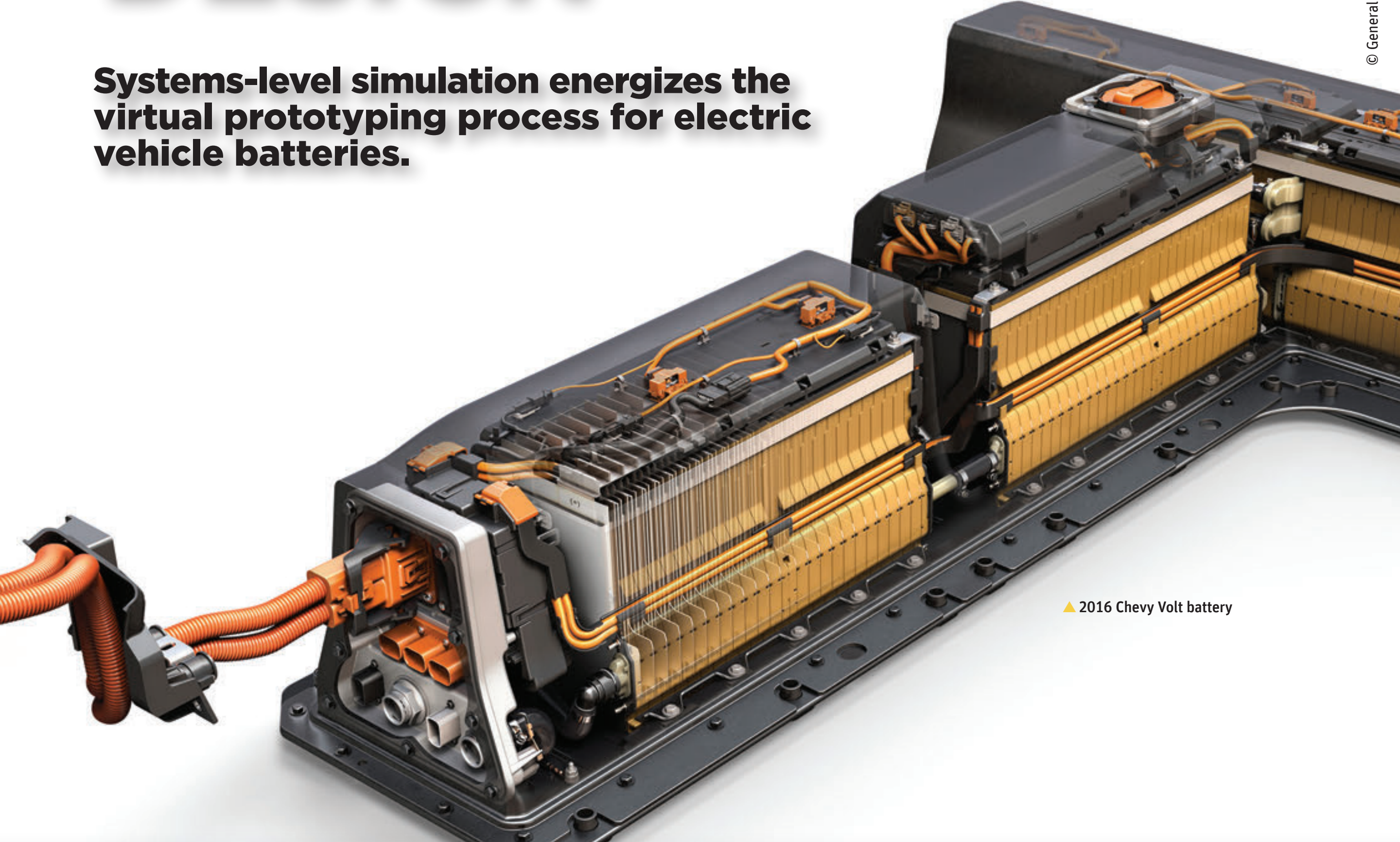
```
if (Ctxt_App_Main.init) {
  Ctxt_App_Main.init = kcg_false;
  SM1_state_act = SSM_st_Init_SM1;
 }
 else {
  SM1_state_act = Ctxt_App_Main.SM1_
  state_nxt;
   }
   switch (SM1_state_act) {
    case SSM_st_Run_SM1 :
     Ctxt_App_Main.SM1_state_nxt =
     SSM_st_Run_SM1;
        switch (thread) {
         case low :
          copy_tbMidType(&Ctxt_App_
          Main.tbMid, &_L8_SM1_Run_
          WhenBlock_low);
          copy_tbHighType(&Ctxt_
          App_Main.tbHigh,
          &_L7_SM1_Run_WhenBlock_low);

Main_Low(&InBufLow,&_L7_SM1_Run_
WhenBlock_low,&_L8_SM1_Run_WhenBlock_
low,
     &Ctxt_App_Main._2_Context_2);

kcg_copy_tbLowType(&Ctxt_App_Main.
tbLow,&Ctxt_App_Main._2_Context_2.
tbLow);
```

```
     kcg_copy_
OutBufLowType(&OutBufLow,&Ctxt_App_
Main._2_Context_2.OutBufLow);
     break;
    case mid :
     copy_tbHighType(&Ctxt_App_Main.
     tbHigh, &_L8_SM1_Run_WhenBlock_
     mid);
     Main_Mid(&InBufMid,&_L8_SM1_Run_
     WhenBlock_mid,&Ctxt_App_Main.
     tbLow,
     &Ctxt_App_Main._1_Context_2);

kcg_copy_tbMidType(&Ctxt_App_Main.
tbMid,&Ctxt_App_Main._1_Context_2.
tbMid);

kcg_copy_
OutBufMidType(&OutBufMid,&Ctxt_App_
Main._1_Context_2.OutBufMid);
     break;
    case high :
     Main_High(&InBufHigh,&Ctxt_App_
     Main.tbMid,&Ctxt_App_Main.tbLow,
      &Ctxt_App_Main.Context_2);

kcg_copy_tbHighType(&Ctxt_App_Main.
tbHigh,&Ctxt_App_Main.Context_2.
tbHigh);
```

```
kcg_copy_OutBufHighType(&OutBufHigh,&
Ctxt_App_Main.Context_2.OutBufHigh);
     break;
   }
   break;
  case SSM_st_Init_SM1 :
   Ctxt_App_Main.SM1_state_nxt =
   SSM_st_Run_SM1;
kcg_copy_tbLowType(&Ctxt_App_Main.
tbLow, (tbLowType *) &tbLowInit);
kcg_copy_tbMidType(&Ctxt_App_Main.
tbMid, (tbMidType *) &tbMidInit);
kcg_copy_tbHighType(&Ctxt_App_Main.
tbHigh, (tbHighType *) &tbHighInit);
kcg_copy_OutBufLowType(&OutBufLow,
(OutBufLowType *) &OutBufLowInit);
kcg_copy_OutBufMidType(&OutBufMid,
(OutBufMidType *) &OutBufMidInit);
kcg_copy_OutBufHighType(&OutBufHigh,
(OutBufHighType *) &OutBufHighInit);
   break;
 }
```

AUTOMATING BATTERY PACK DESIGN

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



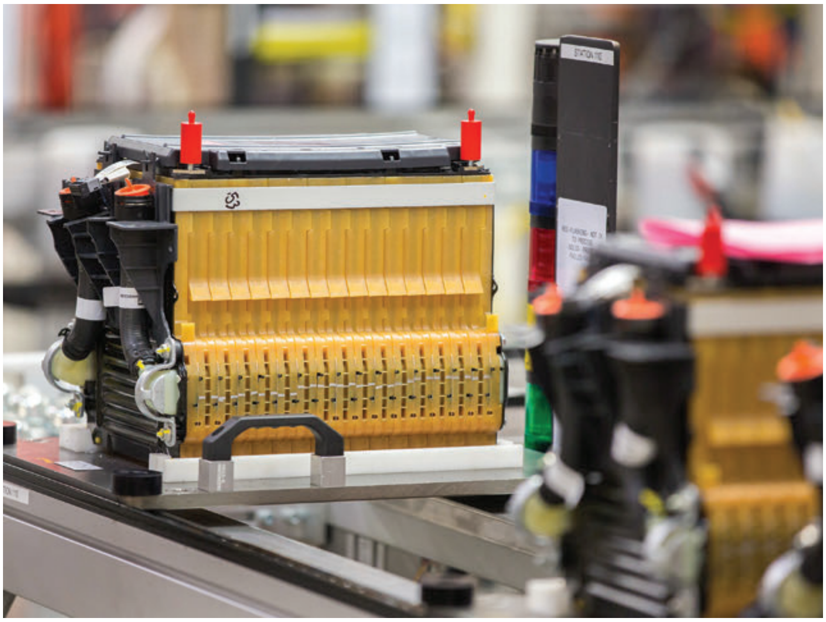
© General Motors.

2016 Chevy Volt battery

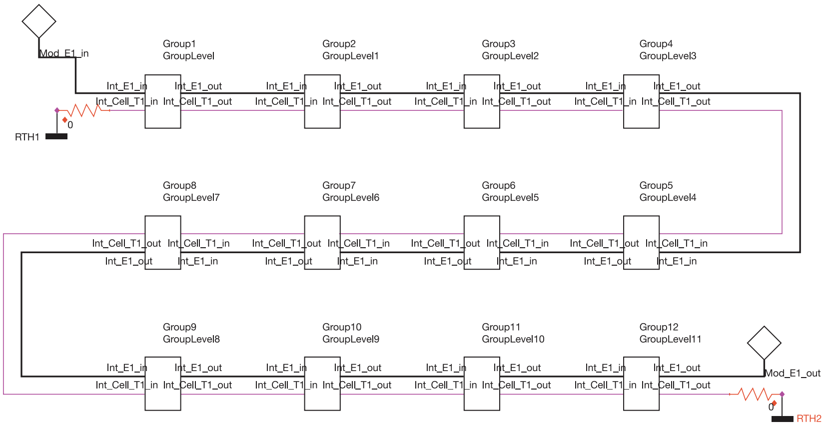
By Erik Yen, Senior Researcher, and **Taeyoung Han**, Technical Fellow, Vehicle Systems Research Laboratory,
General Motors Research and Development Center, Warren, U.S.A.
Sameer Kher, Senior Manager, Software Development, ANSYS

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

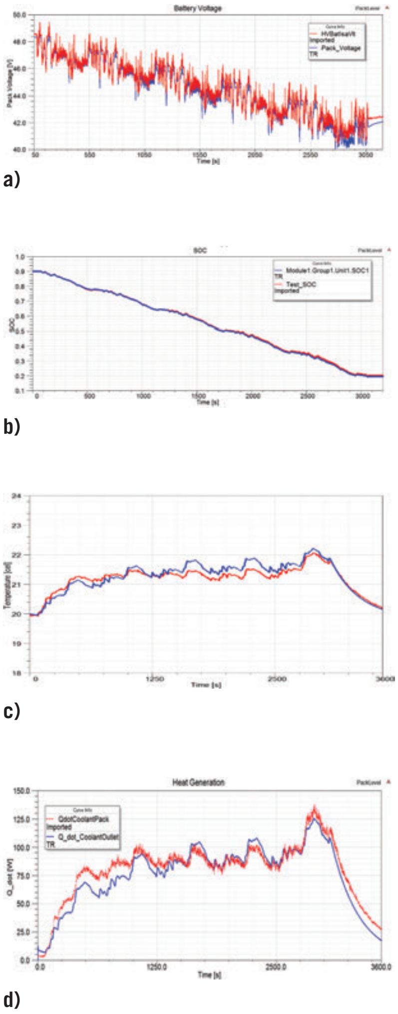
For electric vehicle makers, designing an efficient and robust cooling system for the battery pack is a key engineering task.


© 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.

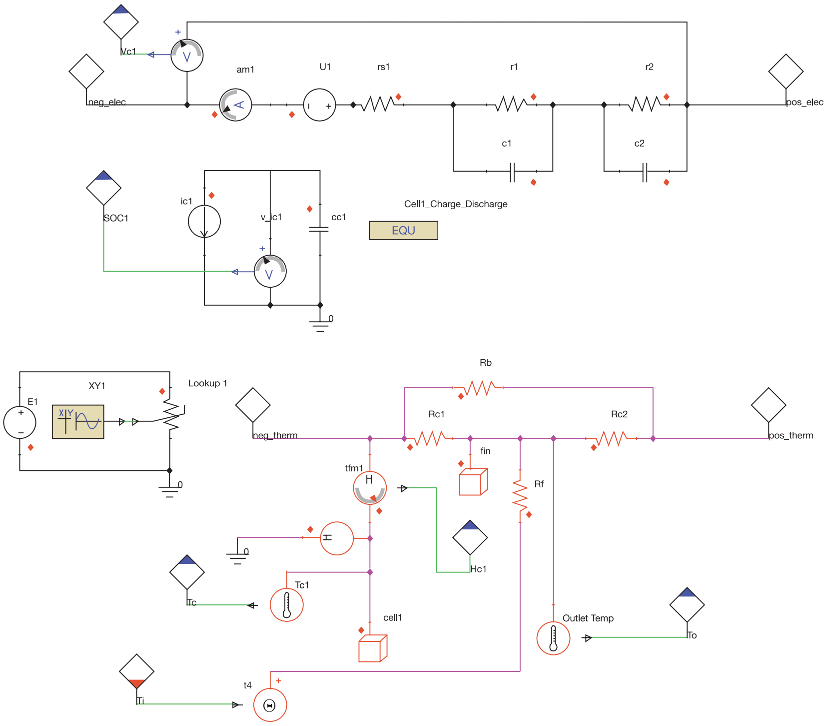
To analyze the coupled electrochemistry and heat transfer of a pack, it is desirable to have predictions based on fine spatial resolution of the entire system of battery cells. However, such information may be available only through resource-intensive, time-consuming full-field simulation, which is not 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



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

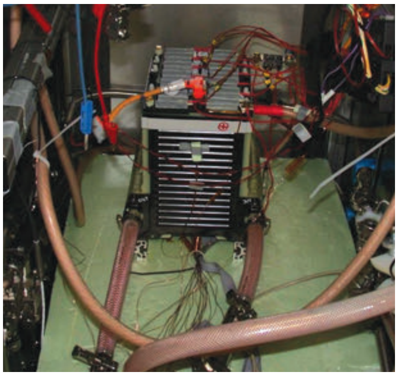
and thermal behavior, the research team created several unit-model variations based on theoretical and empirical formulations for both circuit and heat-transfer modeling. Once completed, these units are easily stored in a Simplorer user library for later use by a production pack designer.

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.

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

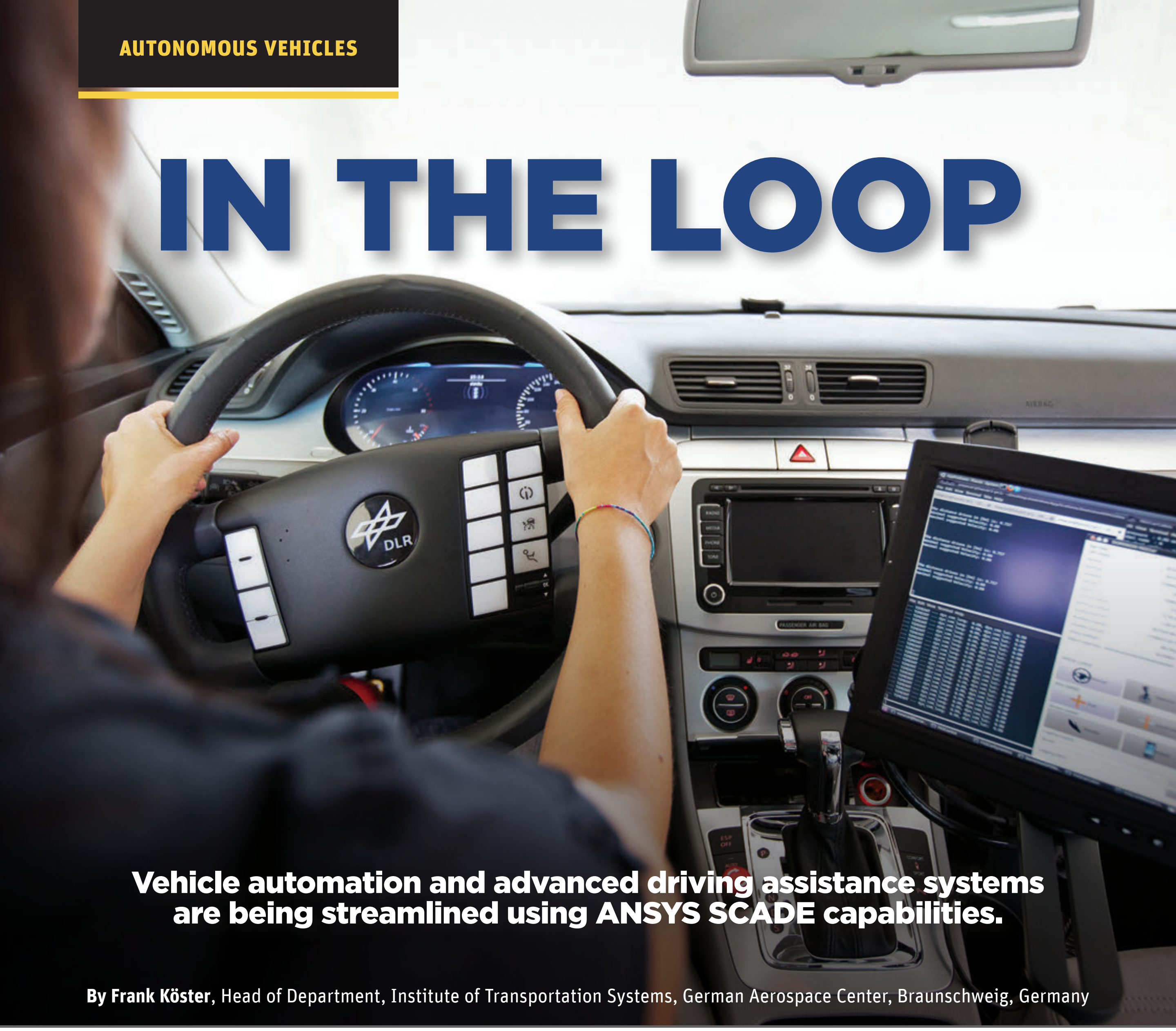
VALIDATION AND FUTURE WORK

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.


© General Motors.

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.

ROLE OF THE HUMAN–MACHINE INTERFACE

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

HMI on heads-up display for highly automated driving

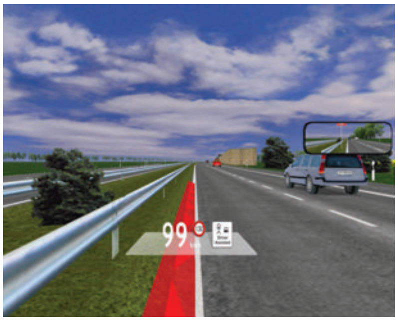
By simulating behavior using the model, it is possible to identify and correct defects as well as gain critical insights early in the design process to rapidly improve system performance.

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.

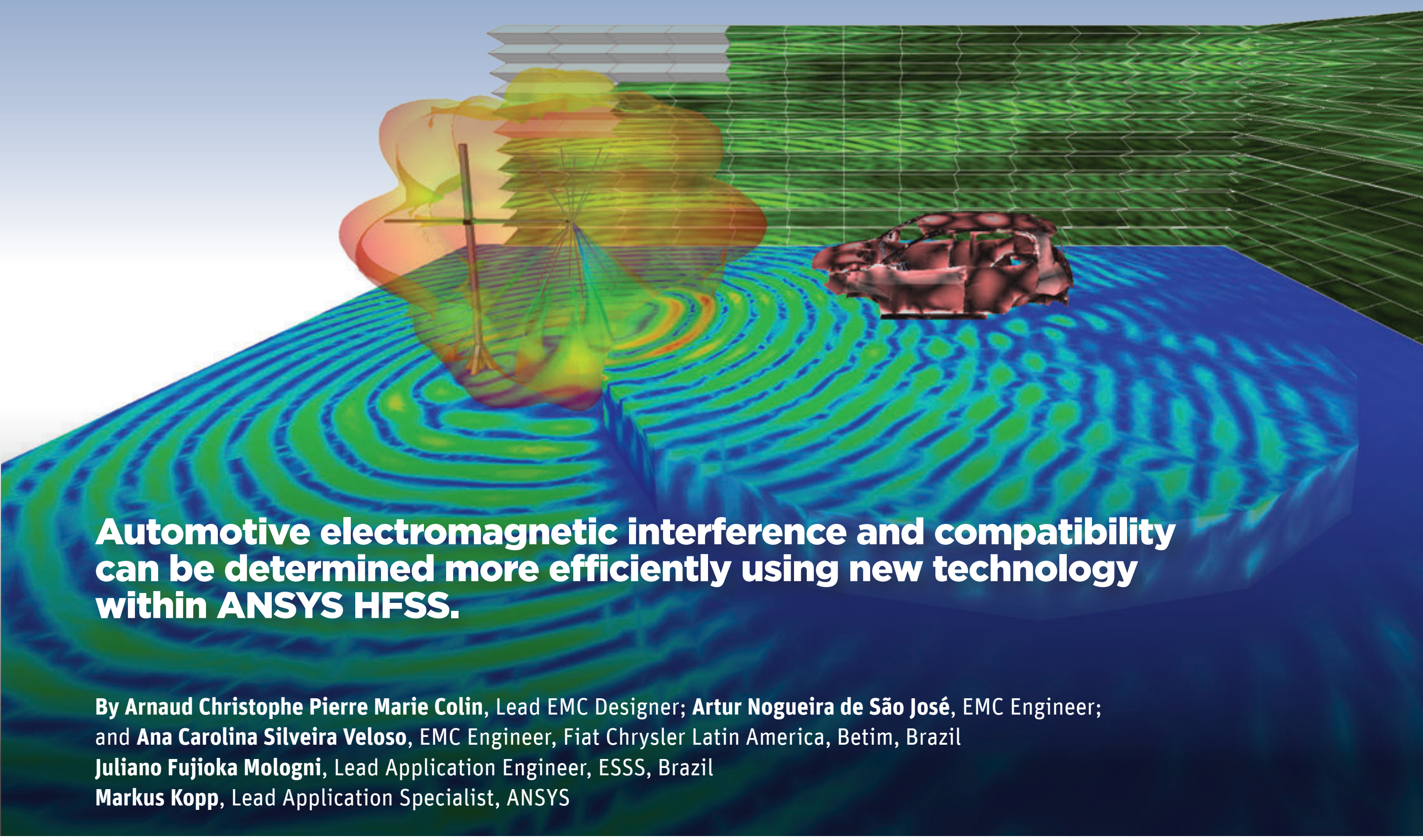
Engineers simulate the behavior of the model and receive immediate feedback on its performance. Test cases are run in the virtual PC environment rather than in the more-complicated and expensive target environment. For example, for each new iteration of the code, engineers must check hundreds of different 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.

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.

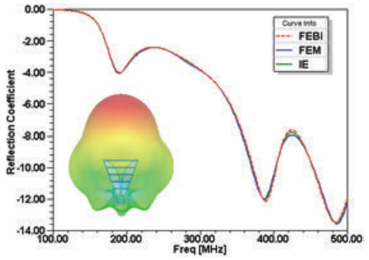
Many standards, directives and regulations are designed with vehicle safety in mind. This ensures that all onboard systems continue to function properly during exposure to EMI and then return to normal, either automatically or by a manual reset operation after exposure. A major concern for Fiat Chrysler's 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.

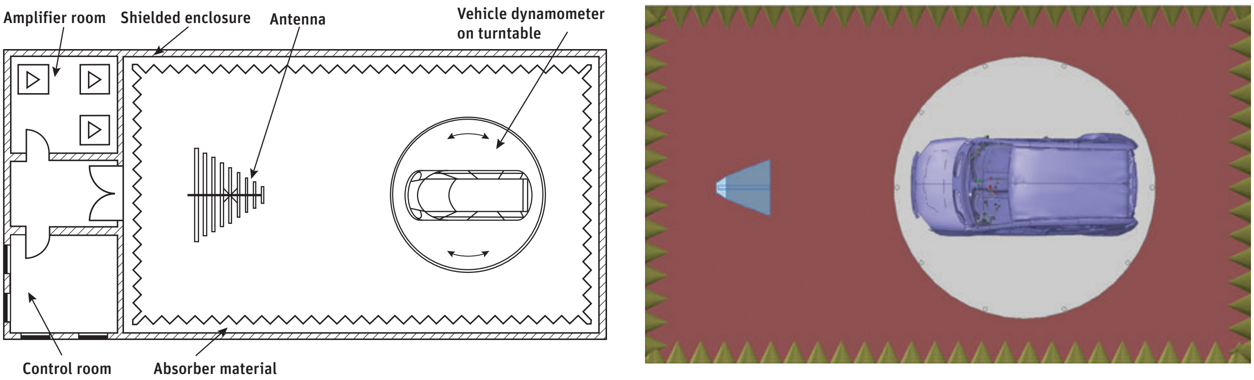


A comparison of far-field behavior shows that the FE-BI method is accurate compared to traditional methods.

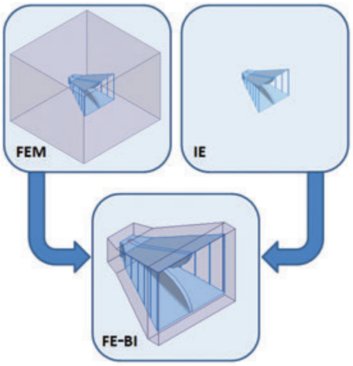
ELECTROMAGNETIC SIMULATION OF ANTENNAS INSTALLED INSIDE VEHICLES: AN AUTOMOTIVE EMC APPROACH
ansys.com/92emi

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.

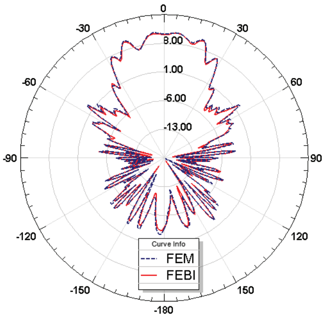
To demonstrate the capability of the FE-BI methodology, the Fiat Chrysler team worked with ESSS, the ANSYS channel partner in South America, to conduct



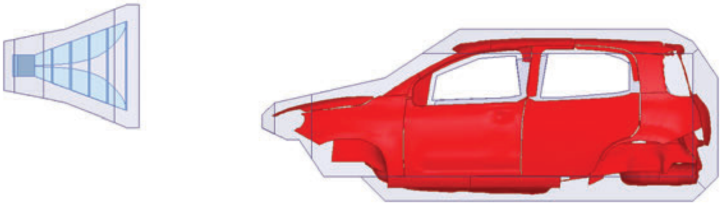
ISO 11451-2 test apparatus (left). Virtual test chamber used for ANSYS HFSS simulation (right)



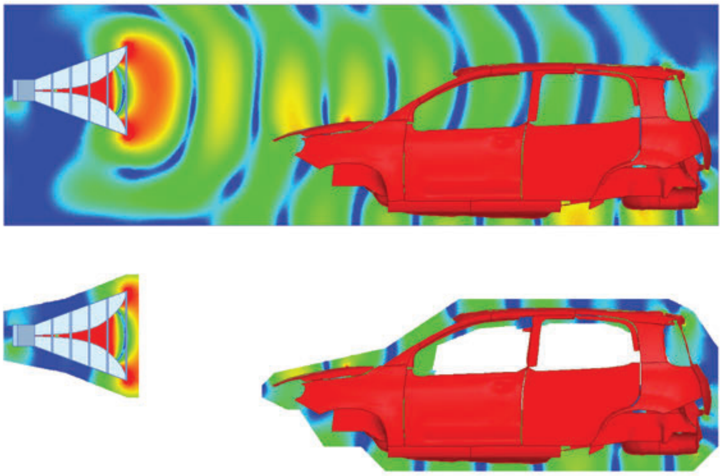
▲ Comparison of FEM, IE and FE-BI models. The reduced size of the solution region when using FE-BI (as compared to traditional FEM or other numerical 3-D field solvers) leads to a faster simulation.



▲ 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.

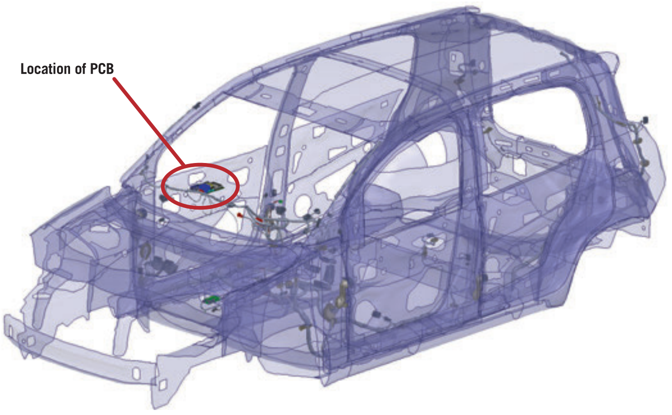
Fiat Chrysler engineers did not model the absorber elements in this simulation because the IE boundary in FE-BI is equivalent to a free-space simulation, which is the same as absorbing material used in a physical measurement. The total computation time of just 28 minutes represents more than a 10-fold 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.

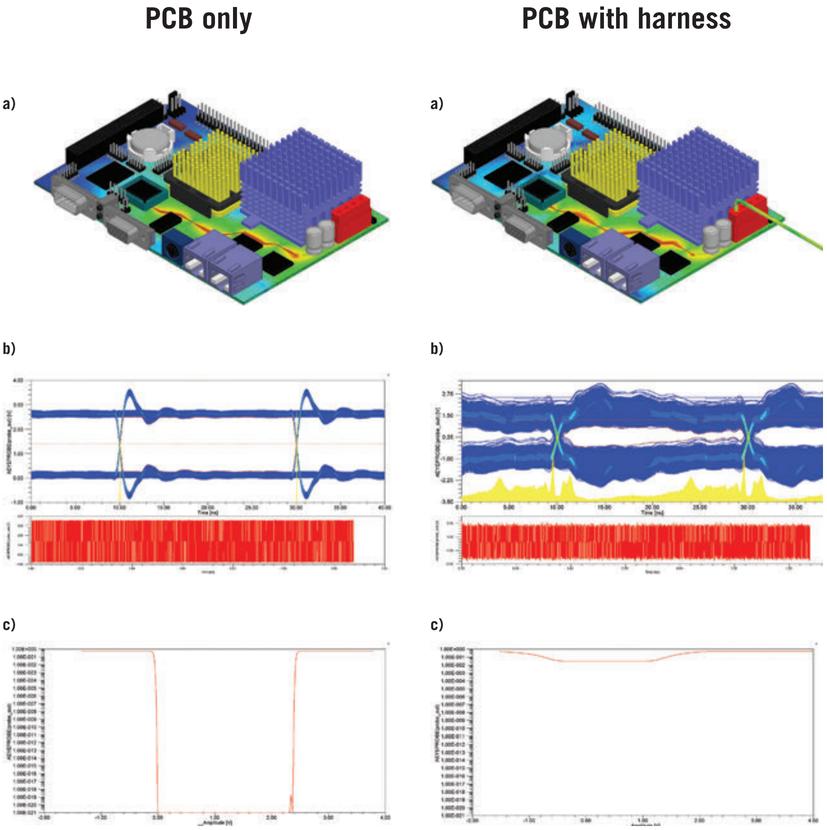
The FE-BI approach also can be used to test the immunity of embedded control unit modules. To demonstrate this capability, the engineering team introduced a printed circuit board (PCB) connected to the engine wiring harness into the simulation. The transmitted signal travels from a sensor, located at 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.

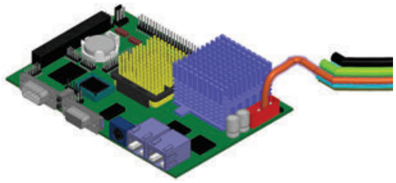
first included the PCB and wiring harness along with the chassis and source antenna. For the second simulation, the team removed the wiring harness, and the random CAN J1939 signal was applied directly into the PCB connector instead of at the sensor location at the bottom of the engine.

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.

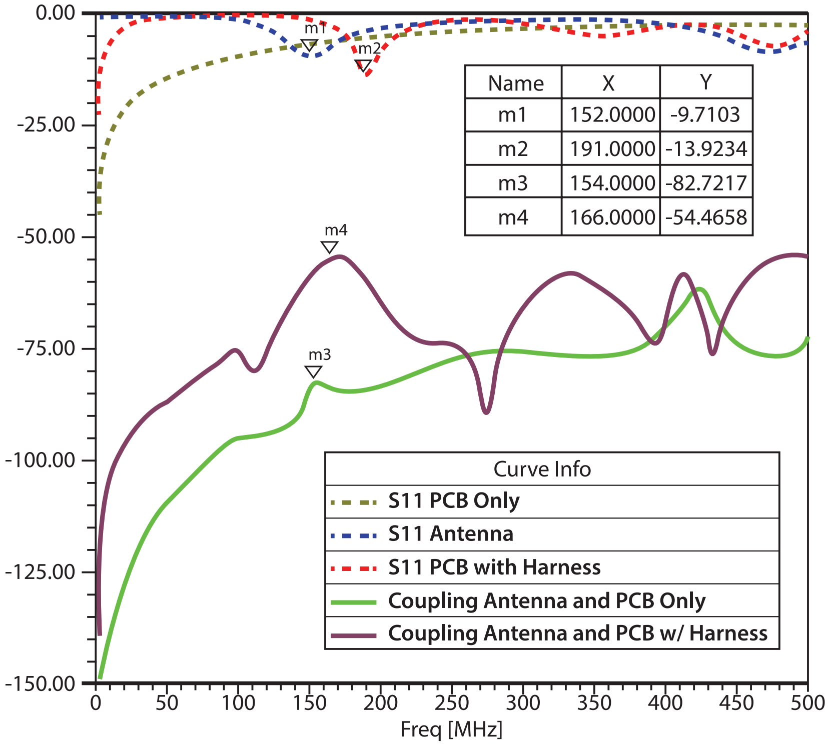
The ANSYS HFSS FE-BI capability is clearly advantageous as a numerical

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

HOT FLASH

Using a reduced-order method, engineers at Fairchild Semiconductor have been able to decrease development time for electronic components for electric and hybrid electric vehicles. By employing ANSYS Icepak for thermal management and ANSYS Simplorer for multi-domain systems design, the engineers performed dynamic thermal analysis under realistic power loss conditions approximately 2,000 times faster than a full 3-D thermal analysis.

By Roy Davis, Senior Manager, Ann Arbor, USA
Klaus Neumaier, R&D Engineer, Fairchild Semiconductor, Munich, Germany

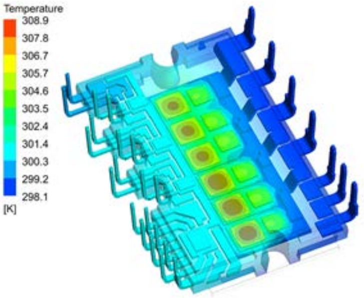
In electric vehicles (EVs) and hybrid electric vehicles (HEVs), many of the systems we take for granted in internal combustion automobiles must be implemented differently. In addition to the main powertrain motor–generator sets, some accessory systems are also electrified. For example, because the engines in these vehicles do not run continuously, electrically driven air-conditioning (AC) compressors and transmission oil pumps may be used instead of usual engine accessory belt-driven systems. All of these electric drives require fast, efficient and reliable power electronic converters.

Fairchild Semiconductor, a pioneer in the semiconductor industry, produces a wide range of power components suitable for these power electronic converters, as well as a complete portfolio of low- to high-power solutions for the mobile, industrial, cloud, automotive, lighting and computing industries. A typical Fairchild inverter consists of an array of six insulated gate bipolar transistors (IGBTs), each of which is paired with an anti-parallel diode, to convert DC voltage provided by the vehicle's battery pack into three-phase AC voltage required to drive the AC motors. Two IGBTs and two diodes form each "half-bridge" powering one phase of the motor. A single IGBT and diode pair form one bi-directional switch and, in combination with the other switches in the inverter, operate according to a pulse-width modulation scheme to generate the AC waveforms necessary to control the motor. Switching the IGBTs on and off at 5 to 20 kHz results in pulsed DC voltages that average out to yield a sine wave on each phase, with the three sine waves being 120 degrees out of phase with each other. The three-phase sinusoidal drive produces a rotating electric field in the electric motor, precisely controlled in a closed-loop feedback system to generate the necessary torque to drive the load.

EV or HEV power train inverters are rated in the 50 kW to 100 kW range, and AC compressors are in the 6 kW range, so the power dissipated in these devices is high, and thermal considerations are critical to their design. The traditional approach is to simulate the circuit to determine power dissipation under specified operating conditions. These power losses are then used as inputs to a 3-D thermal simulation that predicts junction temperatures on the IGBTs and diodes. This approach usually requires about 8 hours to simulate tenths of seconds of operating time needed to thermally characterize a design iteration under a given set of operating conditions. In addition, a considerable amount of engineering time is tied up running separate electrical and thermal simulations and manually passing data back and forth. Fairchild engineers have improved the inverter design process by using ANSYS Icepak to develop a systems-level linear time-invariant (LTI) reduced-order model (ROM) that runs in the ANSYS Simplorer system simulation environment to predict thermal performance in minutes rather than hours or days.

VALIDATING THE ROM METHOD

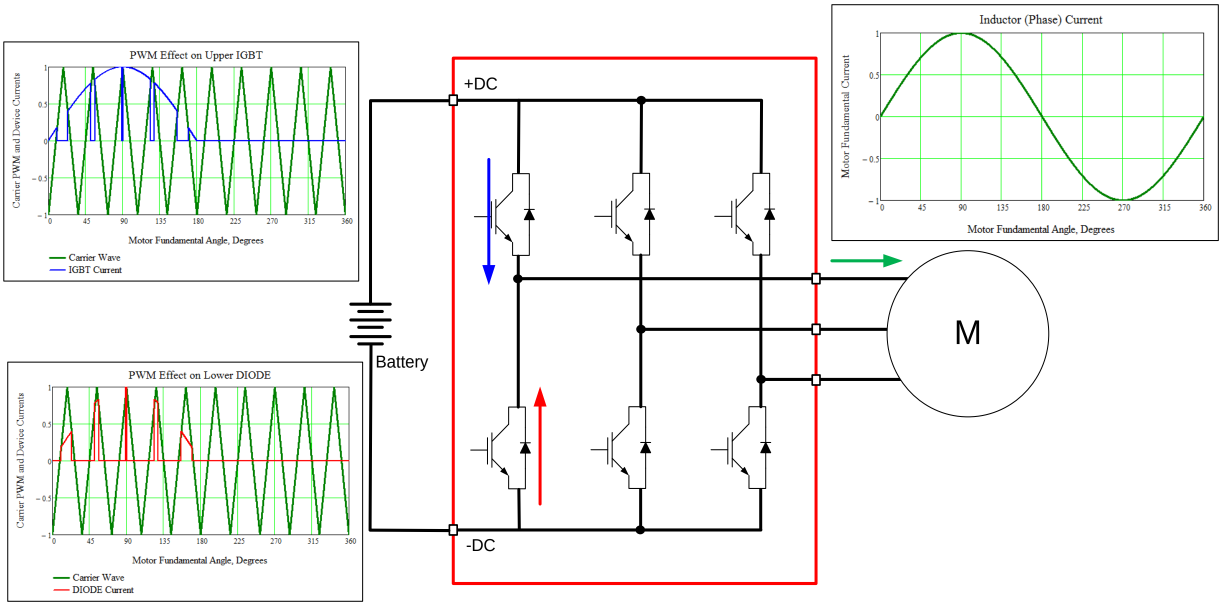
Fairchild engineers created an Icepak model of the three-phase inverter, including its package and enclosure. They ran a few simple simulations to validate the model's accuracy, and performed a steady-state simulation. Then they performed a series of step response analyses in Icepak for a set of input and output quantities to build a compact model. For the inverter, the power dissipation of the six IGBTs and six diodes, as well as the heat sink temperature, is typically used as input, while the junction temperatures provide the set of output quantities. Based on the fully described thermal system, the team then used Icepak to generate a linear time-invariant reduced-order model (LTI ROM) that can be used in Simplorer to simulate specified electrical and thermal conditions in a fraction of the time required for a full 3-D thermal simulation.



▲ 3-D thermal analysis accurately determines thermal status at a high computational cost.

Fairchild engineers evaluated their initial ROM against a full 3-D thermal

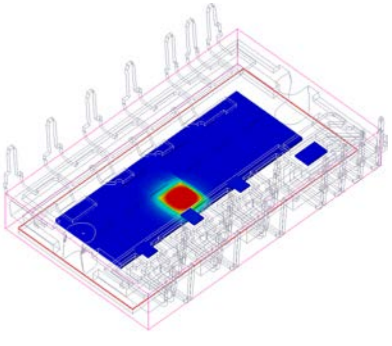
In electric vehicles (EVs) and hybrid electric vehicles (HEVs), many of the systems we take for granted in internal combustion automobiles must be implemented differently.



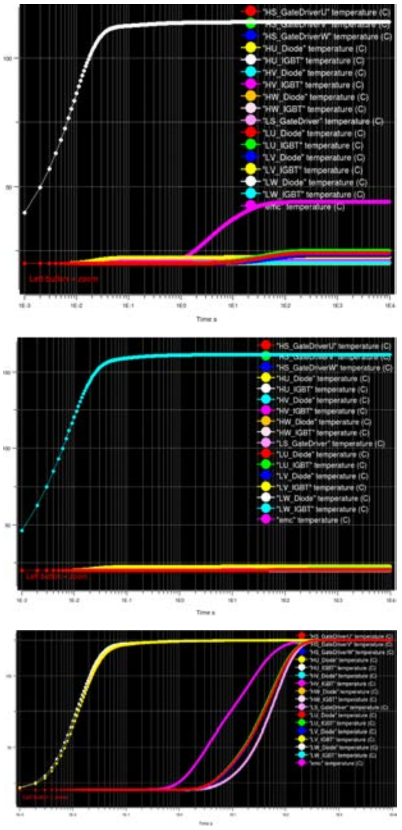
▲ The inverter converts direct current to three-phase alternating current by generating pulses that combine to yield a sine wave.

	Upper U						Lower						
	U		V		W		U		V		W		
Trial No.	D	I	D	I	D	I	D	I	D	I	D	I	Tref
	[W]												[C]
1	300	0	0	0	0	0	0	0	0	0	0	0	0
2	0	300	0	0	0	0	0	0	0	0	0	0	0
3	0	0	300	0	0	0	0	0	0	0	0	0	0
4	0	0	0	300	0	0	0	0	0	0	0	0	0
5	0	0	0	0	300	0	0	0	0	0	0	0	0
6	0	0	0	0	0	300	0	0	0	0	0	0	0
7	0	0	0	0	0	0	300	0	0	0	0	0	0
8	0	0	0	0	0	0	0	300	0	0	0	0	0
9	0	0	0	0	0	0	0	0	300	0	0	0	0
10	0	0	0	0	0	0	0	0	0	300	0	0	0
11	0	0	0	0	0	0	0	0	0	0	300	0	0
12	0	0	0	0	0	0	0	0	0	0	0	300	0
13	20	20	20	20	20	20	20	20	20	20	20	20	125

6 IGBT, 6 Diode, Tref



13 Analyses≈26h



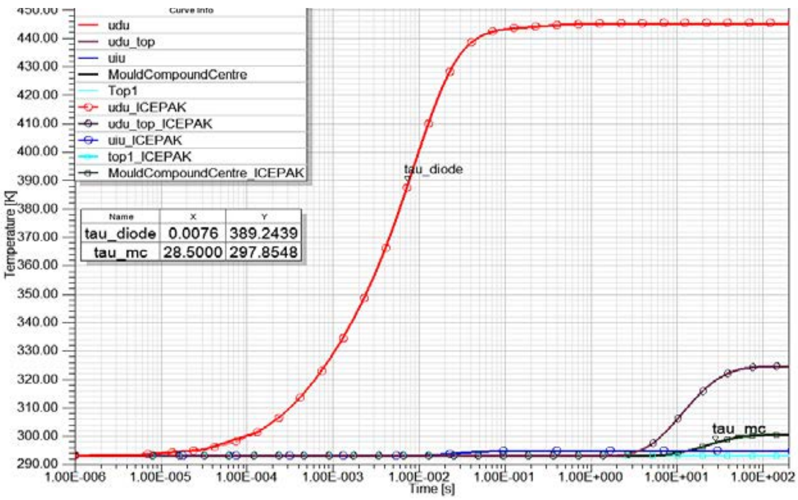
▲ A reduced-order model created by varying load on components and ambient temperature in 13 separate 3-D thermal analysis runs

analysis to test its accuracy. The temperatures of all components as predicted under a step-load test by the ROM were a near perfect match (below 1 percent error) with the predictions made by the full 3-D thermal analysis. The solver time for this full 3-D simulation was about two hours, while the ROM took about one minute to run. Then, a variety of pulse load tests were employed to compare the results of the ROM against 3-D thermal analyses. Again, the ROM results matched the 3-D analysis very closely (below 2 percent error level). In this case, the 3-D model took eight hours to solve for 150 milliseconds of operating time, while the ROM took four hours to solve a much longer 150-second time period. In this case, the ROM provided a 2,000-to-1 speedup, taking half the time to produce 1,000 times more information. Fairchild engineers did not feel the need to validate the results with physical testing because they have validated 3-D thermal analysis against physical testing many times and found a very close correlation every time.

FIRST USE ON ACTUAL PRODUCT

The Fairchild team used the ROM method to develop a three-phase inverter for an air-conditioning compressor for an EV. Engineers used Icepak to create a ROM of the components, package and enclosure. They imported the model into Simplorer and computed the demands on the inverter with the compressor motor spinning at 5,000 rpm when the air-conditioner is turned on. The ROM computed the junction temperatures of each device. Once the ROM was developed, the Simplorer environment enabled integration of device models, load models, package model and control systems elements

ROM (Ideal HS) Versus 3-D Step Response Test



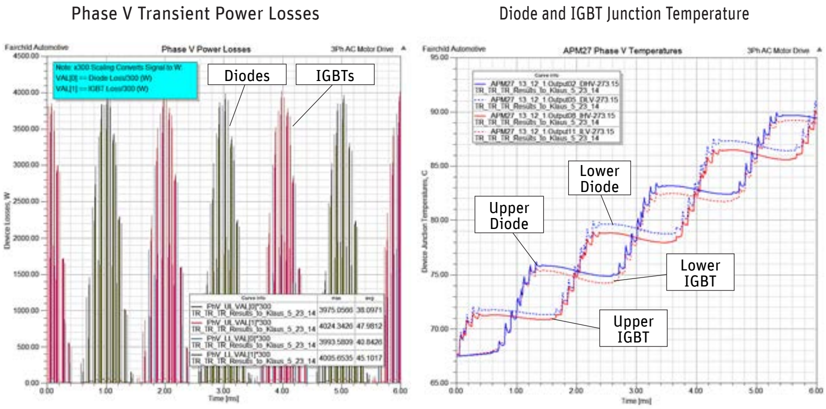
Name	X	Y
tau_diode	0.0076	389.2439
tau_mc	28.5000	297.8548

▲ A reduced-order model provides a near perfect match to 3-D thermal analysis results.

SYSTEM-LEVEL SIMULATION USING ANSYS ICEPAK AND ANSYS SIMPLORER — WEBINAR
ansys.com/system101

Fairchild engineers have improved the inverter design process by using a reduced-order model to predict thermal performance in a minute or two.

The ROM took half the time compared with the previous method to produce 1,000 times more information.



Phase V Transient Power Losses

Diode and IGBT Junction Temperature

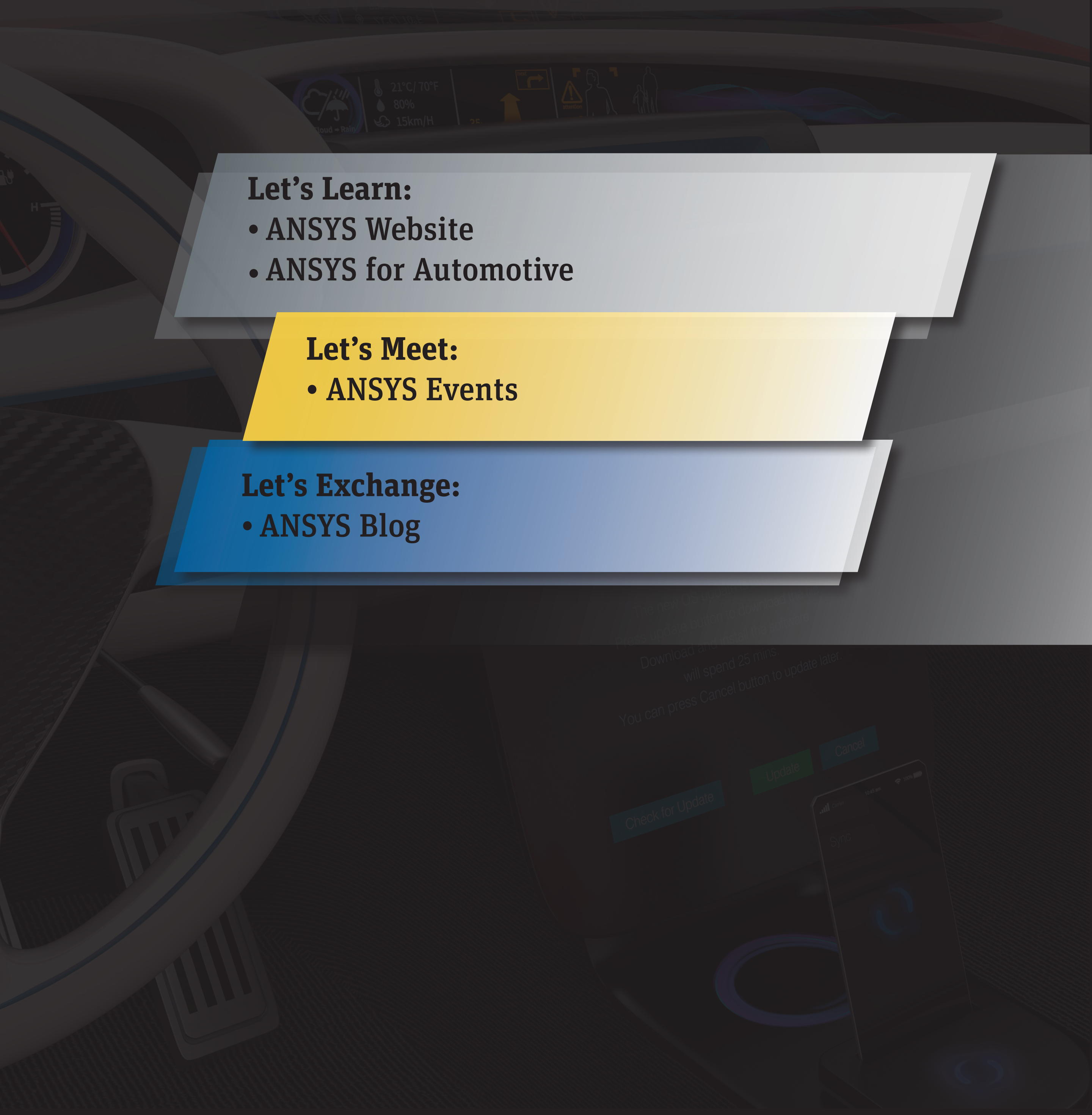
▲ **ANSYS Simplorer determines power losses and uses ROM to compute junction temperature.**

to generate systems-level performance projections and design optimization. The engineer can also change circuit parameters to run under different operating conditions.

In this project, the ROM enabled Fairchild engineers to react to changes in design specifications and evaluate design alternatives in much less time than was possible in the past. For example, originally the customer said the heat sink would run at 85 C, but physical tests showed that it would actually run at 105 C. Engineers simply changed one parameter in Simplorer, and in a few minutes the ROM produced updated results. Motor operating conditions (such as switching frequency) also changed several times during the design process; Fairchild engineers were able to quickly evaluate the impact of each change.

Fairchild engineers expect substantial time savings in developing robust, reliable inverters. Key engineering team members involved in the project are already experiencing significant workload reductions. They fully expect that completely integrating ROM-based analysis methods with ANSYS tools into the development process will reduce engineering expenses, encourage innovation, and enable the company to take on more product development projects.

Integrating ROM-based analysis methods with ANSYS tools into the development process will reduce engineering expenses, encourage innovation, and enable the company to take on more product development projects.



Let’s Learn:

- ANSYS Website
- ANSYS for Automotive

Let’s Meet:

- ANSYS Events

Let’s Exchange:

- ANSYS Blog

ansys.com/Automotive