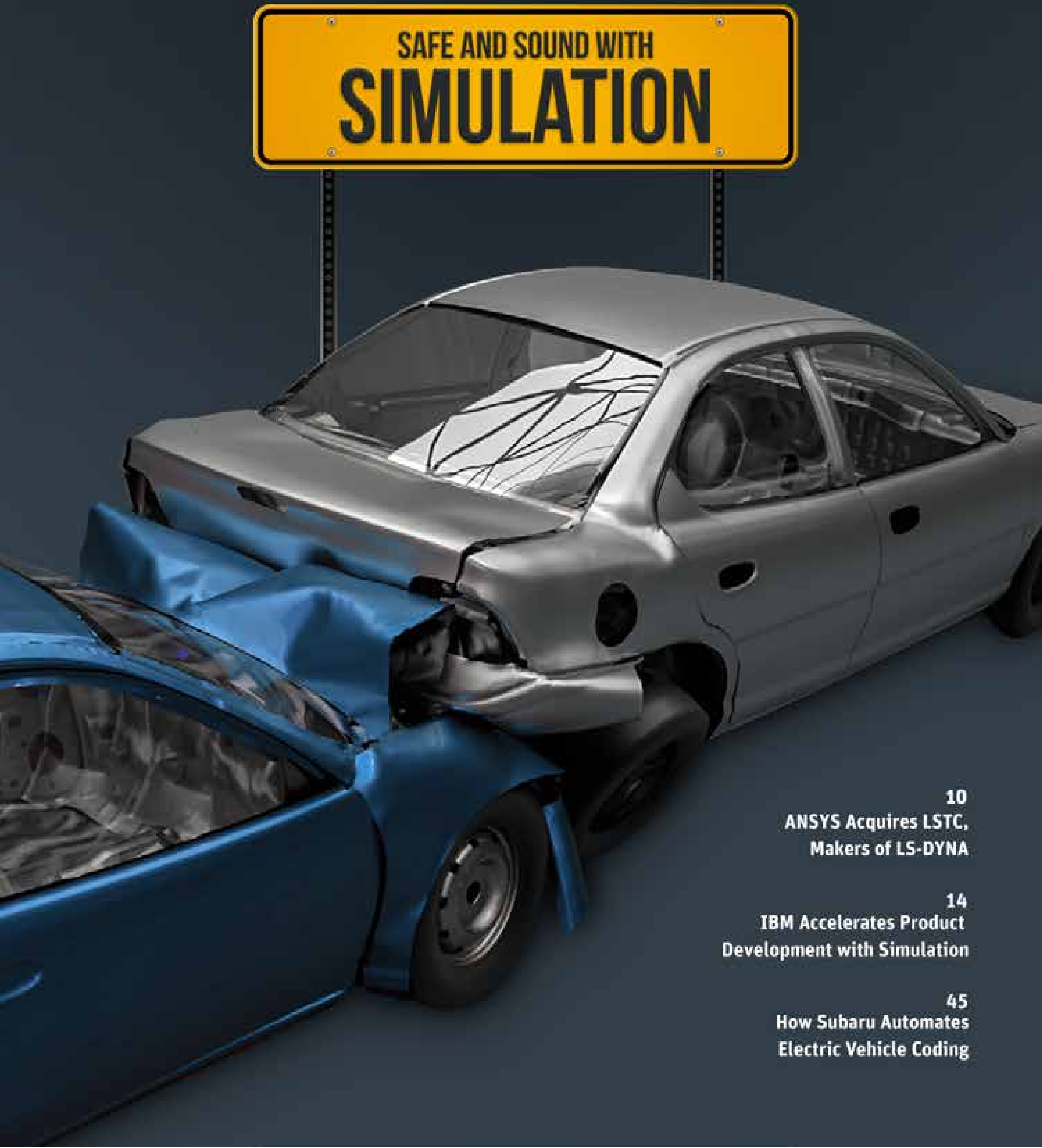
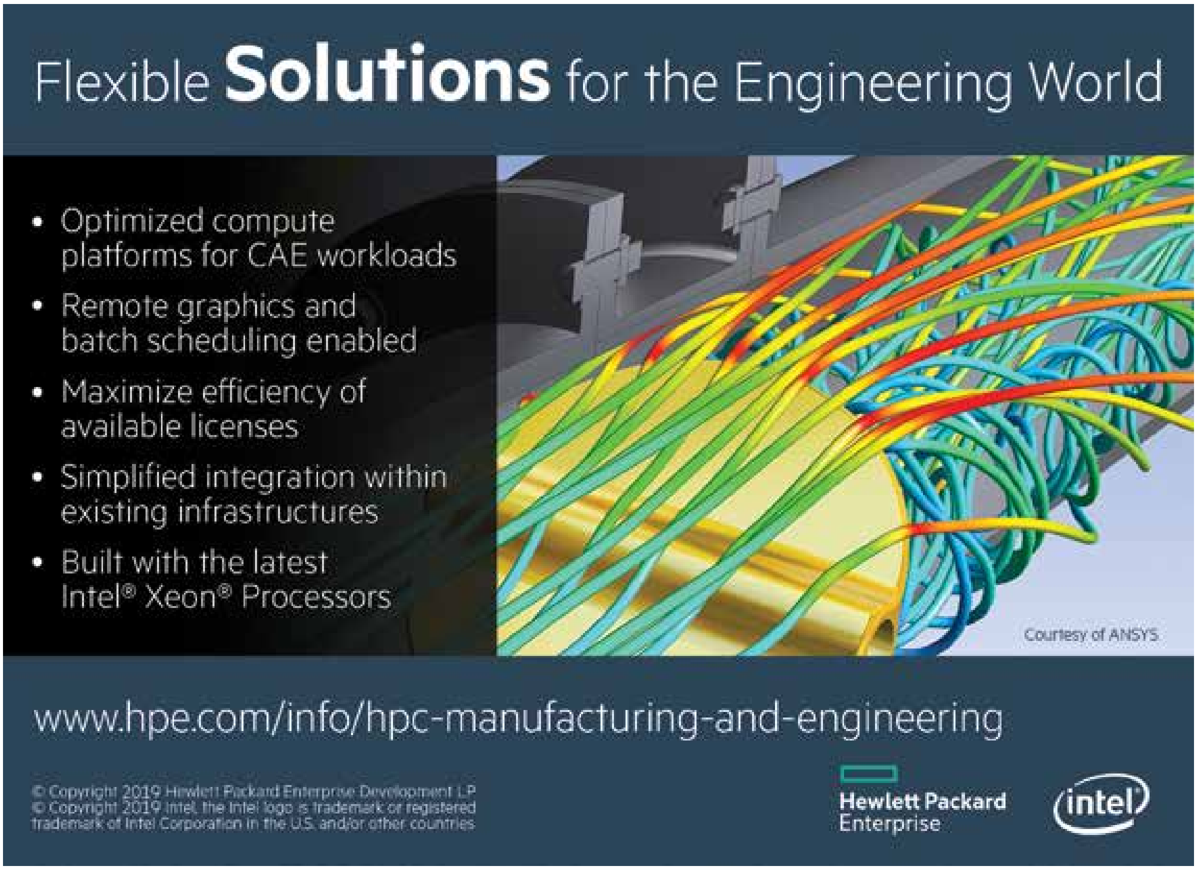
ANSYS

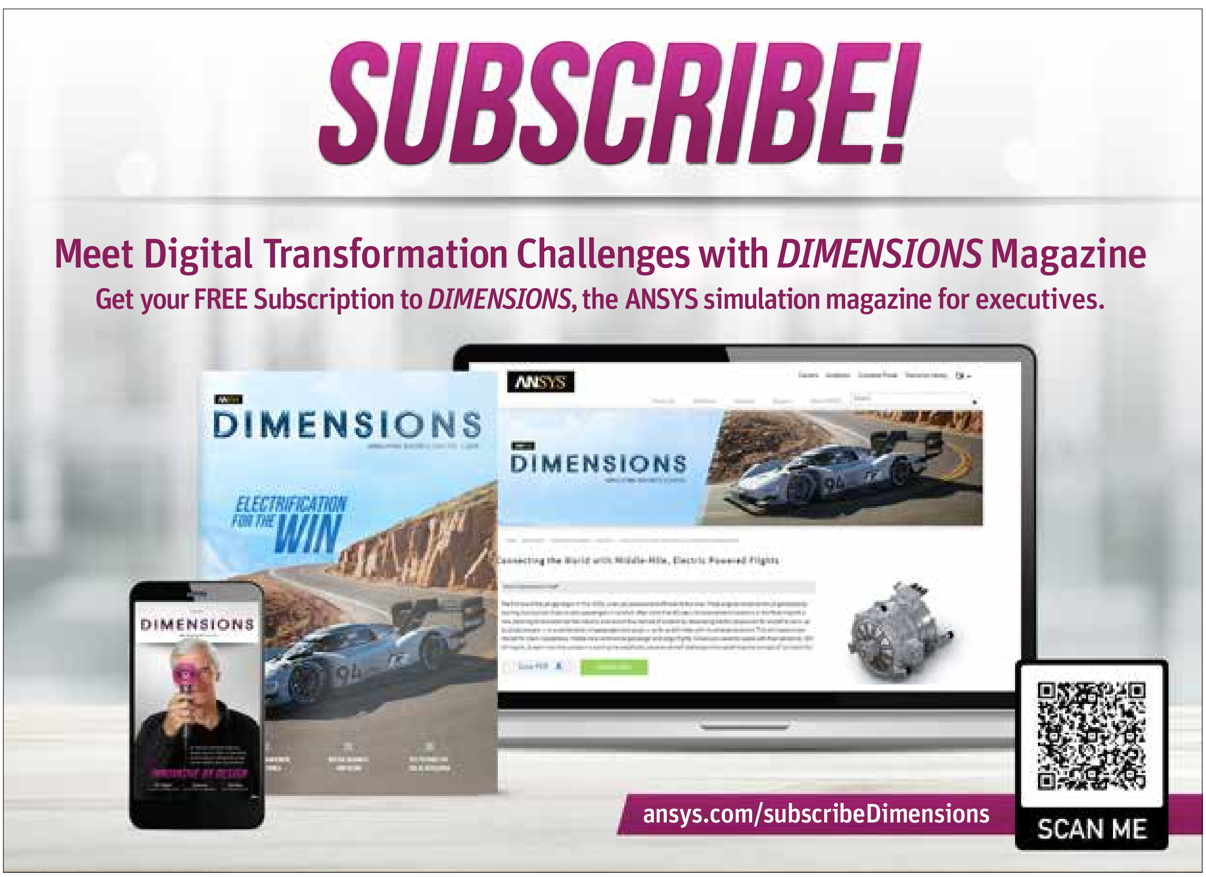
Excellence in Engineering Simulation

ADVANTAGE

ISSUE 3 | 2019




Flexible Solutions for the Engineering World
• Optimized compute platforms for CAE workloads
• Remote graphics and batch scheduling enabled
• Maximize efficiency of available licenses
• Simplified integration within existing infrastructures
• Built with the latest Intel® Xeon® Processors
Courtesy of ANSYS
www.hpe.com/info/hpc-manufacturing-and-engineering
© Copyright 2019 Hewlett Packard Enterprise Development LP
© Copyright 2019 Intel, the Intel logo is trademark or registered trademark of Intel Corporation in the U.S. and/or other countries
Hewlett Packard Enterprise
intel


SUBSCRIBE!
Meet Digital Transformation Challenges with DIMENSIONS Magazine
Get your FREE Subscription to DIMENSIONS, the ANSYS simulation magazine for executives.
DIMENSIONS
ELECTRIFICATION FOR THE WIN
ANSYS
DIMENSIONS
Connecting the World with Middle-Mile, Electric Powered Flights
DIMENSIONS
ansys.com/subscribeDimensions
SCAN ME

Making the Best-in-Class Even Better

For decades, ANSYS Mechanical has been the industry standard for structural analysis — the tool of choice for solving complex problems including fracture and fatigue. Over the years, the software has steadily evolved to meet our customers' needs to solve ever-larger and more complex engineering challenges. Today, this best-in-class solution has reached a new level of performance and user friendliness, capable of solving tens of millions of degrees of freedom quickly. Welcome to a new era of structural simulation.

By **Al Hancq**
Senior Director of Software Development – Mechanical
ANSYS

When the technology underlying ANSYS Mechanical made its debut in 1969, it was nothing short of revolutionary. For the first time, engineers could conduct finite element analysis (FEA) and other mechanical studies in an engineering-centric environment, making predictions about how products would perform before physical prototypes were ever constructed. It significantly reduced the time and financial investments needed to develop products, while also increasing engineers' confidence in their designs. The technology was nothing short of a game changer.

While early analysts had to define and construct their own FEA studies by hand — which required a deep understanding of physics, meshing and numerical FEA calculations — in the 1990s all that changed. I was part of a team that reimagined the software so that it could do more of the heavy lifting while also designing a more user-friendly interface.

Automating processes like FEA abstraction, contact specification and meshing significantly accelerated product development while supporting creativity and risk-taking. In nearly every industry, ANSYS Mechanical helped fuel a new level of speed and design exploration, resulting in many product innovations that changed our world.

Throughout the 2000s and 2010s, growing concerns about fuel efficiency and product lightweighting forced engineers to work with razor-thin material margins while still ensuring uncompromising safety in mission-critical applications. Simulations became ever more complex as engineers began to conduct multiphysics analyses, apply more operating parameters, and investigate both new materials and new manufacturing processes. And they had to accomplish all this with lower budgets, fewer engineers and tighter launch deadlines.

ANSYS responded by capitalizing on advancements in hardware, data management and processing, analytics, and high-performance computing to make ANSYS Mechanical equal to these challenges — faster and stronger than ever.

Extreme Power with Extreme Ease

Backed by decades of experience and world-leading technical expertise, ANSYS remains at the forefront of the next era of structural simulation.

Some recent revolutionary changes make ANSYS Mechanical even more powerful — with new capabilities in fracture and acoustics analysis — while also making it more accessible via a new user interface. More of the heavy lifting in ANSYS Mechanical happens behind the scenes, which means more members of any product development team will be able to fully utilize the software without the need for lengthy or extended training. This expands the use of simulation beyond traditional analysts and amplifies our customers' human resources.

This issue of *ANSYS Advantage* is packed with impressive examples of ANSYS Mechanical at work in the real world. These customer stories demonstrate how ANSYS Mechanical continues to deliver the extreme degree of accuracy and fidelity needed to ensure performance in the most demanding, mission-critical applications.

Ongoing Innovation: Our Commitment

The original users of ANSYS Mechanical would be stunned by the degree of complexity encountered by today's product development teams — and the way ANSYS software has evolved to meet these challenges. While early users of ANSYS Mechanical were solving models on the order of 10,000 nodes, now solving 10 million nodes is routine — and our solvers and algorithms are simply unmatched in their ability to handle these numerically large, incredibly complex structural problems.

Looking ahead, there's no limit to what can be achieved via simulation. In collaboration with other ANSYS solutions, ANSYS Mechanical enables the increasing adoption of system-level studies, additive manufacturing, digital twins and other forward-looking engineering practices. With its ability to gather and analyze millions of data points, ANSYS Mechanical will surely also support the future use of artificial intelligence and machine learning.

As new technology trends emerge and industry needs evolve, ANSYS Mechanical will remain the world's most innovative mechanical simulation solution and the industry standard for years to come.

© 2019 ANSYS, INC.

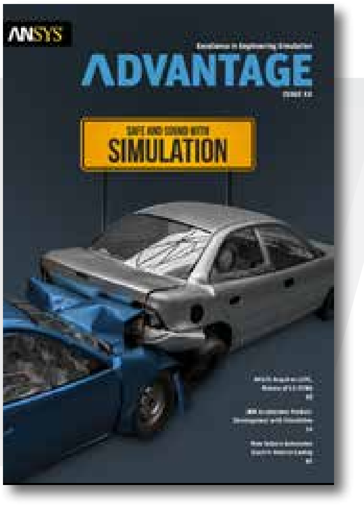
Table of Contents

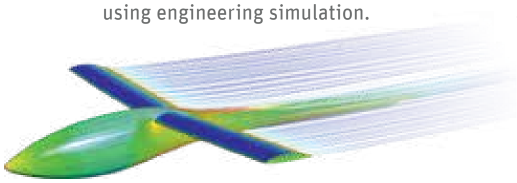
SPOTLIGHT ON MECHANICAL

ABOUT THE COVER

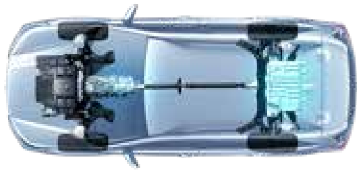
ANSYS has acquired LSTC, makers of the gold standard in automotive crash simulation, LS-DYNA. The cover image shows an LS-DYNA simulation after it has been separately rendered.

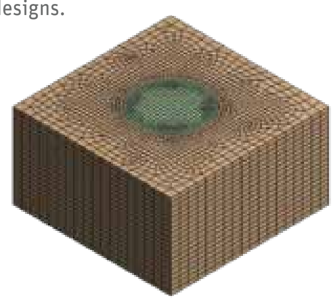
Rendered cover image courtesy of Ed Helwig.





SIMULATION@WORK





DEPARTMENTS

Welcome to *ANSYS Advantage*! We hope you enjoy this issue containing articles by ANSYS customers, staff and partners.

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

Executive & Managing Editors
Chris Reeves, Jamie Gooch, Beth Harlen

Editorial Advisers
Amy Pietzak, Tom Smithyman

Editorial Contributor
ANSYS Customer Excellence North America

Senior Editor
Tim Palucka

Editors
Erik Ferguson, Walter Scott, Kara Gremillion, Terri Sota, Mark Ravenstahl, Scott Nyberg

Art Director
Ron Santillo

Designer
Dan Hart Design

ANSYS, Inc.
Southpointe ,
2600 ANSYS Drive
Canonsburg, PA
15317
USA

Subscribe at ansys.com/magazine

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

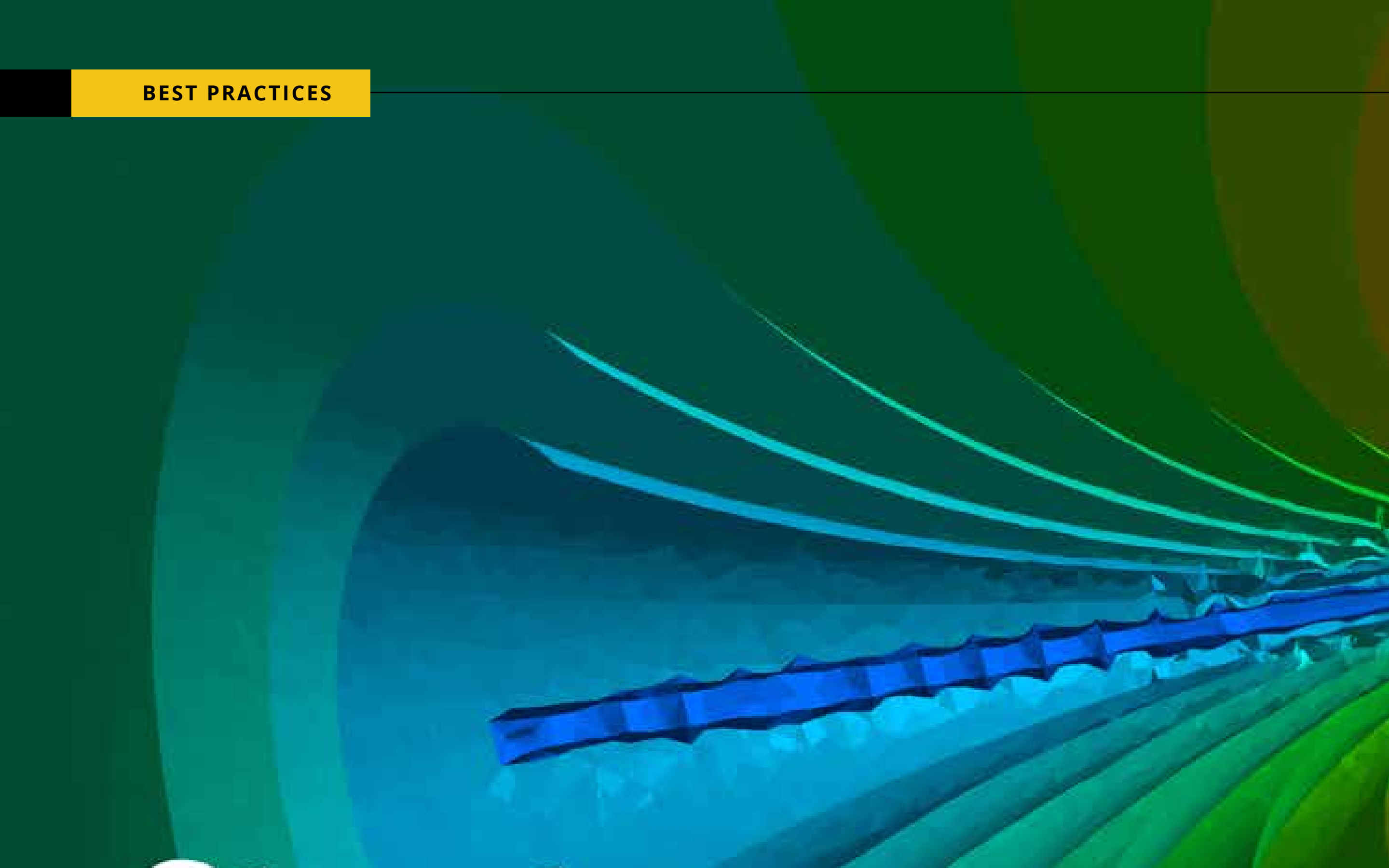
ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer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

Neither ANSYS, Inc. nor Dan Hart Design guarantees or warrants accuracy or completeness of the material contained in this publication.

Trademarks
ACT, Additive Print, Additive Science, Additive Suite, AIM, Aqwa, Autodyn, BladeModeler, CFD, CFD Enterprise, CFD Flo, CFD Premium, CFX, Chemkin-Pro, Cloud Gateway, Customization Suite, DesignerRF, DesignerSI, DesignModeler, DesignSpace, DesignXplorer, Discovery Live, EKM, Electronics Desktop, Elastic Licensing, Enterprise Cloud, Engineering Knowledge Manager, EnSight, Explicit STR, Fatigue, FENSAP-ICE, FENSAP-ICE-TURBO, Fluent, Forte, Full-Wave SPICE, HFSS, High Performance Computing, HPC, HPC Parametric Pack, Icepak, Maxwell, Mechanical, Mechanical Enterprise, Mechanical Premium, Mechanical Pro, Meshing, Multiphysics, Nexxim, Optimetrics, OptiSLang, ParICs, PathFinder, Path FX, Pervasive Engineering Simulation, PExprt, Polyflow, PowerArtist, Q3D Extractor, RedHawk, RedHawk-SC, RedHawk-CTA, Rigid Body Dynamics, RMxprt, SCADE Architect, SCADE Display, SCADE LifeCycle, SCADE Suite, SCADE Test, SeaHawk, SeaScape, SIwave, Simplorer, Solver on Demand, SpaceClaim, SpaceClaim Direct Modeler, Structural, TGrid, Totem, TPA, TurboGrid, Twin Builder, Workbench, Vista TF, Realize Your Product Promise, Sentinel, Simulation-Driven Product Development

ICEM CFD is a trademark licensed by ANSYS, Inc. LS-DYNA is a registered trademark of Livermore Software Technology Corporation. nCode DesignLife is a trademark of HBM nCode. All other brand, product, service, and feature names or trademarks are the property of their respective owners.

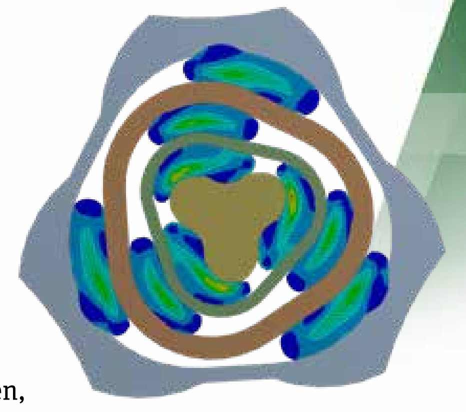
© 2019 ANSYS, INC.



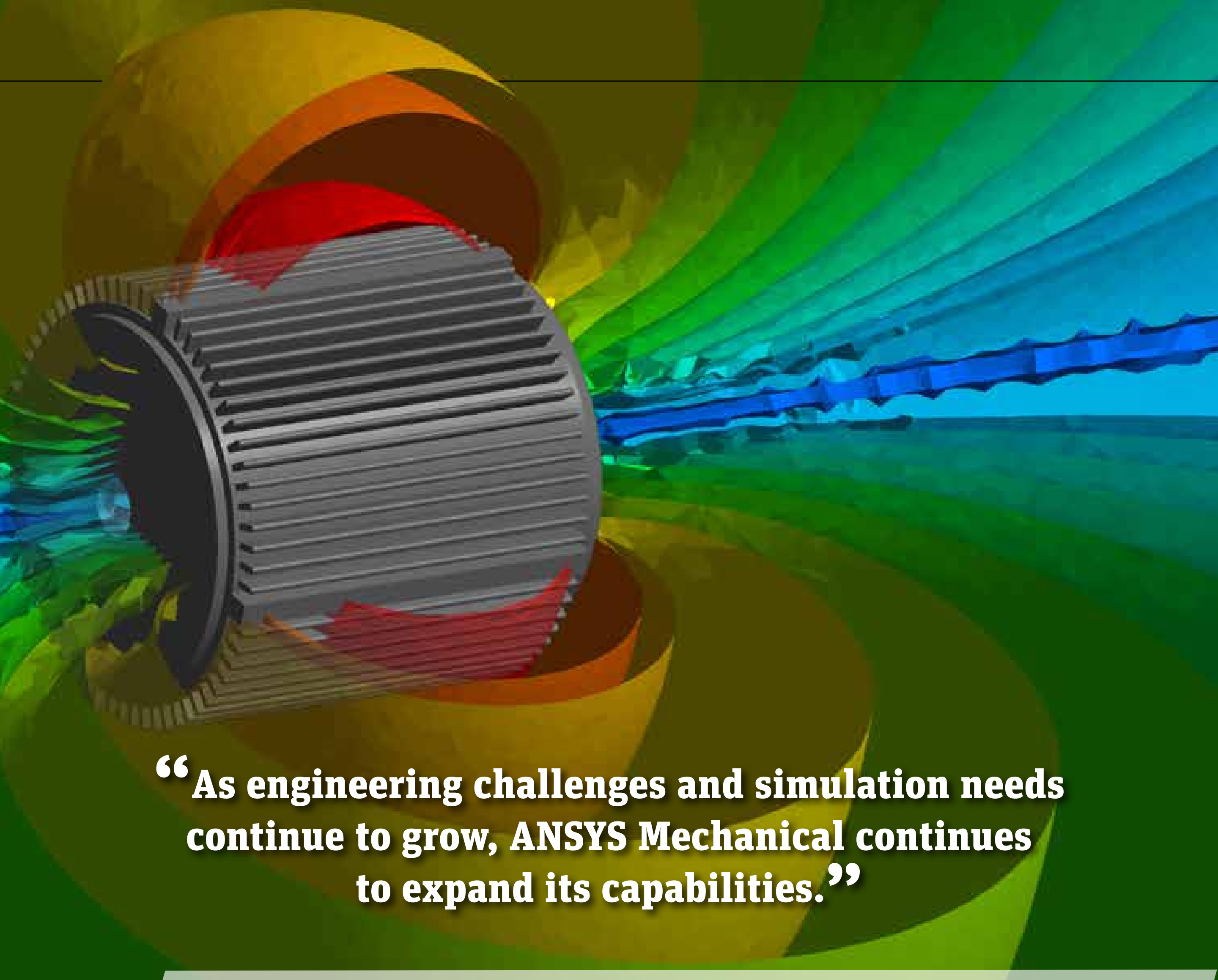
Structure a New Industry Standard

By **Richard Mitchell**
Principal Product Manager — Mechanical
ANSYS

When the technology behind ANSYS Mechanical was introduced in 1969, it changed everything — streamlining and accelerating the complex analysis required to ensure the structural strength and stability of product designs. Since then, it has been leveraged by product development teams in every industry to cut time and costs, without compromising performance in mission-critical applications. Recent updates to ANSYS Mechanical have made it even faster, more powerful and more capable of solving complex problems like fracture and acoustics. In addition, an enhanced user interface speeds adoption and simplifies common tasks — making the entire engineering team more productive.



Nonlinear Adaptivity (NLAD) in ANSYS Mechanical automatically remeshes models as the solution progresses.



“As engineering challenges and simulation needs continue to grow, ANSYS Mechanical continues to expand its capabilities.”

Acoustic features in ANSYS Mechanical include sound pressure level plots and waterfall diagrams to plot noise levels at various motor speeds in a single graphic. Recently added was the ability to create a sound file from the simulation.

ANSYS Mechanical is the ANSYS flagship structural simulation solution used by thousands of product development teams around the world and in every industry. ANSYS Mechanical was the first software to streamline, accelerate and automate the complex calculations involved in finite element analysis (FEA) and other structural studies. It has been used to develop hundreds of product designs that have, in turn, become industry standards.

The challenges faced by product development teams have changed significantly since the technology underlying Mechanical was developed in 1969 — and Mechanical has evolved to anticipate and answer these challenges. With thousands of users worldwide, ANSYS collaborates with customers and listens to their needs to deliver practical, value-added software enhancements. With the increase in global competition in every industry, more frequent product launches and more innovative designs are required. Software developers at ANSYS have responded with faster solvers and novel parallel processing schemes to drive enhanced speed and technical depth for structural analysis — capitalizing on improved hardware, cloud hosting and high-performance computing.

Growing concerns about energy efficiency — coupled with tightening regulatory standards — have forced structural engineers to explore advanced, lightweight materials and keep engineering margins as thin as possible. Ongoing enhancements to ANSYS Mechanical equip product development teams to tackle these issues head on by modeling new materials and making intelligent trade-offs between material weight and structural strength.

Companies are now evolving from highly trained structural engineers, often with doctoral degrees, to more diverse product development teams that include designers. ANSYS has accommodated this shift by embedding more complex capabilities in straightforward workflows — for example, fracture modeling or coupled field analysis — while making ANSYS Mechanical easier to learn and use.

This issue of *ANSYS Advantage* features many applications for ANSYS Mechanical that would have been impossible to imagine even 10 years ago, simply because of their physical complexity, solver times and numerical solution size.

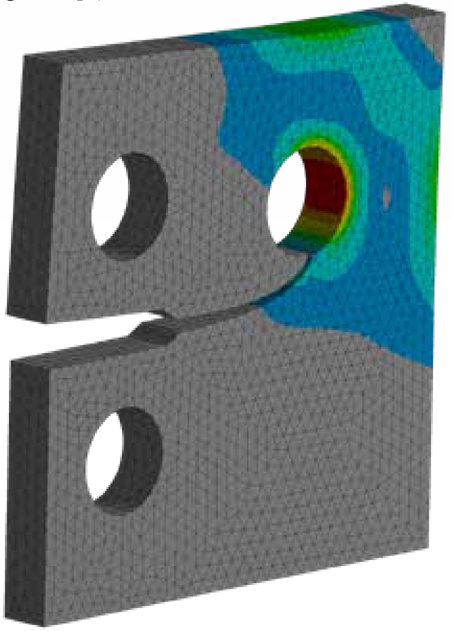
As engineering challenges and simulation needs continue to grow, ANSYS Mechanical continues to expand its capabilities.

FASTER, MORE INTUITIVE, EASIER TO USE

ANSYS Mechanical contains a re-imagined user interface that is designed to help all users leverage its simulation capabilities while increasing speed and productivity for the entire product development team.

Based on customer feedback, the intuitive interface was designed with four tenets in mind:

- **Ease of use.** The software is designed to improve workflows, reduce mouse travel, easily identify key functions and provide access to shortcut keys that save time. A contextual ribbon groups similar commands, improving accessibility to the features needed to complete the current task.
- **Discoverability.** Relevant tools and data are at hand, with larger icons for common tasks and easy-to-use search engines. An expanded Tool Tips feature offers practical information and guidance with just a few mouse clicks, along with easier navigation. A Quick Launch option helps users easily search for a desired feature or interface, then automatically access it.
- **Customization.** Users can customize ANSYS Mechanical through screen layouts, quick-access buttons and add-in user macros. A customizable graphics toolbar allows them to create new navigation systems that place their most used features within easy reach.
- **Extensibility.** Because ANSYS Mechanical is employed by engineers to perform many tasks, it features an improved ability to add extra functionality through extensions via external software and custom code. This means Mechanical's already expansive capabilities can be extended across many more engineering applications.

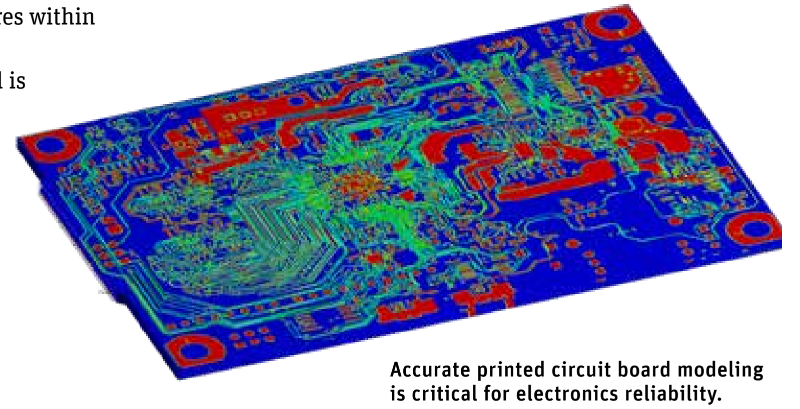


With SMART (separating morphing and adaptive remeshing technology) fracture modeling functionality in ANSYS Mechanical, crack growth analysis is not dependent on the mesh.

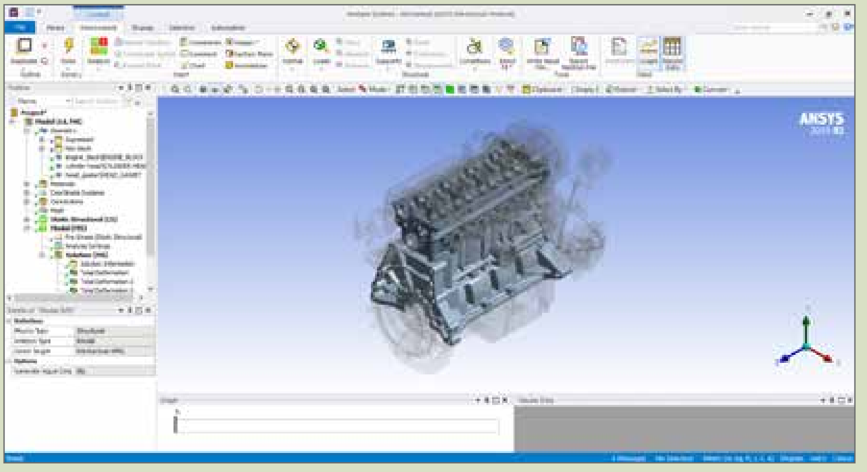
MORE POWERFUL MODELING CAPABILITIES

Increasing consumer demands, harsher operating conditions, tightening regulations, the incorporation of smart functionality — all these factors contribute to more complex problems and numerically larger solution sizes. ANSYS Mechanical features a range of functionality aimed at managing this complexity, while delivering a higher level of modeling and solving accuracy. A few of these new features include:

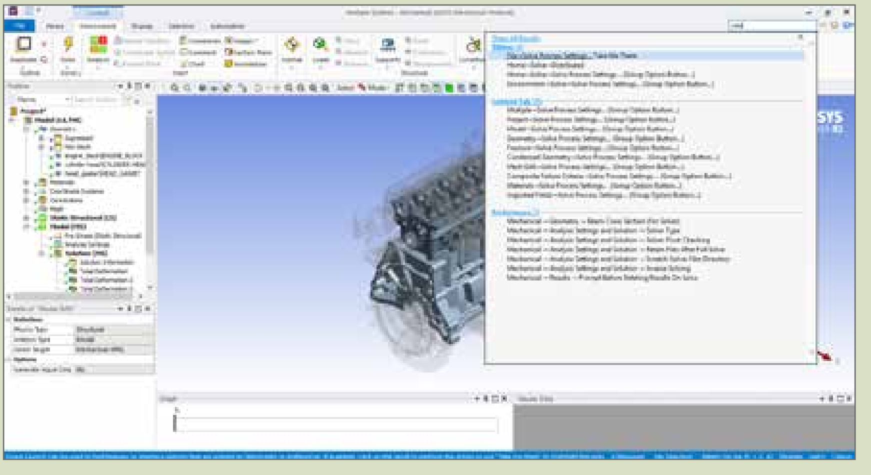
- **Mixed-dimension analysis.** Rendering and solving an entire, highly complex product design in 3D is time-consuming and expensive — and, often, it is simply not necessary to model the entire design in high detail. ANSYS Mechanical supports mixed-dimension analysis, in which structurally important parts of the design are modeled in 3D, while other aspects can be geometrically modeled in 2D, 2D-axisymmetry or even 1D. This balances the need for accuracy with greater speed and cost-effectiveness.
- **Enhanced SMART fracture modeling.** In the past, accurately predicting crack growth required expert analysis to construct the mesh correctly. Thanks to SMART (separating morphing and adaptive remeshing technology) fracture modeling functionality in ANSYS Mechanical, crack growth analysis is not dependent on the mesh — and will not be influenced by model setup. Fracture modeling is faster and more accurate, and requires much less special expertise.
- **An automatic mesh-based connection for shell and beam models.** Patent-pending batch



Accurate printed circuit board modeling is critical for electronics reliability.



ANSYS Mechanical has a new user interface designed to help different types of users leverage its simulation capabilities, allowing the entire product development team to be more efficient.



A Quick Launch option helps users easily search for a desired feature or interface, then automatically access it.

connection technology in ANSYS Mechanical provides a fast, fully automated, high-fidelity way to connect large beams and shell structures. No shared topology is required, and intersections among parts are resolved at the mesh level. Backed by a high-speed quad mesher, this functionality supports a meshing and connection time of just a few minutes, even when more than 15,000 beams and sheets are involved.

- **Improved acoustic analysis**. Advanced nonlinear capabilities in ANSYS Mechanical make it capable of assessing the complex problem of acoustic noise — which is becoming more critical because of stricter noise regulations. Features include waterfall diagrams and plotting of noise levels at various motor speeds in a single graphic. A sound file creation feature means engineers can hear what they are working on, well before it is built.

SUPPORT FOR NEW MATERIALS AND PROCESSES

ANSYS Mechanical includes expanded capabilities for capitalizing on two of the most important trends in product development today: the ever-increasing use of composite materials and the growing adoption of additive manufacturing (AM) production methods.

A specialized capability in Mechanical, ANSYS Composite PrepPost, features upgraded modeling capabilities so engineers can build models layer by layer — reflecting the way composite materials are actually structured. Product development teams can more accurately predict delamination and other sources of structural failure. In addition, improved algorithms more precisely capture failure modes and materials performance.

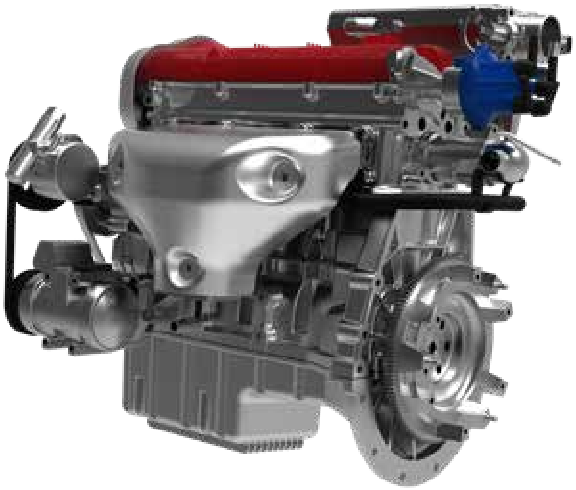
ANSYS Mechanical users can also simulate additive manufacturing workflows to optimize designs for a range of printer and material configurations. They can understand and predict the stresses and deformations that may occur as a result of thermal and structural stresses during the manufacturing process. Based on these insights, engineers can optimize the AM part build strategy and support structures upfront — minimizing the costs associated with materials waste and unnecessary test runs.

IMPROVED DATA ACCESS AND MANAGEMENT

Following the ANSYS acquisition of Granta Design, Mechanical now offers new capabilities that reflect Granta's expertise in materials data management, curation and selection. As development teams take advantage of next-generation materials and manufacturing methods like AM, Granta's data can help them make smart decisions, ensure simulation accuracy and more effectively predict how products will perform.

With more than 600 materials — including metals, plastics, composites and AM powders — now embedded within ANSYS Mechanical, product development teams can significantly reduce the time spent looking for material properties as they set up their models.

The ANSYS GRANTA Materials Data for Simulation data package, which is part of ANSYS Mechanical, covers virtually every material type and provides fast, easy access to the key data needed to perform structural analysis.



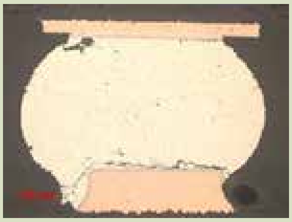
Simulating large assemblies requires efficient workflows and, often, many materials.

BUILDING FOR THE FUTURE

The recent enhancements to ANSYS Mechanical are diverse, ranging from universal improvements in the interface to changes in the way very specific problems, like fracture, are solved. All these updates have one thing in common: They are based on real-world feedback from the customers who use Mechanical every day.

ANSYS SHERLOCK: NEW CAPABILITIES FOR ELECTRONICS DESIGN

With the acquisition of DfR Solutions, ANSYS now offers a turnkey technology for analyzing the physics of failure for electronics systems and their components. Part of ANSYS Mechanical, Sherlock Automated Design Analysis helps electronics manufacturers meet the need for smaller, denser package sizes — while ensuring that products can withstand drops, heat, cold and moisture.

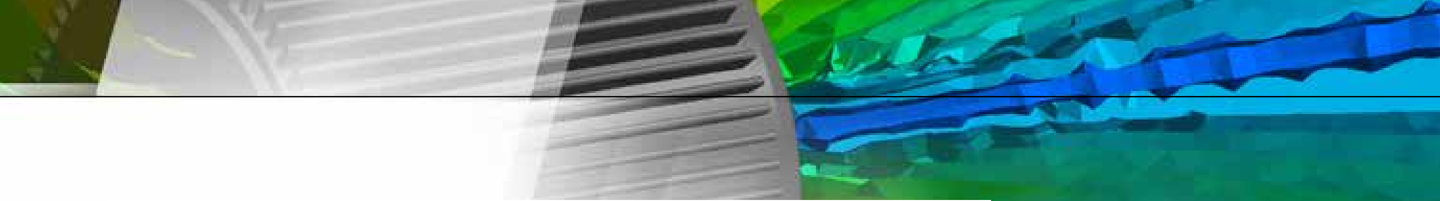


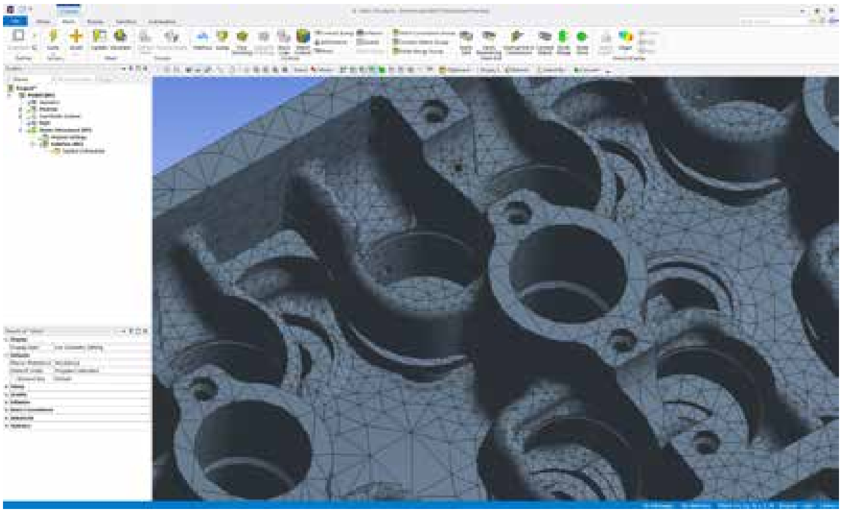
Engineers can use ANSYS Sherlock's validated algorithms to subject their product designs to temperature fluctuations, thermal cycling, power–temperature cycling, thermal shock, random vibration, harmonic vibration, mechanical shock and flexure. By identifying and addressing any potential performance or manufacturing issues early in the design cycle, engineering teams can launch innovative products much faster, and at a lower cost.

In building a virtual product model, development teams can choose from Sherlock's library of over 500,000 components — including a variety of parts, packages, materials, solders and laminates — as they quickly generate an FEA model. They can simulate a range of operating conditions on this model by capitalizing on Sherlock's close integration with other ANSYS solutions, as well as third-party CAE tools.

A significant portion of product development costs for electronics are spent on the test–fail–fix-repeat cycle. By enabling electronics development teams to simulate the lifetime performance of their products at the component, board and system levels — in the earliest stages of design — ANSYS Sherlock helps minimize that financial investment, while maximizing product integrity.

Reliability Physics in Your Design Process
ansys.com/reliability-physics



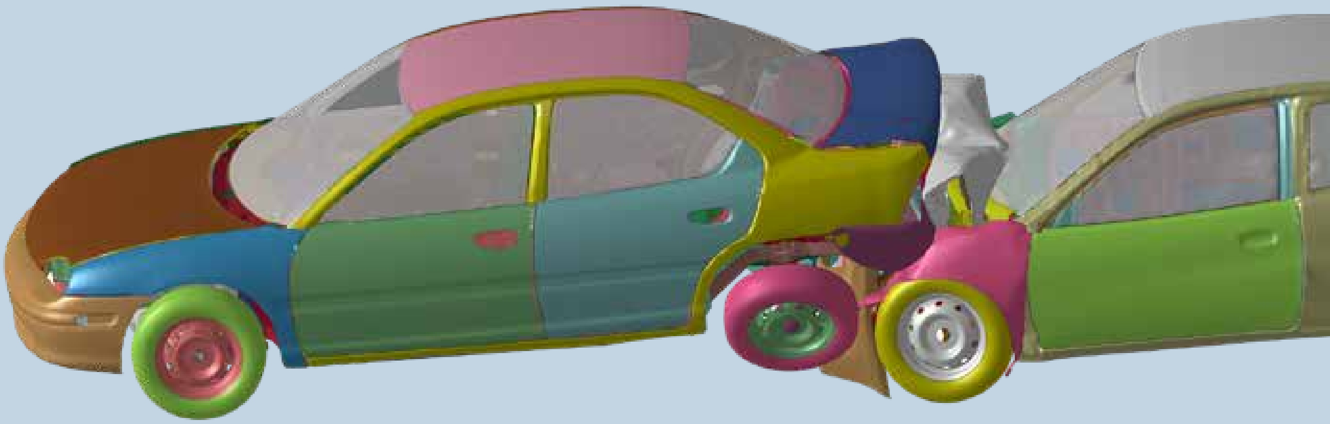


Tools that are relevant for the task users are working on help speed workflows inside ANSYS Mechanical.

At ANSYS, we are focused on understanding our customers' challenges and responding with best-in-class simulation capabilities that are constantly evolving. These capabilities go beyond ANSYS Mechanical to encompass our entire portfolio. Mechanical may be our flagship product, but it integrates with our newest products, like ANSYS VRXPERIENCE, which can be used to generate audio files to better address complex problems like acoustics.

Fifty years after the ANSYS FEA software technology revolutionized product design and development, ANSYS remains committed to identifying the most innovative, advanced simulation capabilities and placing them into the hands of users — where they can have a dramatic impact on our customers' success. No matter how the business world and technology continue to evolve, that will remain our customer promise.

© 2019 ANSYS, INC.



ANSYS Expands Access to Advanced Multiphysics Simulation with LSTC Acquisition

By **Siddarth Shah**, Principal Project Manager, Structures, ANSYS

By acquiring longtime partner Livermore Software Technology Corporation (LSTC), ANSYS can more deeply integrate LS-DYNA's explicit dynamics solver to make it easier for engineers with different skill sets to solve short-duration events. The code's origins lie in highly nonlinear, transient dynamic finite element analysis (FEA) using explicit time integration, such as those that occur during automobile crashes, bird strikes on aircraft and explosions. The pairing of the ANSYS and LS-DYNA solvers creates a solution that helps engineers understand the elaborate combination of nonlinear phenomena found in such events.

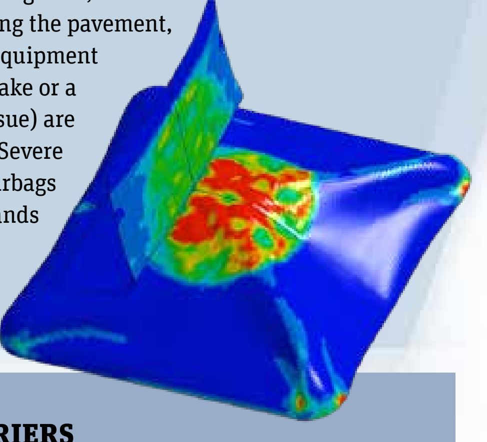
With a focus on speed and accuracy, LS-DYNA, the LSTC flagship product, has been the gold standard in the automotive industry for crashworthiness and occupant safety simulation for decades. LSTC estimates that LS-DYNA is the primary crash analysis tool for over 80% of the world's major automotive manufacturers, and that the code is used by 90% of tier 1 suppliers. It excels at simulating the response of materials subject to short periods of severe loading, such as those that occur during crashes, drops and even metal forming.

ANSYS customers have benefited from the combined strengths of ANSYS Mechanical and LS-DYNA since ANSYS LS-DYNA was first released in 1996. Integration of LS-DYNA with ANSYS Workbench in 2013 made it easy for engineers of different experience levels to perform LS-DYNA simulations using the familiar Workbench user interface.

The combination of ANSYS Workbench and LS-DYNA has helped explicit dynamics simulation expand beyond experts with specialized domain knowledge. At the same time, access to the computational power needed to perform multiphysics simulations has become more affordable, allowing LS-DYNA's scalability to shine on models with higher and higher degrees of freedom.

MULTIPLE DOMAINS, PARTS AND PHYSICS IN ONE SOLUTION

Engineers of all types now use ANSYS LS-DYNA to study product behavior via simulations involving severe material deformation or failure. The software makes it easy to study the interaction between parts, enabling users to easily evaluate how parts and assemblies behave together, or how the product behaves as part of a larger system. A bike helmet striking the pavement, a turbine engine containing fragments of a broken blade, defense equipment reacting to a blast, a hydroelectric dam being rocked by an earthquake or a server crashing to the floor (as IBM illustrates on page 14 of this issue) are all examples of interactions that have been studied with LS-DYNA. Severe material deformation behavior could be related to anything from airbags deploying to metal bending during hydroforming. There are thousands of such short-duration events and contacts that could be analyzed using ANSYS LS-DYNA.



ANSYS LS-DYNA can simulate explicit events, such as delamination. >

CRASHING THROUGH BARRIERS

When John Hallquist began developing a solver with explicit time integration to study nonlinear dynamic problems while working at Lawrence Livermore National Laboratory in 1976, he couldn't have predicted its evolution into LS-DYNA, the flagship of LSTC, which he founded in 1987. Over the years, LS-DYNA's functionality has been expanded. Below are just a few examples of the different analyses and methods LS-DYNA supports.

- Arbitrary Lagrangian Eulerian Method
- Incompressible Computational Fluid Dynamics
- Conservation Element/Solution Compressible Fluids
- Discrete Element Method
- Electromagnetism
- Element-Free Galerkin Method
- Fluid–Structure Interaction
- Implicit Simulations
- Noise, Vibration and Harshness
- Smooth Particle Hydrodynamics
- Thermal Transfer

In addition to the interaction of parts, ANSYS LS-DYNA also features lesser known but powerful capabilities to solve strongly coupled multiphysics problems that cannot readily be evaluated by physical testing. The software's explicit and implicit time stepping makes it possible to simulate static and dynamic tests with the same model. Simulations such as these are possible thanks to the ability of the LS-DYNA solver to handle multistage, multiscale, multiphysics problems with one solver: the internal shorting behavior of an electric vehicle battery; the noise, vibration and harshness (NVH) of a golf club hitting a ball; the splashing and hydroplaning behavior of a car tire driving through a pool of water or even the complex behavior of an aortic artificial heart valve opening and closing as blood pumps through it.

Viewing a rendered LS-DYNA crash simulation from below.
Rendering courtesy of Ed Helwig.

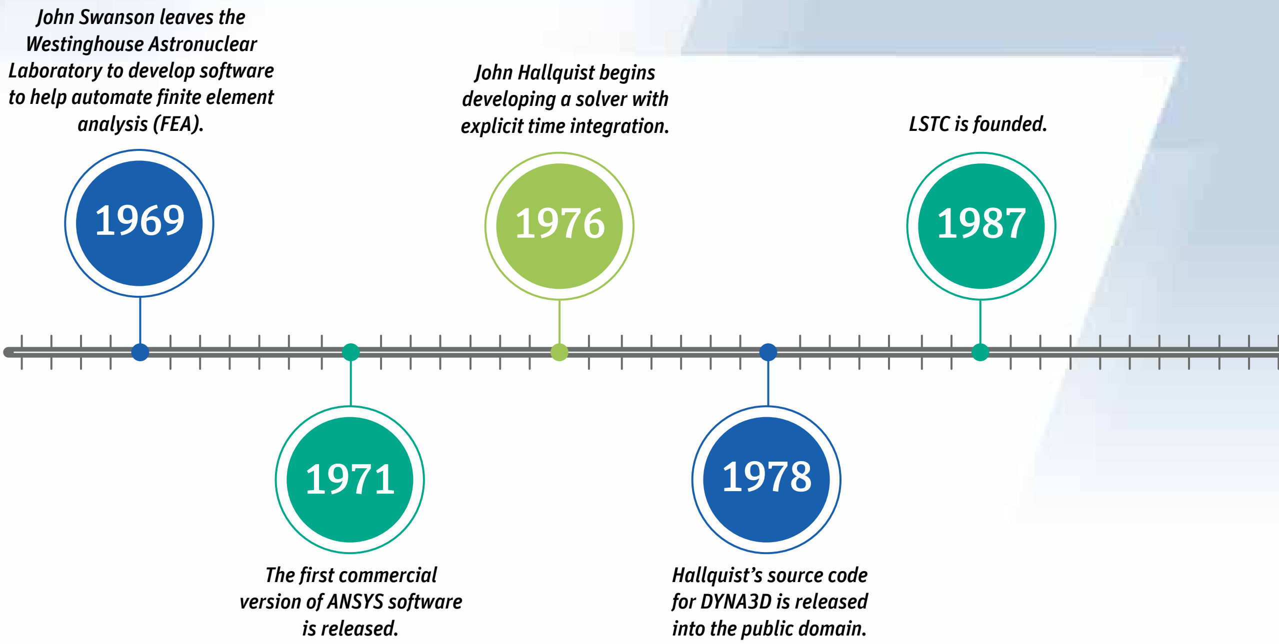
The left side of a vehicle after crash testing. Simulation can be used to minimize the number of physical tests needed.
Image courtesy of National Highway Traffic Safety Administration.

BETTER TOGETHER

The unified ANSYS LS-DYNA solution delivers the pre-processing and post-processing tools available in the ANSYS Mechanical environment — including automated meshing and the ability to define material, contact, and loads and boundary conditions. Engineers can perform parametric studies on CAD geometries that are ready for explicit analyses without leaving the intuitive ANSYS Workbench environment. ANSYS LS-DYNA Workbench users can access ANSYS SpaceClaim for geometry modeling, as well as bidirectional computer-aided design (CAD) connectivity.

ANSYS LS-DYNA provides a wide range of accurate low- and high-order element formulations in solids, shells and beams. These can be applied on a per-part basis through the ANSYS Mechanical interface, so engineers can place high-fidelity elements only in the areas where they are necessary to save time. Contact can be automatically detected between separate parts, within parts and within individual elements.

Engineers and simulation analysts using ANSYS Mechanical and LS-DYNA find that the Workbench integration can decrease their time to solution. Part of that speed increase stems from LS-DYNA's parallel calculations that can scale from desktop computers to clusters of thousands of processors using Linux, Windows and UNIX. ANSYS LS-DYNA users also benefit from the workflow efficiencies and ease of use inherent in ANSYS Mechanical.



MORE TO EXPLORE

With the acquisition of LSTC by ANSYS, customers of both companies can look forward to even deeper integration of technologies that provide the best of both worlds. The goal of the acquisition is to help our customers meet the challenges of engineering the complex systems of today and tomorrow, such as those in autonomous and electric vehicles.

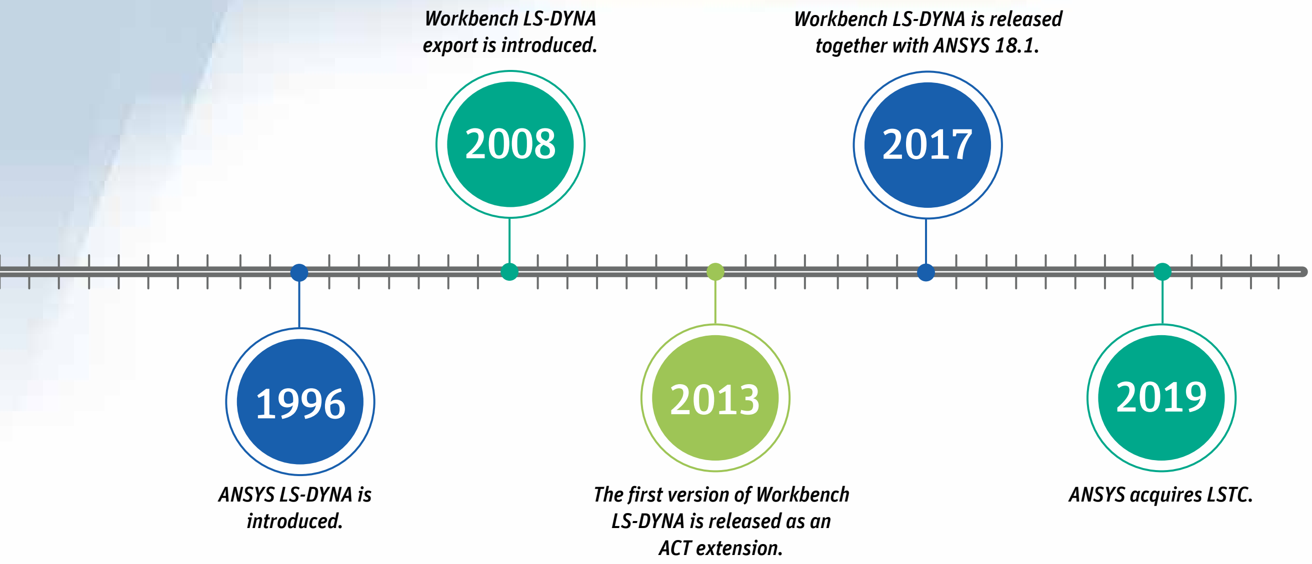
"Leading companies in every industry are using simulation to develop more-complex products, faster and at lower cost than ever before," says Ajei Gopal, ANSYS president and CEO. "To meet those challenging market demands, our customers need best-in-class solutions that enable them to innovate at the speed of thought."

Deeper integration between LS-DYNA and ANSYS Workbench will help optimize the multiphysics product design and development workflows needed to capitalize on disruptive technologies such as electrification, autonomous vehicles and 5G as they continue to spread beyond automotive, aerospace and telecommunications into virtually every industry. Bigger, more complicated problems require faster, more accessible solutions — and that's what ANSYS is providing its customers with the acquisition of LSTC.

As ANSYS tightens the integration of LS-DYNA into Workbench, customers can be assured that they will be at the heart of the company's decision-making processes.

"I expect that the combination of Workbench and LS-DYNA will expand our user base by at least an order of magnitude," says John O. Hallquist, founder of LSTC. "Here at LSTC, nothing makes all of us happier than when our research enables more customers to imagine, design and implement ambitious projects that were previously impossible." Λ

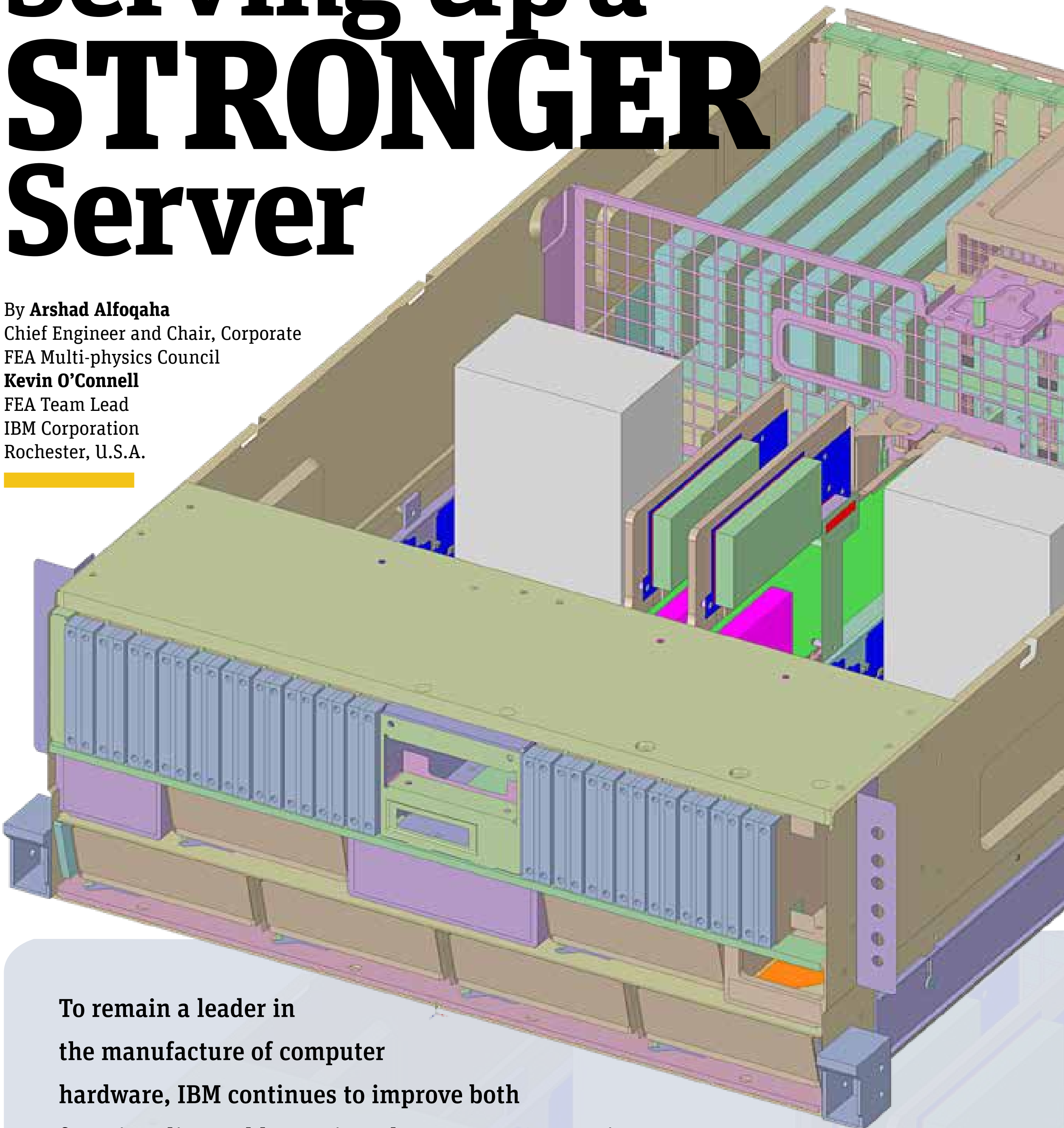
LS-DYNA Applications
Automotive Crash & Occupant Safety NVH & Durbility Battery Reliability
Aerospace Bird Strike Blade-off Containment Crash
Manufacturing Forming Stamping Machining
Consumer Products Packing Switches
Civil Engineering Blast Proofing Earthquake Safety Tents
Electronics/Hi-Tech Drop Analysis Package
Defense Projectiles and Weapons Blast and Penetration Underwater Shock Analysis
Bio-Medical Devices & Equipment Medical procedures



© 2019 ANSYS, INC.

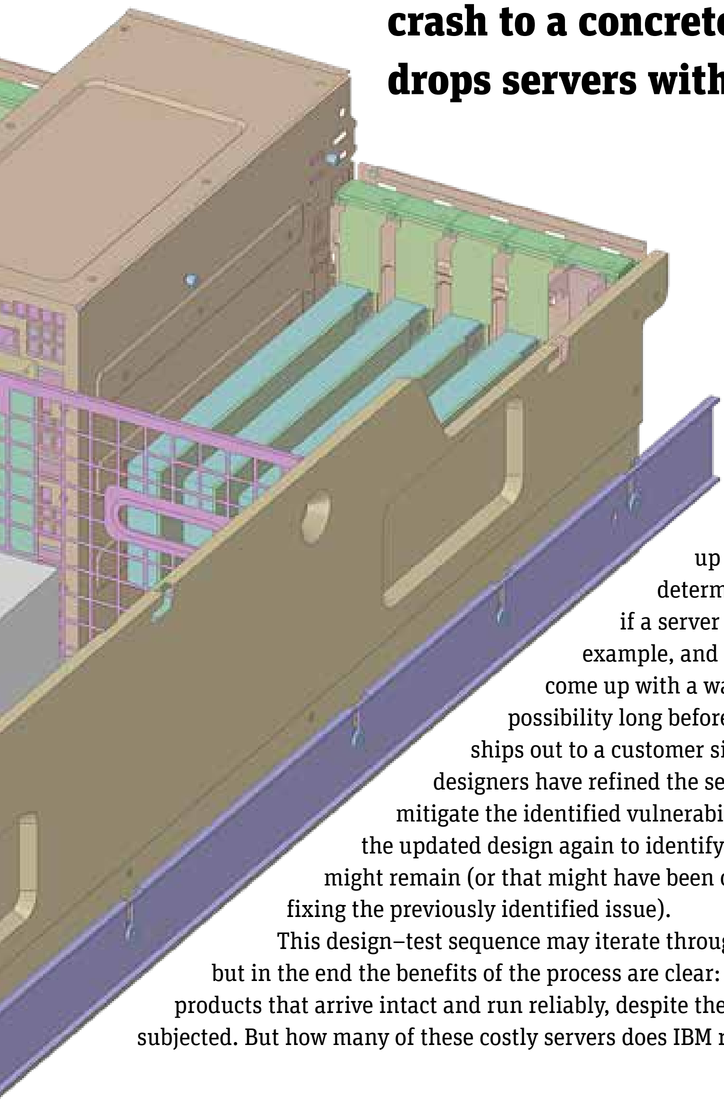
Serving Up a STRONGER Server

By **Arshad Alfoqaha**
Chief Engineer and Chair, Corporate
FEA Multi-physics Council
Kevin O'Connell
FEA Team Lead
IBM Corporation
Rochester, U.S.A.



To remain a leader in the manufacture of computer hardware, IBM continues to improve both functionality and longevity. The company's premier servers can command a premium price, but only if the hardware can stand up to the demands of the environment and ensure superior reliability. IBM ensures the integrity of its system designs with help from ANSYS Mechanical and ANSYS LS-DYNA.

Intro to the Workbench LS-DYNA User Interface
ansys.com/ls-dyna

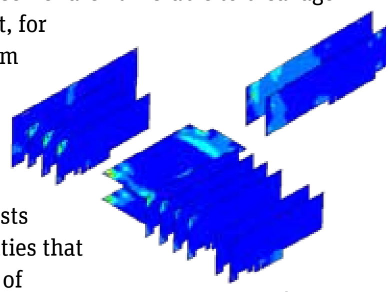


"Instead of watching 30 or 40 physical servers crash to a concrete floor, the MSPE team virtually drops servers within ANSYS LS-DYNA."

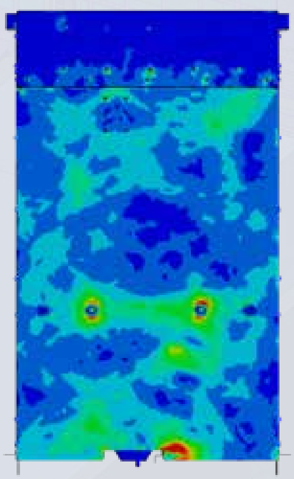
A bare metal IBM Power System S924 server — rack-mountable, four standard rack units (4U) in height — weighs in at 88 pounds when loaded up with a full complement of multicore IBM POWER9 processors, solid-state storage devices and communications cards. The engineers on the mechanical simulation and predictive engineering team (MSPE) at IBM get paid to drop them on a concrete floor.

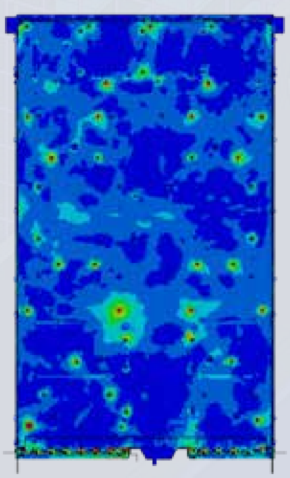
There is a purpose to this, of course. The MSPE team tests the physical design of the system to help the new product designers build a server that will stand up to the rigors of the real world. The MSPE team's job is to determine what parts of the server are vulnerable to breakage if a server slips during shipment, for example, and to help the design team come up with a way to mitigate that possibility long before the first server ships out to a customer site. After the system designers have refined the server's design to mitigate the identified vulnerability, the MSPE team tests the updated design again to identify any other vulnerabilities that might remain (or that might have been created in the process of fixing the previously identified issue).

This design–test sequence may iterate through 30 or 40 testing cycles, but in the end the benefits of the process are clear: IBM can deliver leading products that arrive intact and run reliably, despite the bumps, drops and strains to which they might be subjected. But how many of these costly servers does IBM really want to drop during the design process?

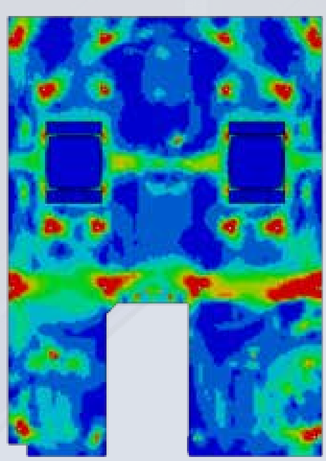


Upper maximum principal strain for the 2U peripheral controller and power supply cards





Von Mises stress in top cover (left) and bottom chassis (right) for 2U Power System



Upper maximum principal strain for the 2U main PCB board

A 4U IBM Power System Server with the cover off

A 2U IBM Power System Server with the cover off

“Simulation helps accelerate product development while simultaneously reducing design cycle times and costs.”

Thanks to software from ANSYS, the MSPE team at IBM can keep that number as close to one as anyone could reasonably hope. The team has found that the measurements they obtain when conducting physical drop tests of these expensive servers are almost the same as simulated drop-test results obtained using ANSYS LS-DYNA. Instead of watching 30 or 40 physical servers crash to a concrete floor, the MSPE team virtually drops all those servers within ANSYS LS-DYNA.

ITERATING THE DESIGN

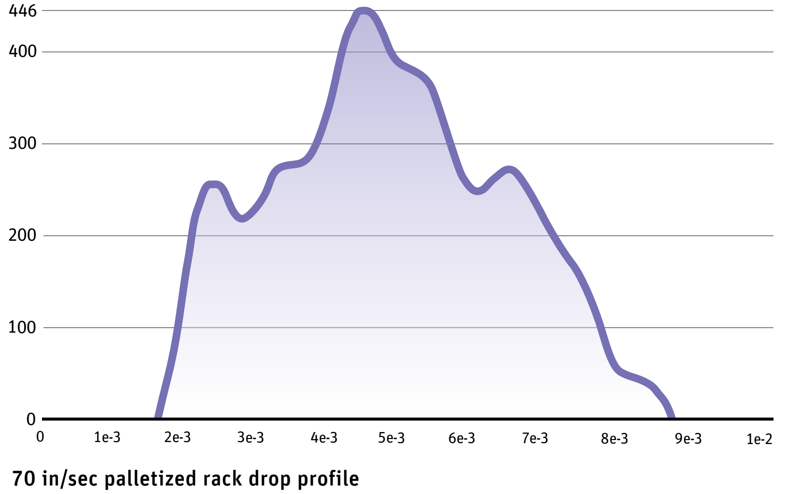
ANSYS Mechanical and ANSYS LS-DYNA lie at the heart of the simulation and testing efforts at IBM. The design team provided the MSPE team with raw 3D CAD files of the Power series servers (which are configurable in both 2U and 4U versions). Using ANSYS SpaceClaim with ANSYS Mechanical, the MSPE team optimized those 3D CAD files for finite element analysis (FEA). To reduce meshing and computation time for the Power server simulation tests, the MSPE team removed or simplified all the server features and geometry that would have no significant effect on the structural behavior of the systems.

The fully configured 2U and 4U models consisted of approximately 3,000 bodies with a combined count of over 500,000 shell and solid elements. The solid elements include four-node tetrahedral and eight-node hexagonal linear explicit elements; the shell elements include three-node triangular and four-node quadrilateral explicit elements.

SIMULATING A SUDDEN SLIP

IBM wanted to understand what would happen to the physical structure of a Power server system during shipment. What happens to the rails, rear brackets and front latches if the shipping vehicle itself is bouncing down the road? What would happen to the critical components themselves — the motherboard and the cards clipped into the slots on the motherboard?

To answer these questions, the MSPE team imported the ANSYS Mechanical files into ANSYS LS-DYNA. They then loaded simulations to replicate the dynamics of a palletized rack drop test. The simulation used the nonlinear explicit solver to model the server falling at a rate of 70 inches per second, with a peak acceleration of

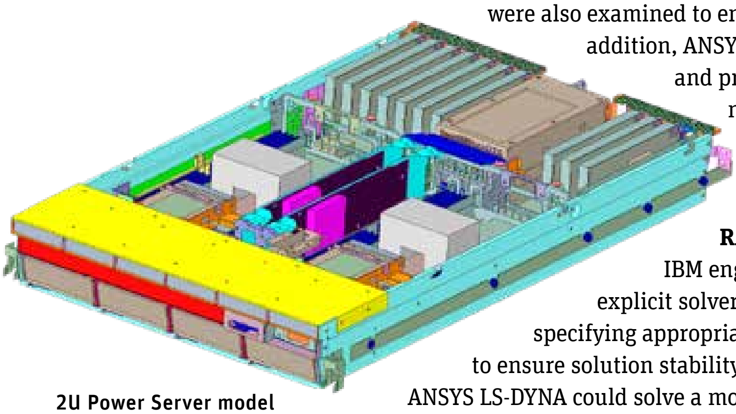


70 in/sec palletized rack drop profile

“The teams were able to conduct more than 30 design/test cycles in a matter of weeks, a fraction of the time it would have taken without ANSYS simulation tools.”

approximately 50 g for 8.2 ms. Standard earth gravity was loaded into the simulation as approximately 385 in/sec^2 applied vertically.

During the explicit solution, results were checked to ensure accuracy. The energy ratio was plotted to check its value (which should be around 1.0). Kinetic energy, total energy, internal energy, damping energy and sliding energy were also examined to ensure that they were within acceptable limits. In addition, ANSYS LS-DYNA predicted stresses (such as von Mises and principal), strains (such as effective plastic and maximum principal elastic), internal energy, and nodal displacements and acceleration for all components in the systems.



2U Power Server model

RAPID REFINE AND RETEST

IBM engineers on the MSPE team note that the LS-DYNA explicit solver handles nonlinearities with relative ease — specifying appropriate time steps of approximately 1 x 10^{-7} seconds to ensure solution stability. When pulse durations were brief (about 30 ms), ANSYS LS-DYNA could solve a model in five to seven hours. If the pulse duration extended to 90 ms, it could take ANSYS LS-DYNA 15 to 20 hours to solve the simulation. The model utilized 18 cores of a 3.10 GHz Intel Xeon CPU along with 128 GB of RAM.

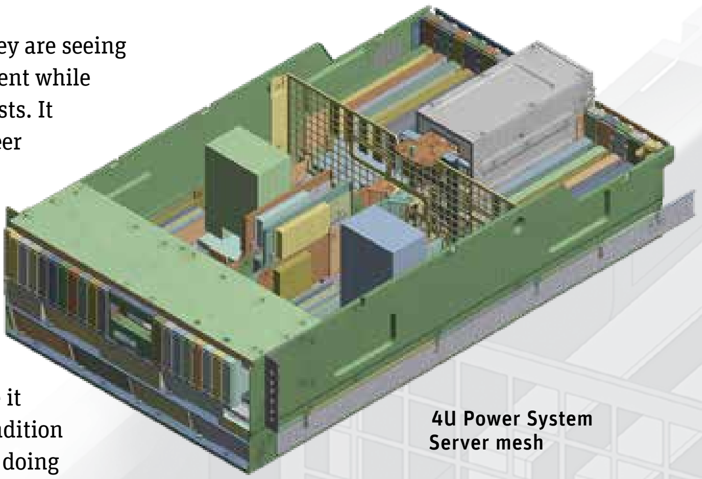
Because the MSPE team can provide insight to the design team within hours — and because the design team can quickly remodel components right in the CAD files, which the MSPE team can then bring right back into ANSYS LS-DYNA for retesting — the Power system design and testing teams were able to conduct more than 30 design–test cycles in a matter of weeks. That is a fraction of the time it would have taken to achieve the same design goals had IBM not been using ANSYS simulation solutions.

And, as noted, the MSPE team’s reliance upon simulation significantly reduced the costs associated with server development. Early on, when the MSPE team did conduct a real-world drop test of fully configured 2U and 4U Power server systems, they discovered that the measurements derived from their strain gage rosettes were in strong agreement with the measurements derived via ANSYS LS-DYNA. The test verified that further physical destruction of these costly servers was unnecessary. The results that ANSYS LS-DYNA produced were just as accurate and far less costly.

SIMULATION IS A BUSINESS BEST PRACTICE

Executives at IBM have noticed these outcomes. They are seeing that simulation helps accelerate product development while simultaneously reducing design cycle times and costs. It streamlines resources, too, as a single MSPE engineer using ANSYS LS-DYNA can conduct all the tests associated with a given development project. This enables other members of the team to work on other projects in parallel.

While hearing praise from IBM management is always gratifying, there is one group — the customers — from whom the MSPE team at IBM hears very little. But that feels even better, because it means that the servers are arriving in excellent condition and that the members of the MSPE team at IBM are doing their jobs well.



4U Power System Server mesh

Automating *the* Daily Grind

By **Michael Butler**
Project Manager and
Mechanical Design Engineer
ANCA Machine Tools
Melbourne, Australia

To develop a computerized numerical control (CNC) machine that could run autonomously with no human intervention for 50 hours, ANCA Machine Tools engineers turned to ANSYS Mechanical simulations for help. The solution helped a metal-cutting tools company to eliminate the night shift and free up weekends for their employees without reducing their production load.

Machine tools have been used for centuries to shape wood, metal or other solid materials. Machining generally involves cutting, boring, grinding or shearing a solid block to achieve the desired final shape of the part. Until the advent of modern computing systems in the second half of the last century, highly skilled human operators manually adjusted the position and speed of the cutting or grinding tool to machine the part to the required tolerances. Computers largely transformed machinists into high-tech craftsmen who program the speed and orientation of the cutting tool for automated operation.

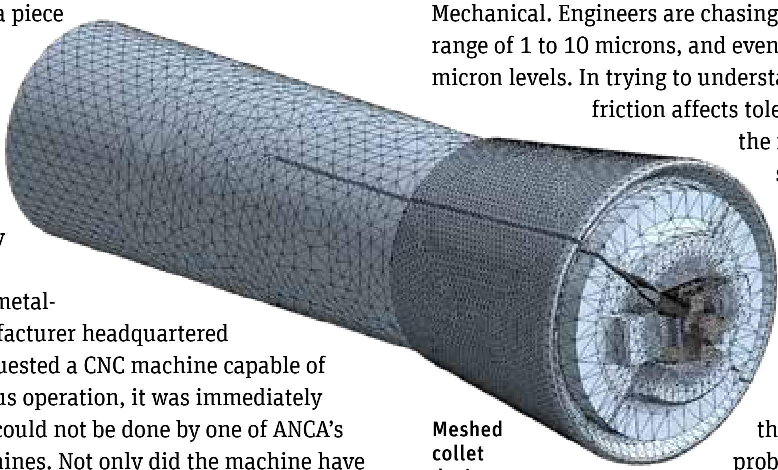
ANCA Machine Tools, founded in Australia in 1974, offers a comprehensive range of computerized numerical control machines that can produce cutting tools — milling tools, drills, taps, indexable inserts, etc. — with the flexibility provided by five-axis machining technology. The five axes include the standard x, y and z spatial axes along with two rotation axes (a and b) around the x and z axes. Using five axes instead of three ensures that a piece can be oriented at all angles needed for complete machining without the need for manual reorientation at any step in the process.

When Fraisa, a metal-cutting tools manufacturer headquartered in Switzerland, requested a CNC machine capable of 50-hour autonomous operation, it was immediately apparent that this could not be done by one of ANCA's four standard machines. Not only did the machine have to operate unmanned for that long, it also had to be able to switch among several tooling operations during this time, so different products could be manufactured. This required a robotic arm to load and unload pieces from a rotary table containing seven pallets — metal trays with circular holding cells for the feedstock and the finished parts. It also required automated gauging and measurement of finished parts for quality control. A customized CNC machine was needed.

SIMULATION AT ANCA

ANCA has been using ANSYS Mechanical solutions for their customization needs for four years. Prior to that, they tried simulation software from other suppliers but found the software to be hard to learn and use. Only four ANCA engineers were trained to use the simulation software for the frequent custom orders the company received, which created a design bottleneck. After the introduction of ANSYS Mechanical, the engineers found it much easier to learn how to set up and run simulations. Today, 20 ANCA engineers, including all the design and manufacturing engineers, use ANSYS Mechanical regularly.

The increasing demand for tighter tolerances in machining is one reason ANCA needs ANSYS Mechanical. Engineers are chasing tolerances in the range of 1 to 10 microns, and even sometimes sub-micron levels. In trying to understand how contact friction affects tolerances in this range, the nonlinearities of surface friction and how they affect deformation and surface pressure are quite complicated. ANSYS Mechanical simulations solve these complex problems.



Meshed collet design

At the macro scale of the five-axis machines, ANCA engineers use ANSYS Mechanical to understand how the large bodies and complicated movements of five-axis machines affect the machine's modal and harmonic response. They then feed that information back into the simulation to determine how modal and harmonic responses affect the control system. Vibrations and resonances at the macro scale can throw off precision at the micro scale; ANSYS Mechanical is valuable at both ends of the size spectrum.

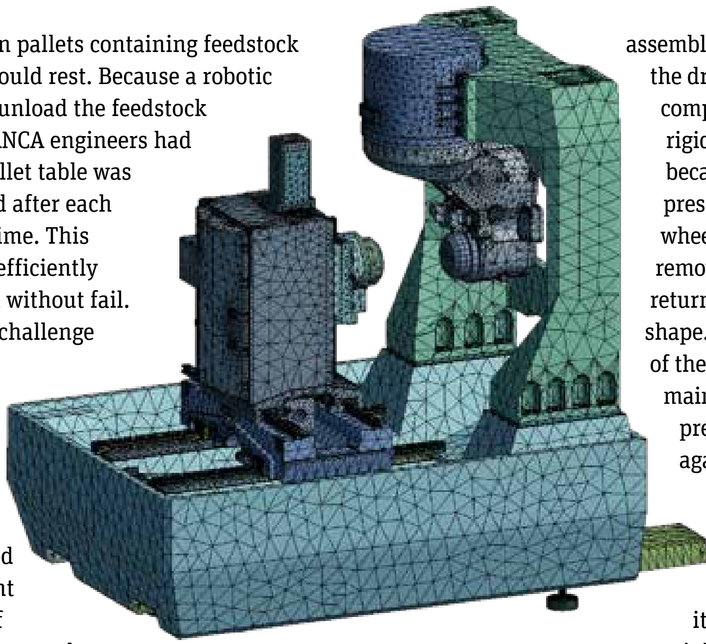
THE FRAISA CHALLENGE

The main challenge presented by Fraisa's request was ensuring the stiffness and rigidity of the rotary table

Powered Up for Accuracy - eBook
ansys.com/accuracy

upon which the seven pallets containing feedstock and finished parts would rest. Because a robotic arm would load and unload the feedstock and finished parts, ANCA engineers had to ensure that the pallet table was accurately positioned after each rotation, time after time. This enables the robot to efficiently pick up the next part without fail.

This was a major challenge because the rotary table supports seven pallets containing between 48 and 100 tools of a variety of weights and sizes. The total weight could be hundreds of kilograms, so the downward deflection of the rotary table had to be considered.


Meshing a CNC machine component for modal analysis

Another challenge was to optimize the design of the grinding wheel dresser for this automated system. A grinding wheel is a solid abrasive wheel used for various grinding and smoothing operations. During use, the face of a grinding wheel might become clogged with machined particles or distorted from its true circular shape, rendering it inefficient for grinding. A wheel dresser, often made of diamond, is pressed against the rotating wheel to trim and clean its surface and return it to its optimal condition. On a manned machine, the dressing operation might be performed by a human operator who presses the dressing tool against the wheel. However, the dressing tool must be included as part of an automated machine. Because there is not a lot of space available in ANCA's standard machines for this tool, engineers must optimize the rigidity of the dresser parts to make sure the dresser would fit into the physical space while maintaining its structural integrity.

ANSYS Mechanical helped ANCA engineers solve both of these challenges for Fraisa.

ANSYS MECHANICAL SOLUTIONS

ANCA engineers used ANSYS Mechanical to measure the deflections on the pallet table based on the changing loading conditions during operations. Deflection increased as parts were added to the pallet and decreased as they were removed. By reviewing the continually changing stresses on the system, they were able to optimize the design and increase the assembly's overall stiffness at the robot load and unload points.

For the machine side dresser, engineers conducted ANSYS Mechanical simulations on the deflection of the assembly to minimize the size of the dresser mounting without compromising the system's rigidity. Rigidity is important because the dresser must be pressed against the grinding wheel with sufficient force to remove embedded particles and return the wheel to its circular shape. Minimizing the deflection of the dresser mounting maintains the necessary pressure of the dresser against the wheel, which is critical to maintaining the true wheel form.

Engineers performed multiple iterations of both the pallet rigidity and the dresser mount deflection until the simulations converged on a solution. The rapid simulation iterations in ANSYS Mechanical made the process quick and easy.

HOW ANCA BENEFITS FROM ANSYS SIMULATIONS

ANSYS Mechanical simulations give ANCA engineers confidence in their design work. At a minimum, without simulation it would have taken an extra four to five weeks to build a prototype of the Fraisa system and test it. Simulation cut the required design time by about 25% overall, to under six months. It allowed them to validate multiple design iterations and have confidence in the designed solution going into final design testing. Because they were designing a one-off custom solution, ANCA engineers really could not afford to prototype, get it wrong, scrap the solution and try again. They had to have confidence that their design was right the first time. Fraisa is using the 50-hour autonomous machining solution now to eliminate the nighttime shift and weekend work for their employees.

ANCA Machine tool is supported by ANSYS elite channel partner LEAP Australia.

Driving Down Time to Market

Danfoss drives are compatible with all shown motor types.

More than 50% of the world's electrical energy goes into powering the electric motors that spin up everything from simple compressors in HVAC to high-precision positioning and synchronization operations in the food and beverage industry. AC drives control the speed of these motors, so bringing more efficient AC drives to market is critical to improving energy efficiency. Danfoss A/S, one of the world's largest manufacturers of AC drives, is dramatically reducing time to market for its new AC drives using ANSYS Sherlock.

By **Amol R. Chopade**, Lead Reliability Engineer
Danfoss Drives A/S, Graasten, Denmark

Reliability Physics and FEA: A Perfect Match in the Electronics Industry
ansys.com/fea-electronics

The world turns on electric motors. They power everything from subway wheels to big turbines, from escalators to baggage carousels. With so many uses, electric motors are often required to operate at varying speeds and torque levels to suit application demand. For example, when there are few or no bags on the carousel at the airport, there is no need for the carousel motor to spin at its top-rated speed or torque and waste electricity.

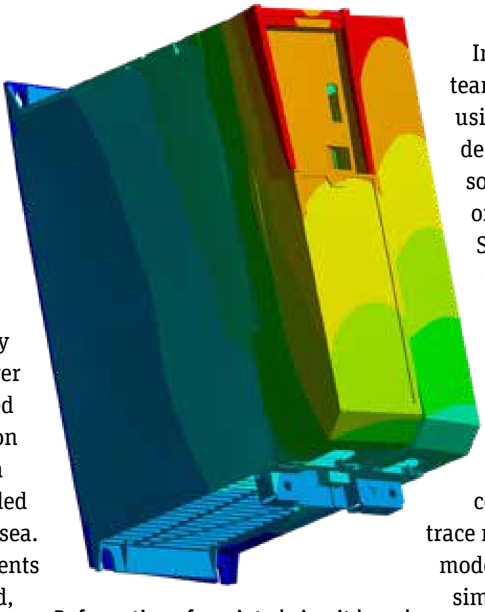
With so much of the world's electrical energy being used by electric motors, anything that more effectively regulates power to them helps reduce energy waste. That is where AC drives from Danfoss A/S come into play. AC drives — also known as adjustable or variable speed drives, variable frequency drives, frequency converters, inverters and power converters — control the speed of electrical motors. Danfoss drives play a key role in the development of smart communities by helping to deliver an uninterrupted, temperature-controlled supply chain, a fresh food supply, building comfort, clean water and environmental protection. In all, the company's line of AC drives has thousands of variants to cover the widest range of applications from a fraction of a watt to several megawatts. Engineers in its R&D facilities — in the United States, Germany, India and China, as well as in the company's headquarters in Graasten, Denmark — work constantly to bring new and more advanced AC drives to market.

The product development cycle for a new AC drive can stretch out several years. Danfoss engineers wanted to explore ways to bring new products to market faster, which led them to ANSYS Sherlock automated design analysis software.

SUBSTITUTING THE VIRTUAL FOR THE PHYSICAL

One of the great challenges in the world of AC drives is that different customers may use the same AC drive in a wide range of deployment scenarios. The same AC drive that one manufacturer uses in a line of washing machines may be used by another manufacturer in a line of radar arrays destined for use in an outdoor installation above the Arctic Circle — or in a line of bilge pumps to be installed in the engine room of a ship at sea.

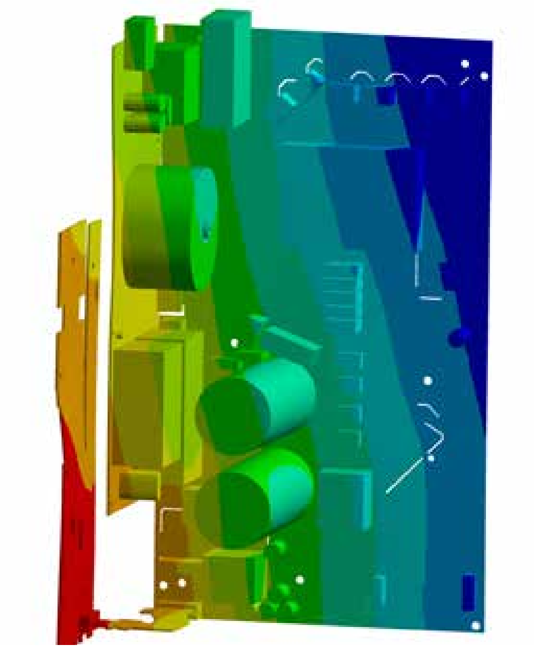
Given the range of environments where an AC drive might be used, the reliability engineers at Danfoss are always testing for drive failure due to solder fatigue arising from thermal, mechanical or physical stresses (vibration, bumps or shocks). If a prototype fails to meet reliability expectations, the team needs to identify the point (or points) of failure, refine the design, build another prototype and retest the drive — and repeat this process three, four or more times until they are confident that the new design will perform in all specified scenarios. Unfortunately, because of the time required to construct a new drive prototype for testing — even if a design modification is slight — each iteration of this design–manufacture–test cycle might take six to eight months to complete. Four design iterations of a single drive might stretch time to market by two or more years.



Deformation of a printed circuit board assembly and components inside a drive using ANSYS Mechanical. Red represents maximum displacement and blue represents minimum displacement.

In 2014, though, the company's team of reliability engineers began using ANSYS Sherlock automated design analysis software to predict solder fatigue in virtual prototypes of its new AC drive designs. With Sherlock, the reliability engineers do not need to wait for physical prototypes to conduct their tests. They simply load the electronic computer-aided design (ECAD) files into Sherlock. Within minutes, Sherlock automatically translates the file's data about components, placements and trace maps into a 3D finite element model. The software then performs simulations on that model based on thermal, mechanical and other parameters that the Danfoss engineers define. Danfoss engineers integrate these printed circuit board assembly (PCBA) finite element models into ANSYS Mechanical to perform simulation on the system level. The stress and strain results from the system-level simulation are then used in ANSYS Sherlock automated design analysis software to perform reliability risk analysis. Upon completion of the simulation, Sherlock provides the reliability engineers with a map of the components examined that shows how long each part is likely to last.

If the results provided by Sherlock indicate that the tested design is not likely to perform as reliably as the specs demand, the designers can immediately refine the virtual model. As soon as the refinement is



A vibration simulation on a complete drive assembly is shown in ANSYS Mechanical. Red represents maximum displacement and blue represents minimum displacement under vibration load.

done, the reliability engineers can reload the refined design into Sherlock and run the simulations again. While Danfoss still produces physical prototypes at different times during the design phase, the reliability engineering team does not need to wait for physical prototypes to perform its reliability risk analysis. Danfoss engineers now cycle through design iterations so quickly that they target bringing new drives to market in less than half the amount of time that it used to take — and delivering higher product reliability right from launch.

MORE INSIGHT INTO MORE APPLICATIONS

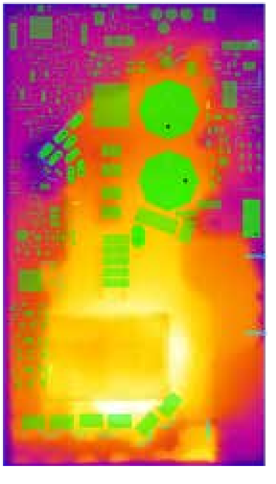
With ANSYS Sherlock, Danfoss engineers can conduct far more virtual tests than before. Instead of physically replicating the thermal variances associated with an arctic radar station or the vibrational characteristics of a spinning washing machine or a ship at sea, Danfoss engineers have developed a library of application load profiles that they can use in Sherlock to simulate these and many other scenarios. It takes only a moment to load a scenario, and most simulations take only three to four hours to complete. When testing physical prototypes, it might have taken Danfoss engineers months or years to set up and run some of these tests.

The benefit of being able to draw upon such a library of simulations is noteworthy in other ways. When Danfoss engineers worked exclusively with physical prototypes, they had neither the time nor the budget to test each prototype in all different usage scenarios. Therefore, Danfoss would occasionally find itself in a position where it had to accept returns from customers whose drives had failed when used in untested scenarios. Today, because the design team can quickly run so many different simulations in Sherlock, Danfoss is seeing far fewer drives being returned due to failure in untested situations. Overall product reliability is improved, as is customer satisfaction.

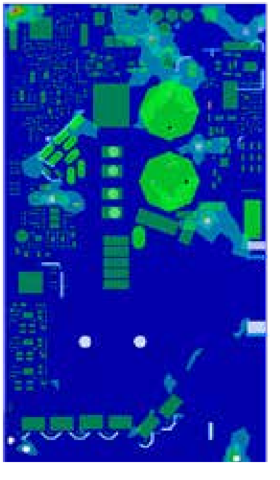
ANTICIPATING FUTURE SAVINGS

Danfoss has a long-standing commitment to responsible environmental stewardship by fostering more efficient use of electric power. By bringing new AC drives to market faster, Danfoss enables customers around the world to accelerate the uptake of these energy-saving devices. In turn, that means the wide range of products that require electric motors can operate more efficiently and waste less power. The baggage carousel does not need to draw more power if there is less luggage — or no luggage at all.

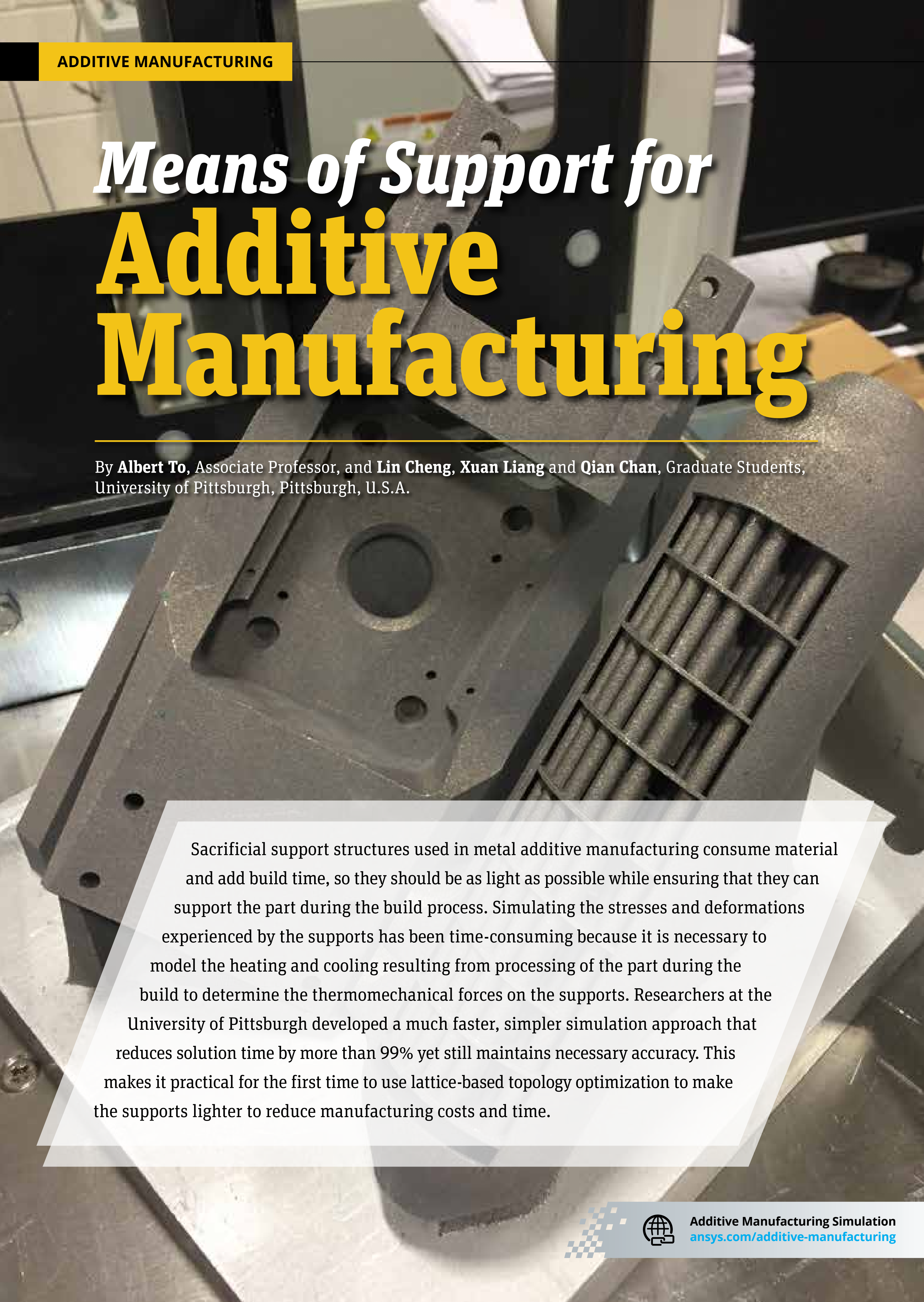
When 50% of the world's electrical power is spent powering electric motors, the opportunity for power leakage and energy waste is rampant. With ANSYS Sherlock accelerating the process, Danfoss is spinning up a new set of solutions that can help make the world a more efficient consumer of the electricity it relies on.



Temperature distribution due to power losses on the component and in the PCB tracks used for solder joint fatigue life estimation. The components in green show no risk of solder joint fatigue; those in yellow show marginal risk. Several design iterations were performed to achieve acceptable lifetime results.



ANSYS Sherlock displays strain distribution on PCBA from system simulation and solder joint or component cracking risk analysis. The components in green reveal that there is no risk, a result achieved after several design iterations.



Means of Support for Additive Manufacturing

By **Albert To**, Associate Professor, and **Lin Cheng**, **Xuan Liang** and **Qian Chan**, Graduate Students, University of Pittsburgh, Pittsburgh, U.S.A.

Sacrificial support structures used in metal additive manufacturing consume material and add build time, so they should be as light as possible while ensuring that they can support the part during the build process. Simulating the stresses and deformations experienced by the supports has been time-consuming because it is necessary to model the heating and cooling resulting from processing of the part during the build to determine the thermomechanical forces on the supports. Researchers at the University of Pittsburgh developed a much faster, simpler simulation approach that reduces solution time by more than 99% yet still maintains necessary accuracy. This makes it practical for the first time to use lattice-based topology optimization to make the supports lighter to reduce manufacturing costs and time.

Additive Manufacturing Simulation
ansys.com/additive-manufacturing

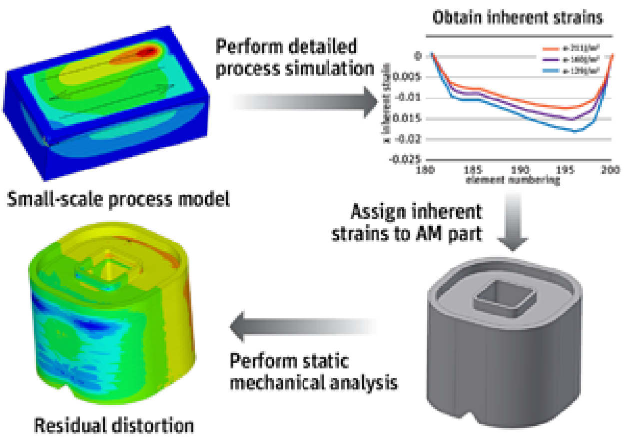
Support structures play an important role in metal additive manufacturing by bracing overhang geometry and serving as pathways to conduct heat from the part during the build. Topology optimization has tremendous potential for minimizing the mass of support structures while ensuring that internal stress does not exceed the yield stress. In the past, thermomechanical simulation of the complex metal additive manufacturing process has taken so long that it has not been practical to run the hundreds of simulation iterations required for this approach. University of Pittsburgh researchers have addressed this challenge by developing a streamlined method for simulating additive manufacturing that can be completed in under a minute, compared to hours or even days for traditional transient simulation. This new process, which enables significant cost reductions in metal additive manufacturing, will be included in an upcoming release of ANSYS Mechanical.

"This streamlined method for simulating additive manufacturing can be completed in under a minute, compared to hours or even days for traditional transient simulation."

METAL ADDITIVE MANUFACTURING PROCESS

The powder bed fusion metal additive manufacturing process produces parts by sequentially melting tiny areas of a powder bed with a moving laser. As each section of the part on the top layer cools, the solid underlying layers resist the thermal contractions, applying a tensile stress to the top layer. Likewise, the top layer applies compressive stresses to the solid area beneath it. These stresses have a significant impact on the loading of the supports.

Engineers can simulate the metal additive manufacturing process with tools such as ANSYS Additive Print and ANSYS Workbench Additive. These simulation tools predict residual stresses and deformation, and their effect on the finished part and supports. This requires a thermomechanical simulation that takes hours to compute. University of Pittsburgh researchers explored using topology optimization to start from the design space allocated for the supports and iterate to an optimized design while altering both the basic shape and dimensions of the supports to minimize their weight and cost.



Modified inherent strain method

But topology optimization typically requires running hundreds of simulation iterations in a batch, so performing the full thermomechanical simulation required to simulate metal additive manufacturing would take too long to be relevant in a real-world engineering environment.

MODIFIED INHERENT STRAIN METHOD

University of Pittsburgh researchers developed a new method that substantially reduces the time required to simulate metal additive manufacturing. The new approach, which they call the modified inherent strain method, begins with using ANSYS Mechanical to perform a detailed thermomechanical simulation of a small section of the build as it is briefly heated to a high temperature by the laser and then cools. This section, called the representative volume element (RVE), is typically several millimeters in length and width and one build layer thick. The simulation calculates the modified inherent strain, which is defined as the difference between the total strain for the initial state when the laser has just heated the RVE and the elastic strain at the final state when the RVE has cooled to room temperature. The modified inherent

© 2019 ANSYS, INC.

“This method makes it possible to use lattice-based topology optimization to iterate to a design that reduces manufacturing costs while ensuring that the support structures can reinforce the part.”

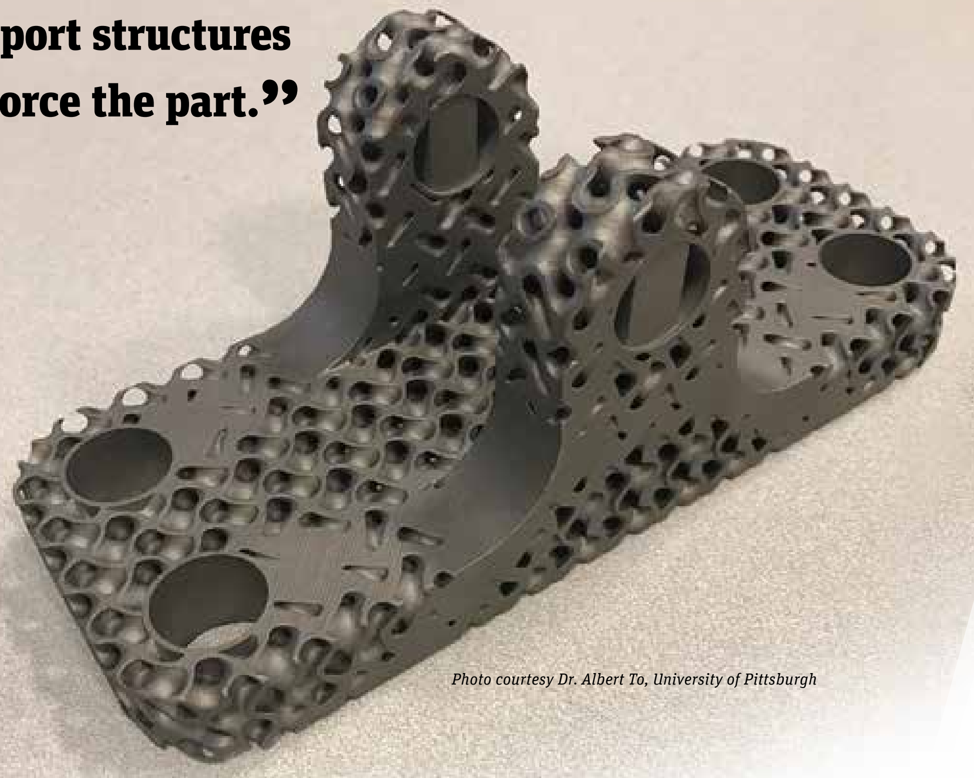


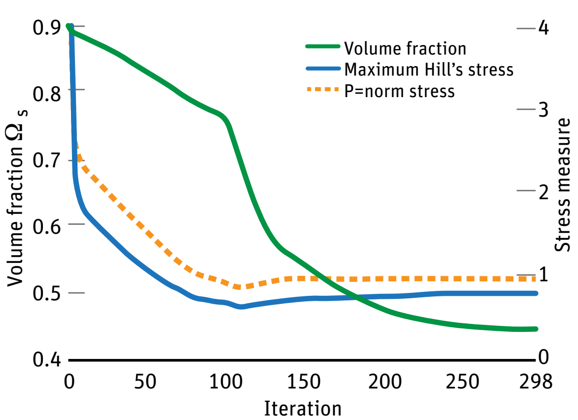
Photo courtesy Dr. Albert To, University of Pittsburgh

strain is assigned to the entire finite element model. Then, a single static mechanical equilibrium analysis, which takes a fraction of the time required for thermomechanical simulation, is performed. The results of the modified inherent strain method match the results of conventional thermomechanical simulation within a few percent.

The researchers have found that the most efficient way to optimize the support structures and, in some cases, the part itself, is to construct them with variable density lattice structures. These structures would be far too expensive to produce with traditional manufacturing but can be produced with additive manufacturing with no cost penalty. The lattice is designed by choosing the maximum bridge span that is self-supporting and determining the diameter of each section of lattice by topology optimization.

ITERATING TO A LOWER-COST SUPPORT STRUCTURE

The researchers developed a stand-alone code to set up a series of ANSYS Mechanical simulations, including the relative density for each element. The inherent strains obtained from the inherent strain method are applied to the finite element model as initial strains. The output from the finite element model is the stress and deformation of each element. Based on these results, the stand-alone code increases the relative density of elements with high stresses and deformations, and reduces the relative density of elements with low stresses and deformations.



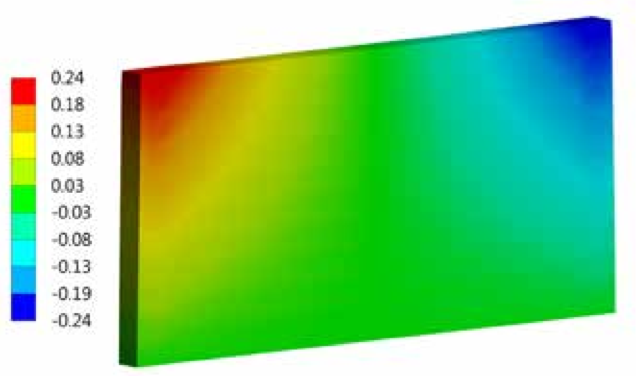
Support mass was reduced by 53% during the optimization process.

The code then executes another ANSYS Mechanical simulation. This process iterates until the support design is optimized. A complete thermomechanical simulation of the additive manufacturing process is performed to validate the optimized design.

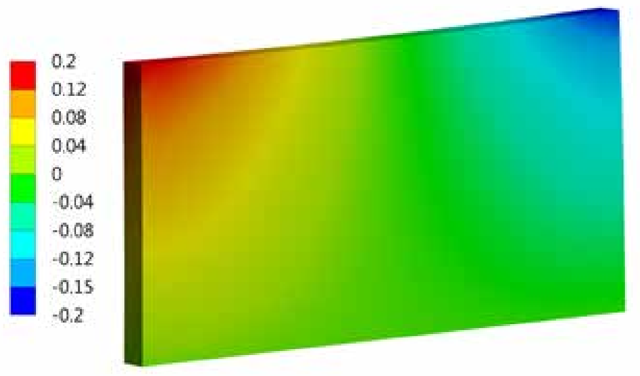
Metal additive manufacturing is experiencing rapid growth, but cost reductions are needed to spur increased adoption. A promising approach is to optimize support structures to minimize material consumption and build time subject to stress yield constraints. Performing a complete thermomechanical simulation of the additive manufacturing process provides a very accurate simulation, but the time required for a solution limits the number of design iterations that can be evaluated. The new modified inherent strain method

"The new modified inherent strain method provides accuracy close to transient simulation in much less time."

developed at the University of Pittsburgh provides accuracy close to transient simulation in much less time. This approach makes it possible to use lattice-based topology optimization to iterate to a design that reduces manufacturing costs while ensuring that the support structures can reinforce the part.



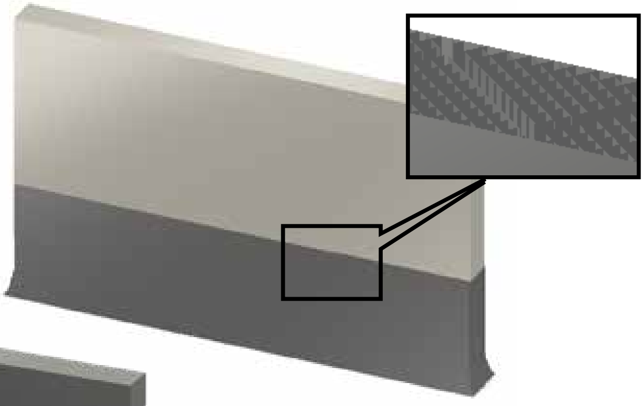
Deformation calculated with complete thermomechanical simulation in four hours



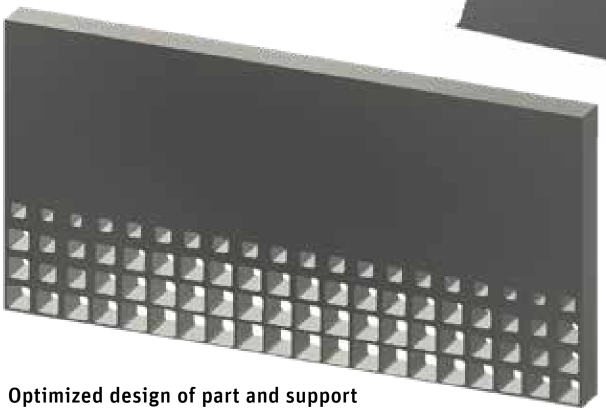
Deformation calculated with modified inherent strain method in less than one minute

Reference

Liang, X.; Chen, Q; Cheng, L.; Yang, Q; To, A.C. "A Modified Inherent Strain Method for Fast Prediction of Residual Deformation in Additive Manufacturing of Metal Parts." Proceedings of 2017 Annual International Solid Freeform Fabrication Symposium, 2017.



Initial design of part (white) and support (gray)



Optimized design of part and support

Take Simulation to the Next Level with Accurate Materials Data

Finding the right materials property data for simulation can be time-consuming and costly. Accurate simulations require accurate property data. Engineers need a reliable data source and to avoid introducing errors as data is transformed and input. These challenges are addressed with a new materials data set, embedded within ANSYS Mechanical and ANSYS Electronics Desktop, providing access to richer data and tools to connect managed corporate material intelligence.

By **Beth Harlen**, Technical Marketing Communications Specialist

Simulation can do incredible things in the world of product development. Simulation models can refine and validate products in development, ensuring that they are optimized for manufacturability, durability, sustainability and other factors that affect the product lifecycle. Assuming, that is, that analysts have access to accurate material inputs and can be assured of their pedigree — for example, through traceability back to the source test data.

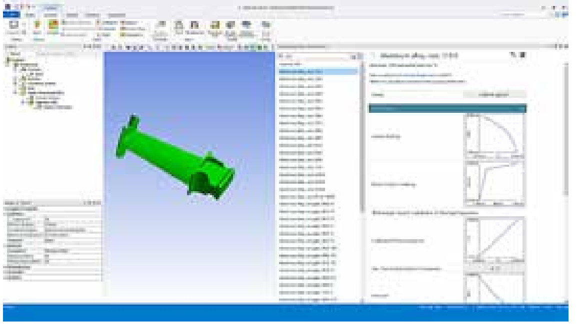
Without validated and consistent materials data, simulation is hindered by design restrictions, errors, delays and costs. Through the acquisition by ANSYS of Granta Design, a Cambridge University spinoff that provides materials information and related software, new opportunities exist for improving the accuracy of engineering simulations and analyses.

Different companies need different solutions to the problem of efficiently finding materials data. A starting point is to have a good set of reliable, easily accessed data that is valid for many simulations. "ANSYS GRANTA Materials Data for Simulation puts validated materials input data right at users' fingertips within their ANSYS simulation tools," says Stephen Warde, who heads the product management and marketing team at ANSYS Granta. "Taking this one step further — which is especially relevant for larger enterprises — ANSYS GRANTA MI helps companies ensure they are making best use of proprietary, in-house materials data along with more in-depth reference information."

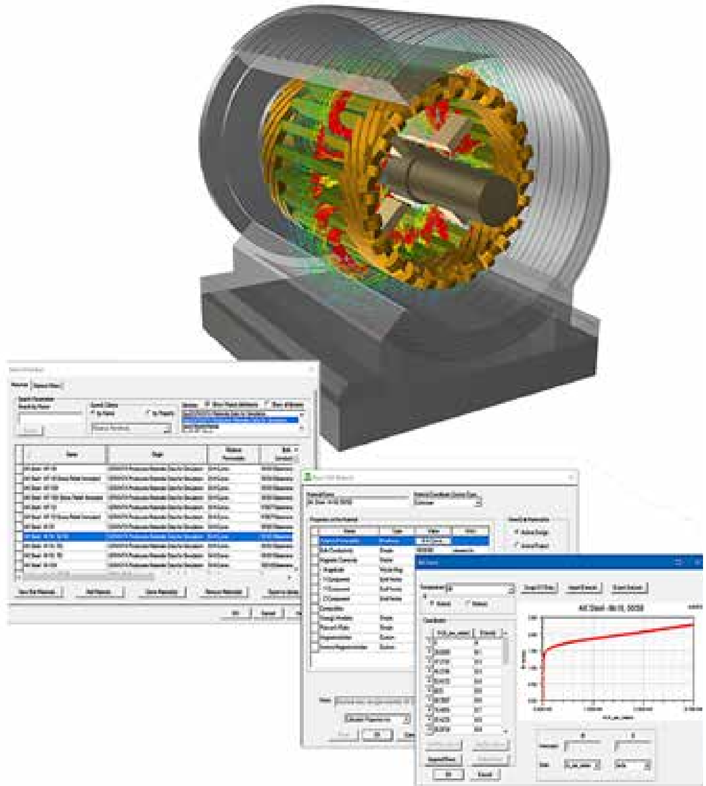
VALIDATED MATERIALS DATA AT YOUR FINGERTIPS

Materials Data for Simulation is a dataset of over 700 materials — including metals, plastics, polymers, composites, magnetic materials, ceramics and more — with properties specifically chosen to support ANSYS simulations. It supports multiphysics workflows by making the same, consistent data available through ANSYS Mechanical and ANSYS Electronics Desktop, so engineers analyzing both structural and electromagnetics issues can benefit from consistent, validated materials.

The Materials Data for Simulation dataset provides the material property data needed for structural and electromagnetic analysis. Room-temperature



A dataset of over 700 materials — including metals, plastics, polymers, composites, magnetic materials, ceramics and more — is available through ANSYS Mechanical.



Access consistent, validated materials data within the ANSYS Electronics Desktop user environment.

materials properties of the following types are available for all 700+ materials:

- Linear, isotropic elastic (Young's modulus and Poisson's ratio)
- Thermomechanical (thermal expansion coefficient)
- Thermal (thermal conductivity and specific heat capacity)

Where relevant, users will also find electrical and magnetic properties for many materials, e.g., electrical conductivity, dielectric constant, dissipation factor, magnetic coercivity and permeability, core loss and B-H curves.

The data are collated and maintained by ANSYS and are based on proven sources, including the Granta Material Universe database and the JAHM simulation data set from JAHM Software, Inc.

"Every datasheet in the GRANTA Materials Data for Simulation dataset represents a generic materials type, rather than a specific product from a materials producer," Warde says. "This means that each record furnishes representative values for the properties offered by the available grades of the material. The goal is to support the early phases of design and to provide a wide-ranging reference source that supports simulation to obtain reliable results quickly."

LEVERAGE CORPORATE MATERIALS DATA

Completing the journey toward best practice requires organizations to think about two factors. First, what wider reference data sources are available to supplement the generic data available in GRANTA Materials Data for Simulation? The ANSYS GRANTA Selector software provides access to the complete Granta library of reference data, including rich sources of grade-specific data for metals and polymers. The software provides an array of tools to analyze this data and export it for use in simulation.

Second, and more strategic, how can organizations ensure best use of their own materials data — particularly where they have in-house expertise focused on analyzing test data to generate the materials models needed for simulation? Here the ANSYS GRANTA MI software can help, Warde explains.

"GRANTA MI is a dedicated materials information management system," he says. "It enables organizations to manage corporate materials data alongside the Granta library, creating a single source for materials data. Capture all of your test data, analyze that data to generate inputs for simulation, and make that input data available via an app within ANSYS Workbench, while ensuring all of this information remains linked for full traceability."

ANSYS GRANTA MI provides an enterprise-level solution to the wider "material intelligence" challenge.

MATERIAL IMPORTANCE

Materials data is critical to the success of simulation. However, users must make a point to ensure that data is validated, consistent and fully traceable. The process must begin with the right materials information.

"If materials input data are not reliable or simply are not available, simulations will never deliver on their true value," says Warde.

Through ANSYS GRANTA Materials Data for Simulation, users can access more than 700 material records directly within ANSYS products, saving costs, minimizing risk and improving their time to market. That value can be built on with a richer source of validated reference data via GRANTA Selector. Ultimately, GRANTA MI materials information management system enables the full lifecycle of simulation-related data to be consistently and securely managed.

© 2019 ANSYS, INC.

< The hybrid part ForgeBrid® was produced by a combination of open die forging and additive manufacturing at Rosswag.

Reducing the Strain of Additive Manufacturing

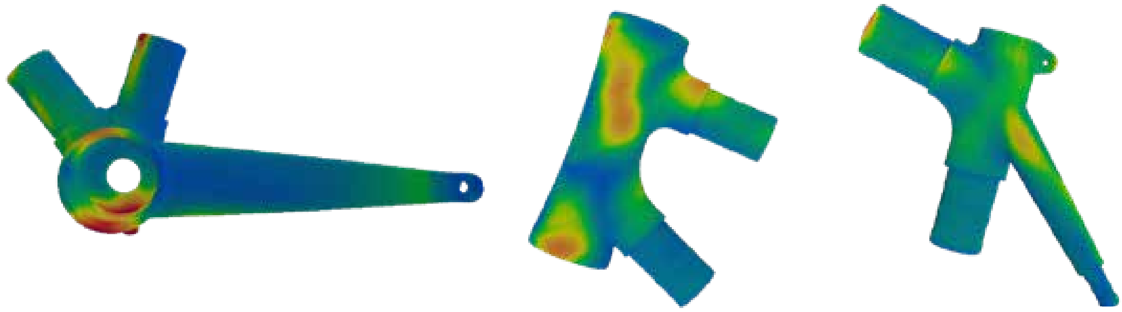
As additive manufacturing's popularity surges, reliability issues that lead to part failure still linger. Leveraging ANSYS Additive Print, Rosswag engineers determine strain prior to printing to eliminate distortion, stress and blade crashes, and reduce the number of builds.

By **Gregor Graf**, Head of Engineering, Rosswag Engineering, Pfinztal, Germany

Additive manufacturing (AM) — also known as 3D printing — is disrupting innovation and revolutionizing how global companies design and develop replacement parts, strengthen products and prototype new inventions. A global leader in metal AM services, Rosswag Engineering innovates functionally optimized metallic parts with incredibly intricate internal structures and complex shapes for numerous aircraft, energy, and oil and gas businesses.

When Rosswag Engineering launched metal AM production, a highly complex project may have required up to 10 printing iterations to produce a desired geometry. Material behavior during the printing process remained unpredictable, which necessitated

Video: Rosswag Staying Laser Focused with ANSYS Additive Print to Deliver Innovative and Complex Metal Printing Products
ansys.com/rosswag



These three different bicycle frame parts, including the saddle clamp, have high tolerance requirements. The parts must be perfectly aligned for assembly. ANSYS Additive Print helps significantly reduce distortion to fulfill all the requirements on time without the need to produce sample parts.

this trial-and-error approach. A single build (print) failure due to internal stresses and thermal distortion could cost the company thousands of dollars in development time, delay part delivery to the customer and destroy parts of the expensive AM machine.

To optimize customer part geometries, reduce print failures and shorten development time, the Rosswag team adopted ANSYS Additive Print. Integral to the company's design process, Additive Print simulates how materials will behave during the printing process. It predicts part shape distortion and stress, and designs optimal support structures for distortion compensation. Jobs are completed months earlier than the competition thanks to the software's unprecedented efficiency. The design of complex jobs can be finished in 20% less time and simple jobs accomplished in 50%–60% less time.

COMBATING STRAIN WITH PRINT PROCESS SIMULATION

To calculate and predict distortion, residual stress, blade crashes and other printing issues, the team utilizes Additive Print. By detecting these issues early through simulation, engineers can determine the appropriate modifications and redesign the part, greatly reducing the number of builds. Featuring three strain detection modes with increasing levels of fidelity, Additive Print simulations are produced within a reasonable timeframe.

ANSYS ADDITIVE PREP

ANSYS Additive Prep simplifies the process for orienting parts and developing advanced support structures without sacrificing machine time or print materials.

Once the part's optimal build orientation has been selected, the software detects the critical regions that require support and inserts the type of supports needed. Within seconds, it auto-generates a complex set of support geometries to reinforce the part during the printing process.

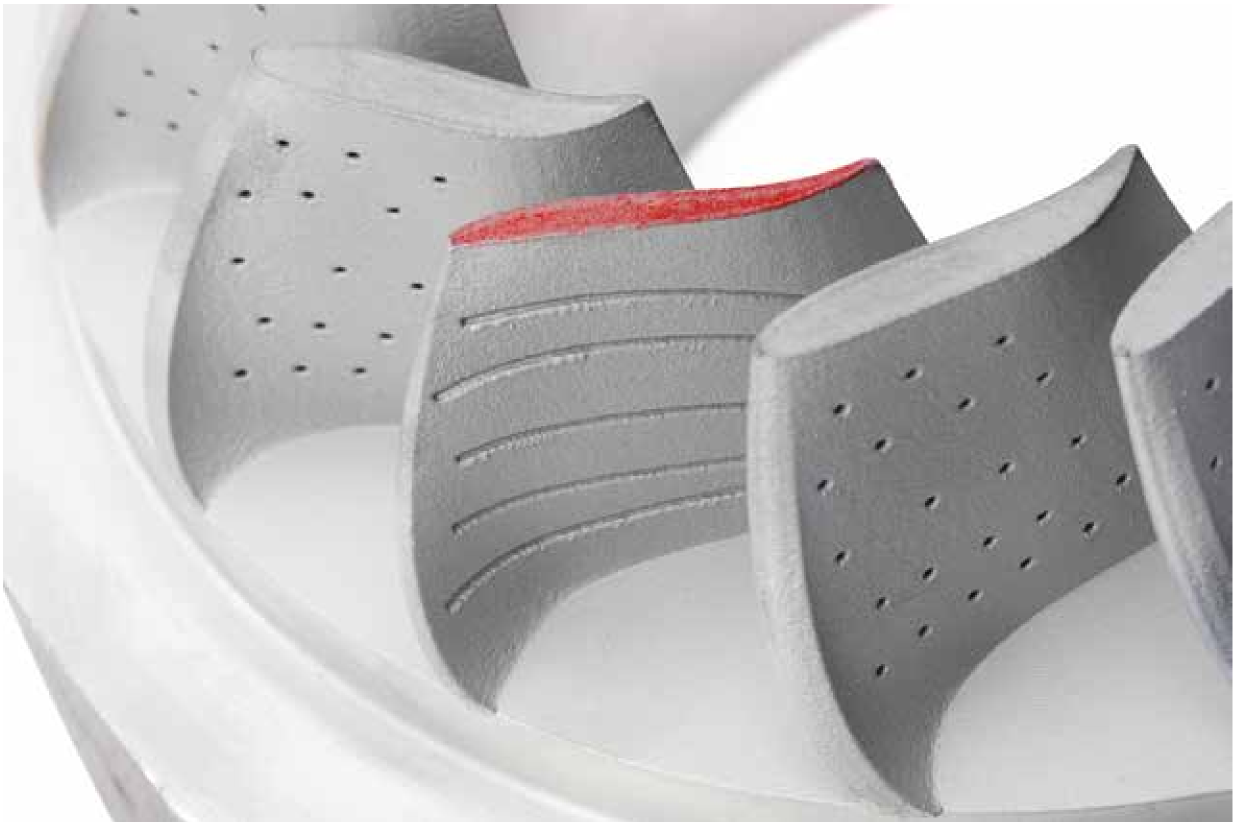
Simulation generates heat maps that identify potential strains and reveal how build orientations will impact support structures and build times.

The team uses assumed uniform strain capabilities to analyze parts in an additive fashion, layer by layer, similar to how the printer builds up layers of material. This simulation provides fast turnaround and a good understanding of the pattern of the displacement that the part will experience.

Scan pattern strain simulation calculates the effects of the laser scan direction on every powder layer and provides the team with a bit more fidelity

“Leveraging ANSYS Additive Print, the team reduces spending, increases reliability and delivers products to customers faster than ever before.”

© 2019 ANSYS, INC.



The patented production process for adding highly complex blade structures with internal channel structures on conventional manufactured parts could be the next game changer for the aerospace and energy machinery industry.

and refinement. This simulation accurately evaluates the elemental difference in the strain within the melt pool based on the direction of the laser's movement.

Using thermal strain simulation to perform a thermal–mechanical analysis, Rosswag designers calculate the heating and cooling of virtually every point within every scan vector of the part. With high-fidelity simulation — up to 15 µm resolution — this capability detects very fine features and differences in the strain, revealing part deformations with high accuracy.

Simulation also helps predict and identify areas on the part that have the highest potential for causing printing accidents, such as blade crashing. This occurs when a part being built lacks sufficient support structures and tilts up to collide with the recoater blade, damaging the part and potentially the machine. Adding support structures offsets thermal strain challenges to stabilize the part, preventing it from curling like a potato chip.

Based on the predictions of these three modes, engineers determine the part's optimal geometry compensation. Should distortion be detected, the team employs the software's automatic distortion compensation tool, which takes the desired geometry and reverse distorts it so that it reverts to the correct shape during the printing process.

GENERATING SUPPORT STRUCTURES TO PREVENT STRAIN

If simulation reveals a strain, the designers use this data to identify points in the geometry where a change in the structure will help mitigate the

Additive manufactured parts with specimens for mechanical testing on a build plate are shown here.

strain. Engineers generate supports using traditional geometry-based algorithms or use Additive Print's automatic generation of physics-based supports to print the part in its desired orientation.

In the next round of simulation, the software reveals the support structures' effectiveness and determines whether more stability may be required to confirm that the part will be built without unexpected distortions. Alternatively, the part's design may change to circumvent possible deformation. Prior to printing, the team will notify the customer of the proposed part geometry modifications and obtain approval.

Next, the Rosswag team prints the part, incorporating supports if needed. Depending on the part's geometry, if there are critical or overhanging edges, the support structure will be used for heat transfer and to reinforce the layer that needs it. Following the print, designers verify the part's geometry by scanning and measuring it with a 3D laser scanner to certify that the part matches the geometry the customer approved.

> **"Using Additive Print for simple parts, the company saves about four build jobs per month. This streamlines Rosswag's operation and saves 100,000 euros (about $112,000) per year."**

STREAMLINING THE WORKFLOW TO SAVE TIME AND MONEY

Additive Print remains integral to increasing the reliability of results, which can be reflected by the low number of printings needed to optimize parts. For example, if the team completes the job in two printings rather than four or five, the reliability of the production process is higher.

Using Additive Print for simple parts, the company saves about four build jobs per month. This streamlines Rosswag's operation and saves 100,000 euros (about $112,000) per year.

For complex parts, the simulation software saves the company 30,000 euros (about $33,000) per project, which equates to 10 printing jobs. So, as each job takes two to three days to complete, this saves 20 to 30 days in printing time.

USING ADDITIVE PRINT TO BUILD A BICYCLE

The Rosswag team first used Additive Print to assist a local college student who needed to fabricate a bicycle for a design competition. He created a sophisticated CAD design, and Rosswag helped him further improve his frame design.

The team challenged themselves to develop AM parts and connect them with parts not produced by AM, which were made of carbon fiber and composed one-third of the frame. This presented a design issue as simulation revealed deformation that prevented parts from being connected. Simulation also revealed minor deformations (cosmetic issues) on some of the outer surfaces.

To address the frame deformations, engineers modified the bike's geometry without adding support structures. Additionally, they adjusted the geometry of its outer surfaces for a polished appearance.

To print the parts, the team used its SLM 280HL selective laser melting system. After printing, Rosswag used a 3D scanning system to confirm that the printed part matched its geometry specifications.

Leveraging ANSYS Additive Print to eliminate strain from their printing process has taken Rosswag's AM abilities to the next level. The company significantly reduces spending, increases reliability and delivers products to customers faster than ever before.

Gliding Farther and Faster

A 3D rendering of the new AS 33 sailplane is shown gliding over Wasserkuppe Mountain.

To reduce the drag on the wings of a sailplane (glider) so it could go faster and farther, engineers needed to shave a small amount of surface area from the wing. This complex task involved fluid, structural and composite material challenges that had to be solved in parallel, and which could only be done using engineering simulation.

By **Ulrich Simon**, Alexander Schleicher Segelflugzeugbau, Poppenhausen, Germany

While most people consider a sailplane to be more dangerous than a motor-powered aircraft, gliding aficionados say it is safer because there is no engine that can fail and cause a disaster. A skilled glider pilot can currently ride the "thermals" — the updrafts of warm air that keep the aircraft aloft — for up to 1,000 km over a 10-hour flight. But increasing the speed and range requires reducing the overall drag on the glider. Engineers at Alexander Schleicher Segelflugzeugbau (AS Sailplanes) used ANSYS fluids, structural and composites simulation software to design a wing with a new composite structure and with a smaller surface area to reduce the drag of the AS 33 aircraft.

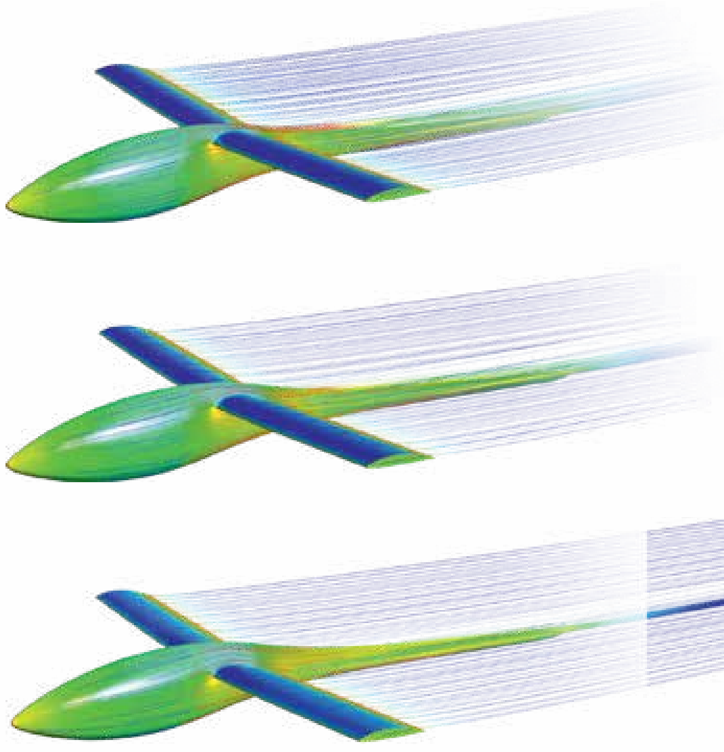
"Without ANSYS CFD, they would not have been able to settle the long-standing argument about the optimal attachment location of the wing to the fuselage."

AERODYNAMIC, STRUCTURAL AND MATERIALS CHALLENGES

Best-in-class gliders used in competitions weigh from 400 kg to 600 kg. They have an 18 m wingspan and a 10.5 m^2 wing surface area, with a wing thickness of only 10 cm — about 4 inches. While the wing span and thickness are very close to practical limits, AS Sailplane engineers believed the surface area had some room for optimization. Reducing the surface area, even by a small amount, can cut the aerodynamic drag significantly. So, they decided to reduce the surface area from 10.5 m^2 to 10 m^2. By keeping the 18m wingspan constant, they produced a wing with a smaller average chord (front-to-back width) and therefore a reduced wing thickness.
While that may not sound like a big change, it is massive in an aircraft that has been around for a long time and is already close to its optimal design. This required the team to overcome a host of challenges.

Reducing the wing surface area produces less lift so the aerodynamics of the system must be improved to compensate. A smaller wing also has less space for structural elements, so engineers needed to improve the design so that the wing can carry the same loads while maintaining strength. Engineers also wondered whether connecting the wing high up on the fuselage was better than a mid-fuselage join in terms of strength and drag. An additional challenge was to optimize the winglets — the small, upturned tips of the wings — which reduce vortex airflow at the ends of the wing, further reducing drag.

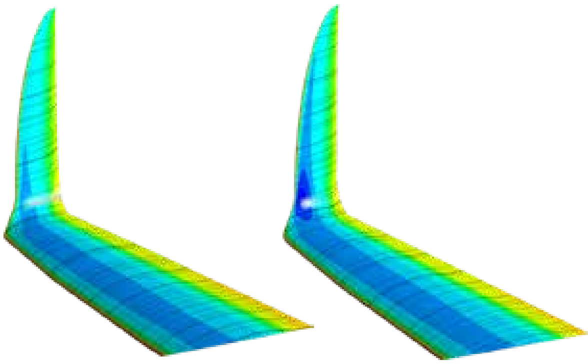
Because most of the sailplane is made of composite materials, except for the metal landing gear and mechanical control system, engineers explored an all–carbon-fiber design instead of the commonly used combination of glass fibers and carbon fibers embedded in a polymer matrix.



Configuration investigated for the wing–fuselage junction including a high-wing position (top) to a mid-wing position (bottom). Calculations revealed minimum drag for the mid-wing position, especially at high airspeeds.

ANSYS Fluent proved to be crucial for aerodynamics calculations, ANSYS Mechanical for structural considerations, and ANSYS Composite PrepPost

© 2019 ANSYS, INC.



ANSYS Fluent CFD revealed unfavorable pressure peaks in the area of the wing–winglet junction, which was originally designed with classical aerodynamic tools. After design iteration with Fluent, it was possible to alleviate this problem and gain some aerodynamic efficiency.

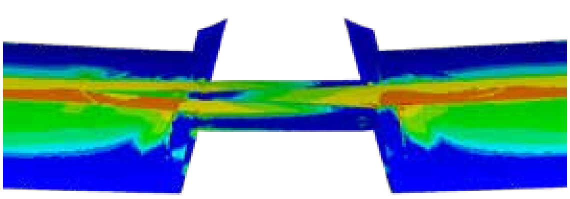
for analyzing the stresses and strains by means of composite failure criteria on the new material.

USING SIMULATION TO OVERCOME THE CHALLENGES

AS Sailplane engineers started the project with an in-house analytical tool that the company had been using for several years, but they soon realized that this tool would be insufficient for the task. Contracting the work out to universities proved to be too expensive, inflexible and not conducive to building in-house expertise. One of the engineers had used ANSYS simulation solutions earlier in his career and suggested that ANSYS finite element analysis solutions would be up to the challenge. A particularly compelling argument was that ANSYS had computational fluid dynamics solutions in Fluent, structural solutions in Mechanical and composite solutions in Composite PrepPost, so only one software supplier was required.

ANSWERS FOR AERODYNAMICS

First, the engineers applied ANSYS Fluent to solve a question that had been puzzling sailplane engineers for years: What is the best place to connect the wing to the fuselage for the least drag? Some had argued that attaching the wing at a high position on the fuselage prevents detachment of the boundary layer airflow from the aircraft body, leading to less drag; others contended that attaching the wing to the middle of the fuselage requires a smaller connection cross section, thus reducing drag, while



Safety factors for the carbon fiber–reinforced plastics structure in the area of inner-wing junction were analyzed with ANSYS Composite PrepPost.

simultaneously increasing the undesirable boundary flow detachment phenomenon. The discussion had persisted for years because there was no way to answer the question definitively using wind tunnel testing. Standard calculations were also insufficient, so simulation was needed.

After analyzing six wing–fuselage connection positions using Fluent 3D CFD calculations, the engineers determined that the mid-wing configuration produced less drag, especially in high-speed cornering situations, so they settled on the mid-wing–fuselage connection point for the AS 33. The detachment of the boundary layer airflow proved to be a smaller factor than previously thought. Without CFD, they would not have been able to settle this long-standing argument. A similar simulation process was used to determine the optimal positioning of the winglets.

STRUCTURES AND MATERIALS

The smaller wing of the new AS 33 sailplane must support equivalent or even higher loads than its predecessors because the weight of the fuselage of the AS 33 was increased to accommodate an enhanced, crash-optimized cockpit, and pilots now carry more electronic equipment with them. Supporting higher loads with smaller wings requires structural improvements. Engineers decided to use an all–carbon-fiber composite material instead of the carbon and glass fiber mix used previously. Carbon fibers are stronger than glass ones and can support higher loads. AS Sailplane engineers realized that they needed finite element analysis and composite simulations available to prove that this new composite construction would be strong enough for the task at hand.

Because of the needle-like proportions of the fibers, the strength of the resulting composite has directional properties based on the fiber orientation and layered plies of the composite. This adds complexity to the design process. AS Sailplane engineers decided to solve this problem using ANSYS Composite PrepPost simulation with its “model as you build it” approach. Composite PrepPost was used to model the individual composite plies and get a close-up, detailed view of the resulting layup. The engineers explored five different composite models in the entire assembly, with the total number of plies ranging from 100 to 300. They then analyzed the structural integrity using composite failure criteria and any problems with the structure. With the combination of very high loads, a small wing and little space for support structure, it was necessary to calculate the stresses and strains in the materials accurately for each ply using Composite PrepPost. Most other analytical tools cannot calculate the stresses precisely and do not reveal the critical stress peaks caused by composite failure modes that could possibly destroy the wing. Engineers then

"ANSYS Composite PrepPost enabled them to look at a full 3D stress state along the wing to detect any potential problem areas."

iteratively modified their simulation models by adding more plies or changing already existing ply properties such as ply extent, fiber orientation or materials at high load points to eliminate any weak points in the design.

They also used ANSYS Composite PrepPost to design the spar that runs the length of the wing on the interior and withstands the bending forces to which the wing is subjected. The spar is made of the same composite material as the wings and fuselage. To create a robust structure, engineers experimented virtually using the ply-wise modeling capabilities of Composite PrepPost, benefiting from the easy-to-use solid modeling features to evaluate this difficult structural part. With Composite PrepPost, they could study the 3D stress state along the wing and detect any potential problem areas. Finally, engineers performed modal analyses to prevent vibrations that lead to wing fluttering in flight — a dangerous situation.

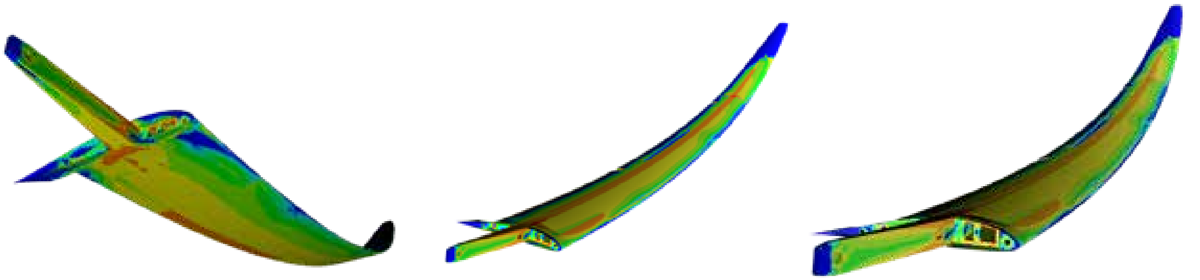
In the largest model used to simulate the vibration modes of the entire glider, engineers produced a mesh consisting of 1.5 million cells, which were mostly quadratic elements. The automated meshing process took about 15 minutes. Using two computer cores, the simulations lasted 20 minutes for a linear solution and 10 hours for a nonlinear solution. They typically performed 10 to 50 simulation runs to optimize the design, detect problems and calculate multiple load cases.

VALIDATION

Ultimately, AS engineers tested the wing structure for the European Aviation Safety agency to gain approval for its use. ANSYS simulations played a major role in demonstrating the structural integrity of the new design that led to the agency's approval.

It would not have been possible for AS engineers to reduce the sailplane's wing surface area by 4.7% (to 10 m^2) and pass this test without ANSYS simulation solutions. The ability to perform numerical calculations to see the high stress areas of the model made this innovation in sailplane design possible.

Some information for this article was supplied by CADFEM Journal.



Safety factors for the composite wing were analyzed with ANSYS Composite PrepPost. The center of the wing's chord has the lowest safety factors (orange color) because of the compression-loaded spar flange in this area.

© 2019 ANSYS, INC.

The Road Ahead: *Simulating Scooter and Motorcycle Design*

The Moto Guzzi V85 is equipped with an 80 HP twin cylinder engine.

As a leading manufacturer of two- and three-wheeled vehicles, the Piaggio Group is constantly improving safety and customer satisfaction. Engineers routinely use software like ANSYS Mechanical to optimize engine design. Now, the engineering team is evaluating ANSYS Motion, a multibody dynamic solution that incorporates rigid and flexible solvers.

By **Riccardo Testi**, CAE Analyst, Development and Strategies, Piaggio Group, Pisa, Italy

In the world of urban mobility, the Piaggio Group is an Italian icon. Europe's largest manufacturer of motorcycles, mopeds and scooters, its stylish brands include Vespa, Moto Guzzi, Aprilia, Gilera and Derbi. The company's legacy goes back more than a hundred years, but its approach to engine development is strictly modern: Engineers depend on simulation software to optimize design, ensure rider safety and satisfaction, and help Piaggio compete against Japan's market dominance.

It was not always that way, however. With a long history of successful product launches behind them, Piaggio engineers were reluctant to part with their tried-and-true — yet expensive and

> **“Engineers found ANSYS Motion’s features to be more efficient, and they appreciated its tight coupling with the powerful set of FEM features that ANSYS provides.”**

time-consuming — approach of building prototypes and making adjustments until the design met performance standards. Accepting computer-assisted techniques was a slow and wary process. It was not until the end of the last century that they developed enough trust in finite element analysis (FEA) simulation to try it.

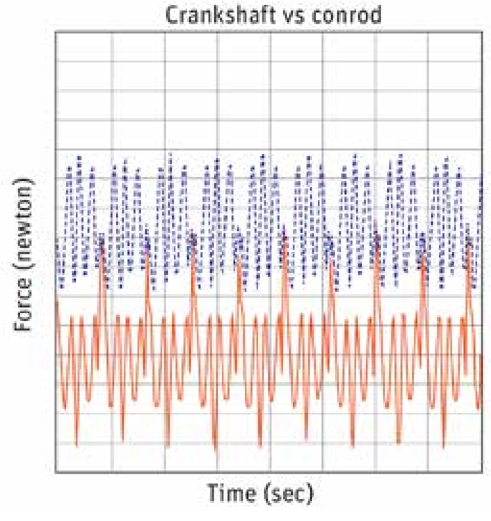
However, as the product portfolio changed over the past decade and engineers sought to better predict the behavior of new components, the adoption of simulation software accelerated. ANSYS structural solutions began to play an increasingly important role in defining component design, analyzing prototype failures and ensuring engine performance.

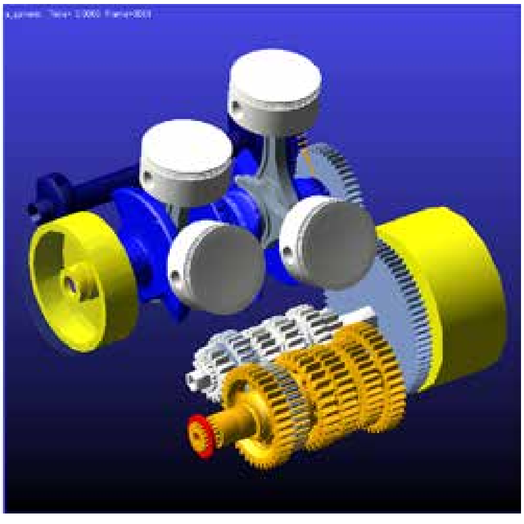
FROM TROUBLESHOOTER TO KEY TECHNOLOGY AND BEYOND

In the beginning, ANSYS software served as a troubleshooting tool that helped with Piaggio’s engine prototyping. Since then, engineers have used it to simulate complex phenomena such as nonlinear frictional load analysis for a crankshaft, making it the key technology used in the development of the company’s best-selling engines. In fact, engineers use ANSYS solutions more than any other simulation products to build virtual prototypes and overcome design challenges. By eliminating the need for multiple physical prototypes, new product development is faster and more cost-efficient.

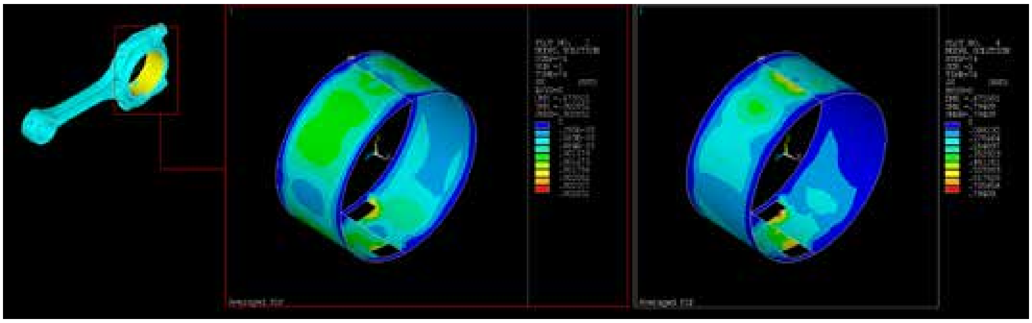
The ANSYS Workbench environment is used across all product lines, from small mopeds to large, powerful motorbikes, for several reasons: Its template-based structure greatly helps in setting up full modeling procedures, it seamlessly integrates with multiple CAE tools, and engineers appreciate its graphical user interface. In addition, they report that Workbench has reduced simulation time by 50% to 70% compared with other software. Currently, engineers are relying on Workbench to design what will be Piaggio’s most powerful four-stroke engine.

Looking even further down the road, Piaggio is evaluating ANSYS Motion multibody dynamics (MBD) software and providing test cases that have shown very promising results.





ANSYS Workbench fretting fatigue study helped determine component wear damage by simulating the exchange of forces between the conrod and the crankshaft while the engine is running.



Engineers customized ANSYS Workbench to assess whether a metallic body will break when it is subjected to time-varying, erosive forces.

THE NEED FOR SIMULATION

Whether the engine capacity is a relatively low 50 cc or a much higher 500 cc, the forces the components exert influence the overall behavior of the vehicle. Many of those forces are difficult to replicate in a lab setting, making simulation the only option.

For example, as Piaggio works to increase market share in the Far East, where bumpy roads are the rule rather than the exception, there is no other way to test vehicle performance than on a virtual landscape. It would never be practical or cost-effective to ship prototypes back and forth or to build a sufficiently bumpy test track. Consider the task of proving suspension reliability. In the past, a rider had to jump the bike off an incline and hope for a soft landing, an undertaking so dangerous that the practice was eventually abolished. Simulation eliminates risks while providing critical insight and information. Simulations can also take into account environmental variables such as wind gusts or downpours that can affect rider safety, giving drivers more control in most weather conditions.

“The company’s legacy goes back more than a hundred years, but its approach to engine development is strictly modern.”

INTEGRATING SOFTWARE FOR POWERFUL PROBLEM-SOLVING

Of course, bad roads and heavy rains are hardly the only forces engineers must contend with. As an example, loading conditions can mean life or death for a component but are difficult to measure — historically, even the finite element method (FEM) has its shortcomings when computing load. The new ANSYS Motion solution can make a difference. With rigid and flexible solvers in a single MBD solution that integrates with other ANSYS products, it helps engineers overcome some of the most significant challenges to safe and enjoyable vehicle operation.

As a theoretical example, consider a motorcycle running at a high speed on a windy day. To predict safety, engineers would perform multiphysics simulations: create a full dynamic model of the bike in ANSYS Motion, use ANSYS CFD tools to simulate the action of the wind and then integrate the resulting simulation data into the model. Engineers could evaluate all the physical conditions (fluid dynamic, structural and multiphysics) acting on the bike’s critical components in ANSYS Motion and assess its stability in the wind. After using the loading data to assess the components’ structural integrity, engineers could streamline the workflow using an ANSYS Workbench project to get results and create a procedural template for future analyses.

While that is a just an illustration, a comparison between a competitor’s results and ANSYS Motion’s provided a real-world view of an investigation into motorcycle noise.

ANSYS Motion
ansys.com/motion

TEST CASE PROVES ANSYS MOTION MERITS

Handling, response and acceleration are all part of the appeal of motorbikes and other two-wheeled vehicles. Although it may seem surprising, for many riders, noise is too. Whether it's the sound of a revving engine signaling strength and performance or the nearly undiscernible hum of an electric powertrain, noise is integral to the buying decision and the riding experience.

But not all noises are desirable. When the Piaggio R&D team discovered an unusual acoustic emission during quick clutch engagement and disengagement, they knew something was not right. Pinpointing the cause, which they believed was related to the way pins and slots engaged in the gears, required simulating the vehicle's translational inertia. To do that, they modeled a virtual engine using a well-known MBD product.

Engineers began their inquiry by recording sound pressure peaks and torque transmissions during clutch cycles. Next, they integrated experimental boundary conditions into the model: low engine torque, low engine speed, negligible vehicle acceleration and quick clutch releases. After singling out flexible elements such as the shaft, tires, wheel hub dampers and clutch dampers, they created an auxiliary FEM model with contact between the gears. The simulation supported their hypothesis that the problem was in the gear box.

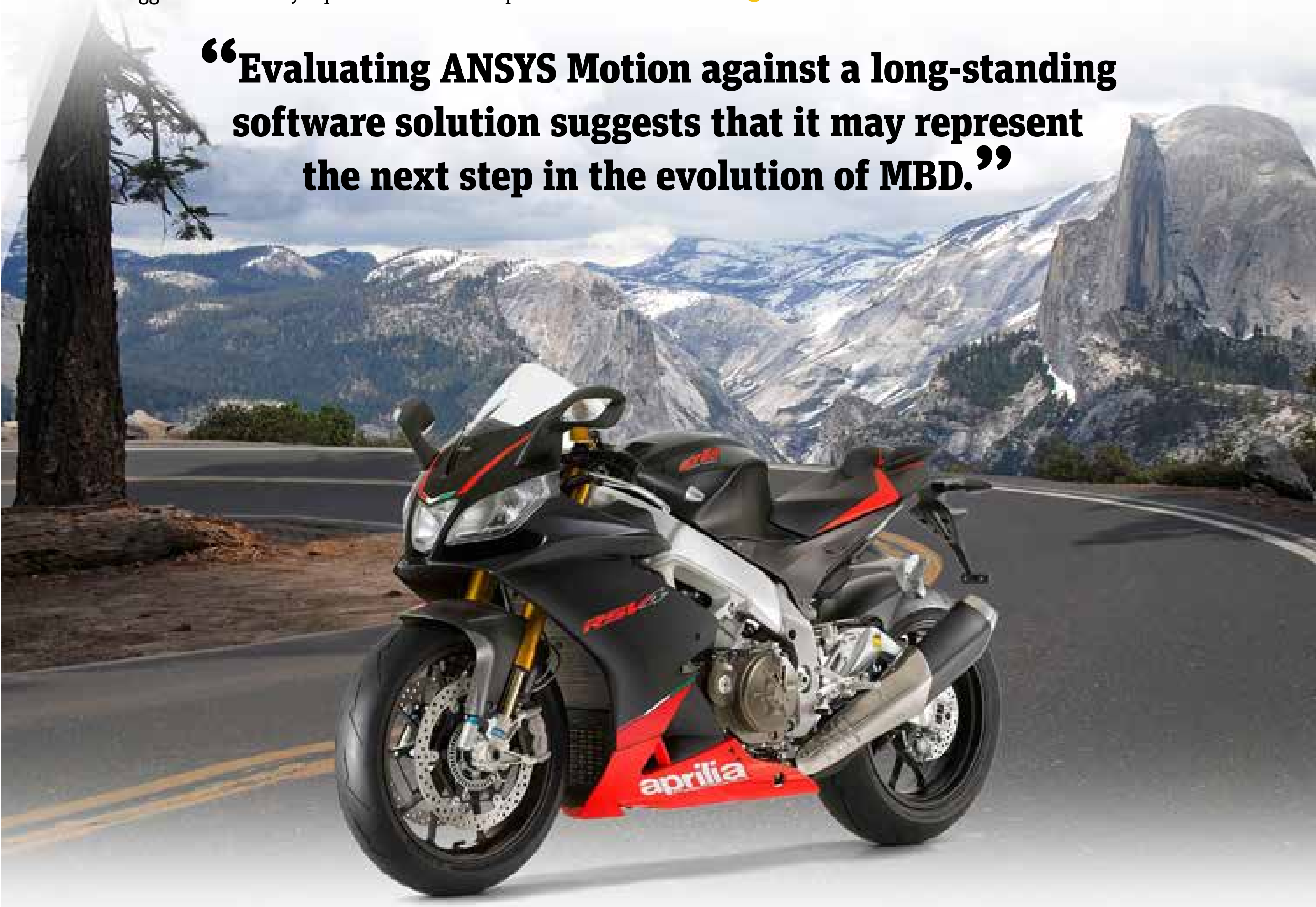
The next step was determining what to do about it. Options included reducing the angle of the gear slot, inserting a cam coupler or altering the shaft design. By simulating various alternatives that took into consideration the coupler's behavior during the clutch engagement/disengagement cycle, engineers learned that reducing the slot's angle reduced noise.

The results from the competitor's product and ANSYS Motion matched: ANSYS Motion revealed the same problem and the same solution. But that is where the similarities ended: Engineers found ANSYS Motion's features to be more efficient, and they appreciated its tight coupling with the set of FEM features that ANSYS provides.

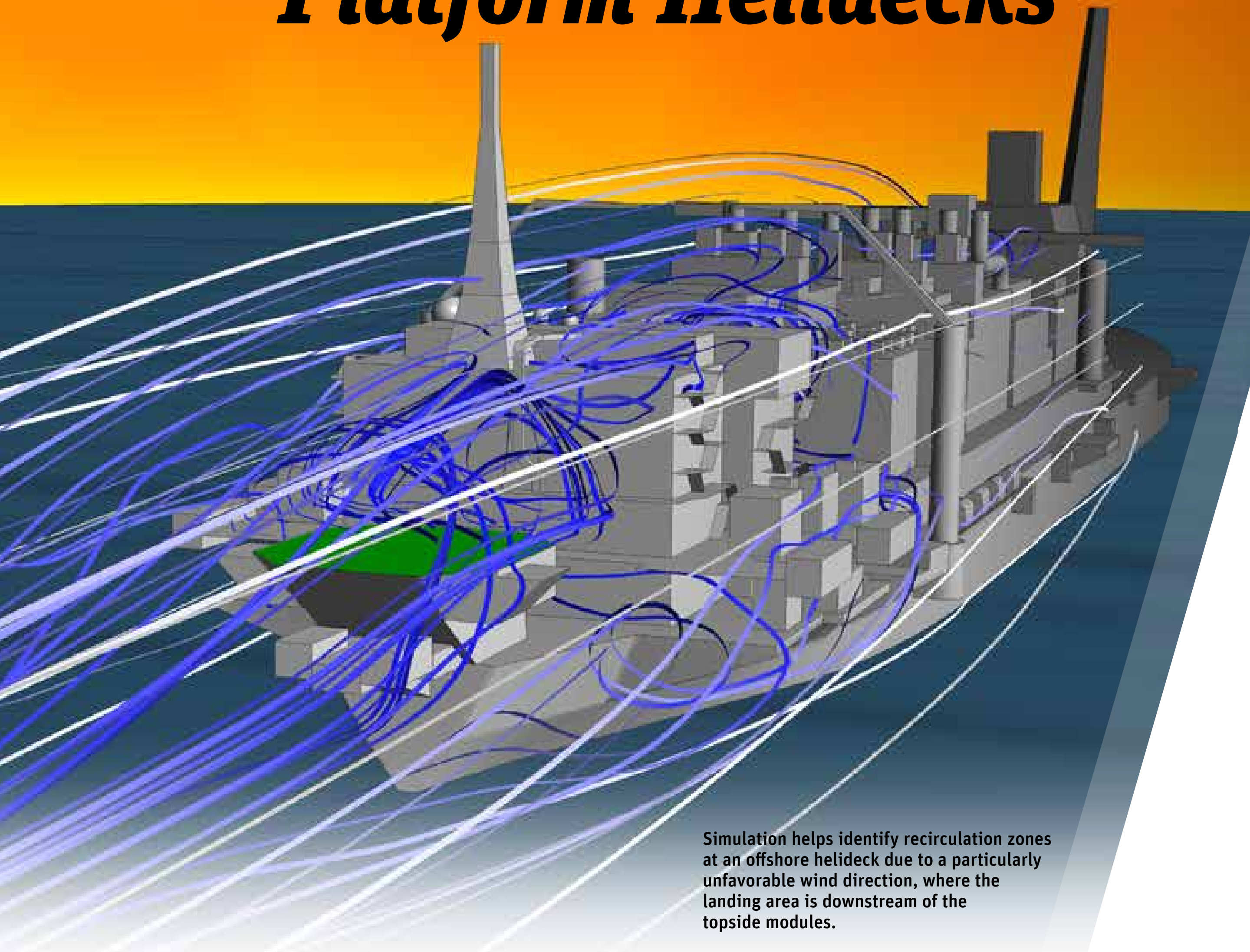
THE EVOLUTION CONTINUES

Product development is a journey: Nothing springs fully formed from the mind of an engineer. The tools that engineers use are also ever-changing and improving. Evaluating ANSYS Motion against a long-standing software solution suggests that it may represent the next step in the evolution of MBD.

"Evaluating ANSYS Motion against a long-standing software solution suggests that it may represent the next step in the evolution of MBD."



Any Way the Wind Blows: *Optimizing Offshore Platform Helidecks*



Simulation helps identify recirculation zones at an offshore helideck due to a particularly unfavorable wind direction, where the landing area is downstream of the topside modules.

Helicopters are the most common method of transporting personnel to offshore oil and gas installations. To ensure pilot and passenger safety, it is essential to understand how airspace conditions affect takeoff and landing on helidecks. Brazilian multinational Petrobras uses ANSYS CFD to model wind flow, turbulence and other conditions to optimize helideck design and positioning.

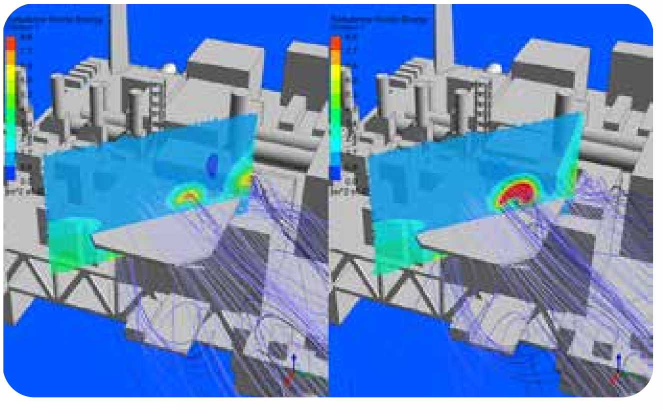
By **Daniel Fonseca de Carvalho e Silva**
Engineer, Petroleo Brasileiro SA (Petrobras)
Rio de Janeiro, Brazil

Ride the Wave
ansys.com/wave

> **"Using ANSYS CFD, engineers were able to define the limits of environmental and operational conditions for helicopter transportation."**

As oil and gas exploration moved offshore, the industry needed an efficient, cost-effective way to exploit, process and store product at sea. The result was the floating production storage and offloading (FPSO) unit, a ship-shaped facility that can be moored hundreds of kilometers from land in water depths up to 2,900 m.

Helicopters are considered the safest and fastest way to get FPSO crew members to work. However, local air flow conditions around FPSO helidecks can make maneuvering helicopters challenging and prevent on-time arrivals and departures.



Before and after modeling of additional turbulence near the helideck created by topside modifications, such as new equipment

To optimize the helideck position in new and existing FPSO installations, Brazilian multinational petroleum company Petrobras regularly simulates air velocity, temperature and gas plumes using ANSYS CFD software. Models are based on each of the company's offshore platforms, typically 150 kilometers from shore.

Simulation enables engineers to accomplish several goals:

- Identify where to locate helidecks on new platforms.
- Ensure greatest availability.
- Safely enlarge operational requirements for offshore units — for example, to fly when wind speeds are higher or with heavier helicopter payloads — which can ultimately reduce transportation costs.

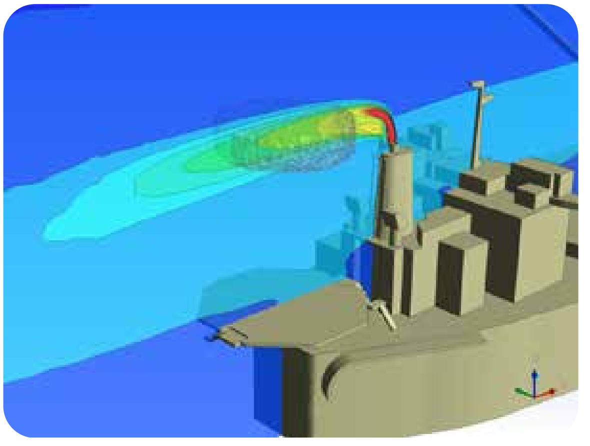
USING SIMULATIONS TO UNDERSTAND FLIGHT CONDITIONS

It takes about an hour for a helicopter to fly the 150 kilometers from the mainland to Campos Basin, where Petrobras operates more than 50 offshore units. Considering that there are 80 flights on average per day and approximately 700,000 passengers make the trip each year, it may seem routine.

However, flight conditions, including the air flow surrounding the helidecks, can vary considerably day-to-day, complicating helicopter maneuvering and influencing platform design. With Petrobras adding new offshore installations even farther from land, engineers are going to great lengths to understand how different airspace variables affect helicopter takeoff and landing.

In accordance with the international CAP437 standard, Petrobras looks at the criteria platform designers are required to address: the effect of turbulence and of hot gas plumes on takeoff and landing operations.

It is not just gusty storms at sea that cause turbulence around offshore platforms. Topside facilities represent obstacles from an air flow perspective and, depending upon wind intensity and direction, may create local turbulence as the wind is forced to flow around, over and between them. Increasing the height of the helideck decreases exposure to turbulence but puts pilots in closer proximity to hot gas plumes from the FPSO's onboard process plant. Thermal-induced plumes from turbogeneration equipment, which can emit gas exhaust as hot as 500 C, disturb air flows and increase the ambient temperature; even a change in air temperature as small as 2 C has to be considered in terms of loss of lift, engine power drop or engine failure in a helicopter — any of which could be disastrous if critical levels are reached. This makes it challenging to pinpoint the optimal landing location and may force pilots to follow a strict approach path that can be difficult to accomplish in bad weather. Only high-quality CFD simulations can provide engineers with the velocity, turbulence and temperature fields required to analyze the helideck airspace and verify wind tunnel measurements. Although engineers could rely solely on wind tunnel experiments, simulations are less expensive and faster, especially considering the time it takes to prepare a reduced-scale model for a wind tunnel. Simulation also has an advantage over wind tunnels when managing spatial resolution and scale effects. And it is significantly easier to simulate hot gas plumes and measure temperature dissipation than it is to use a wind tunnel experiment that requires special gases to determine the relationship between gas and temperature dissipation. In addition, wind tunnel experiments cannot provide comprehensive results for elaboration of a temperature gradient matrix (TGM, as suggested by Norwegian standard NORSOK C-003) that ANSYS CFD can.



The hot gas temperature contours shown near a helideck airspace are due to exhaust gas.

"ANSYS simulation software helps reduce expenses while ensuring safe travel — and that is priceless."

A QUALIFIED APPROACH TO HELIDECK DESIGN

After establishing criteria for the velocity flow field and temperature field, engineers develop a 3D geometry model of each platform and then use ANSYS Meshing to create meshes, typically with more than 5 million nodes. Mesh generation takes about an hour. Engineers simplify the representation of the FPSO platform to consider only the equipment and structures that significantly disturb the air. ANSYS CFD simulations solve for fluid flow under the most critical conditions for helicopter operations: when the wind blows from directions that either lead to turbulent flow or hit gas plumes over the helideck. The simulation strategy uses prism layers on the bottom sea surface, some detailed structures as porous media and steady-state simulations.

By comparing simulation results to wind tunnel measurements, including those derived from particle image velocimetry, engineers have found that turbulence modeling provides useful insights into turbulent flow through a very complex geometry. Despite some local differences, ANSYS CFD accurately predicts the velocity flow field and qualitatively predicts the turbulence field. As a result, Petrobras has established a new internal turbulence criterion for offshore helideck design applicable to many different platform configurations and wind orientations.

SAVING MONEY, MITIGATING RISK

By analyzing the existing helidecks, engineers are able to define the limits of environmental and operational conditions for helicopter transportation. Specifically, ANSYS CFD enables them to:

- Quantify the impacts of airflow on offshore rig modifications and new module installations
- Modify and validate helideck operational limitations
- Optimize helideck position in new installations to minimize downtime

While the use of helicopters may vary depending on the level of offshore activities and the growth of autonomous systems, there will always be a need to transport staff safely to offshore installations. By allowing Petrobras to alter payload limitations while optimizing the location of its helidecks, ANSYS simulation software helps reduce expenses while ensuring safe travel — and that is priceless.

Petrobras is supported by ANSYS elite channel partner ESSS.

Reference

CFD Assisted Offshore Helideck Design, presented on Convergence – ESSS Conference & ANSYS Users Meeting, South America Regional Conference, São Paulo, Brazil, May 5–7, 2015.

Taking Control

Subaru Forester

One of the greatest challenges for electric and hybrid vehicle designers is creating accurate control systems that balance safety, performance and energy efficiency. For nearly a decade, Subaru has relied on ANSYS SCADE solutions to develop the software code that underlies the electronic control units (ECUs) for its electric car program. By leveraging SCADE, Subaru engineers can quickly and accurately generate the mission-critical code that keeps electric vehicles running safely and smoothly, no matter how complex their technology architecture.

By ANSYS Advantage Staff

For the past decade, Subaru Corporation has been at the forefront of hybrid and electric vehicle design, beginning with its hybrid engine design for the Subaru XV in 2013 and continuing with its fifth-generation Forester SUV introduced in 2018, supported by a second-generation hybrid engine called the e-BOXER.

To answer consumer needs and maintain the company's industry leadership, Subaru engineers have included more and more advanced technologies with every new e-vehicle launch. Critical systems like propulsion, acceleration and braking are now complemented by infotainment systems, customized heating and cooling options, and other electronics that add to the driving experience.

Bringing all these sophisticated systems together, safely and seamlessly, means establishing a flawless system of controls. All components must integrate perfectly with one another, and mission-critical functions that could lead to a system failure, such as steering, must be protected.

Maintaining and managing all these systems is the job of the electronic control unit (ECU) that lies at the heart of every hybrid and electric vehicle. Supported by millions of lines of underlying embedded software code — and subject to strict regulatory oversight — the ECU is one of the most crucial elements of any electric car.

While Subaru often partners with major auto parts manufacturers to co-develop vehicle components, the engineering team in Subaru's Tokyo-based electronics department

Yuji Kawakami, senior engineer in Subaru's electronics engineering department

> **"While SCADE solutions save valuable time, they also support Subaru's long-standing commitment to delivering high levels of product quality and passenger safety."**

assumes full responsibility for designing and verifying the ECU. Since 2008, these engineers have relied on SCADE solutions from ANSYS to model and generate the ECU's embedded software code. SCADE has played an essential role in the fast, cost-effective and accurate creation of this code for each of Subaru's hybrid-electric vehicles.

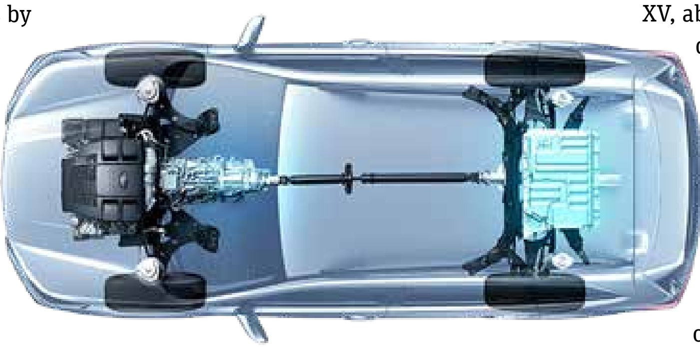
A MORE DIRECT ROUTE TO CODE GENERATION

The process of generating software code for the ECU begins by defining the control logic for bringing all the parts of the car's electronics architecture together.

This set of logic-based rules ensures that all the car's electronics are integrated safely and securely. It manages the system interactions, sends alerts when needed and can also shut down systems in an emergency. It controls vehicle dynamics, engine function, the vehicle's energy consumption and the load management of the electric battery.

For the Forester e-BOXER, which features an innovative, horizontally opposed engine, the ECU delivers added intelligence that balances optimal driving conditions with high fuel efficiency. For example, the e-BOXER's ECU is programmed to support smooth but firm acceleration via a motor assist function, without creating an energy drain on the battery. This adds new complexity to the control logic and places additional demands on Subaru's engineers to ensure the ECU's accuracy and tight control.

To create the control logic needed to drive the ECU, Subaru engineers use MATLAB/Simulink, a common industry practice. However, according to Yuji Kawakami, senior engineer in Subaru's electronics engineering department in Tokyo, the Subaru team continues to apply specialized technology to produce a systems architecture and underlying software code. While many other automotive engineering teams rely on manual methods to accomplish these tasks, Kawakami leads Subaru's effort to significantly accelerate this process by applying SCADE software.



Subaru e-BOXER

"First, Subaru engineers convert the control logic into a SCADE model of the overall system architecture, using SCADE Suite Simulink Gateway," said Kawakami. "Then our engineers apply SCADE Suite KCG Code Generator to create implementation code based on this model."

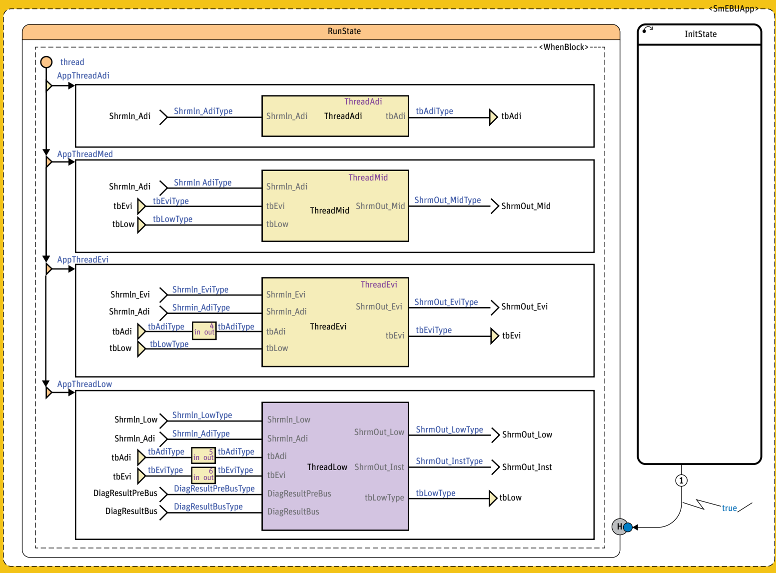
Kawakami noted that this development flow has remained the same at Subaru for nearly a decade, beginning with the ECU for the Subaru XV and continuing through the more complex ECU to support the e-BOXER. A key benefit of this development flow is that many steps — from Simulink to model creation to code generation — are automated, requiring almost no manual intervention.

"In generating control software code for Subaru's first hybrid vehicle, the Subaru XV, about 80% of the development work was automated," Kawakami stated. "As Subaru's engineering team improved its internal processes by using ANSYS SCADE, the amount of automation increased to 95% for the code underlying the e-BOXER."

Today, it only takes Subaru engineers half a day to implement a model for an ECU once the Simulink control logic has been defined. This enables Subaru's developers to modify the ECU's logic and architecture much more frequently and easily as they explore continuing design innovations.

ENSURING OUTSTANDING SAFETY, PRODUCT QUALITY AND COMPLIANCE

Subaru has a long-standing commitment to delivering high levels of product quality and passenger safety. While SCADE solutions save valuable time, they also support that commitment by



Threads management example

delivering extreme accuracy and control for software engineers. Unlike generic tools, SCADE is a specialized tool for developing embedded software code. Its model-based environment and tight scripting language eliminate the potential for human error as it translates the control logic for the ECU.

Kawakami pointed out, "In Subaru's experience, SCADE generates such a highly reliable code that a manual review is no longer required, resulting in a great reduction of tasks. The embedded software code for the ECU in an automobile hybrid system is complicated and numerically large, especially for an intelligent hybrid

"In Subaru's experience, SCADE generates such a highly reliable code that a manual review is no longer required, resulting in a great reduction of tasks."

Because the SCADE Suite KCG Code Generator meets automotive industry standards such as ISO 26262 at the highest levels of safety (ASIL D in that case), the resulting code is automatically in compliance with strict regulations — dramatically reducing the time, effort and documentation required for final code verification. SCADE is a key tool not only for meeting regulatory standards, but for supporting Subaru's commitment to passenger safety.

engine system such as e-BOXER. It typically takes significant time and effort to manually check this code, so eliminating this step has significantly improved productivity and reduced costs for the Subaru development team."

In addition, the use of SCADE facilitates a closed-loop software engineering process. In the event that the overall ECU control logic is modified in Simulink at a later date by Subaru engineers, SCADE solutions automatically and universally reflect these changes in the system model and embedded software code, eliminating costly rework and manual updates.

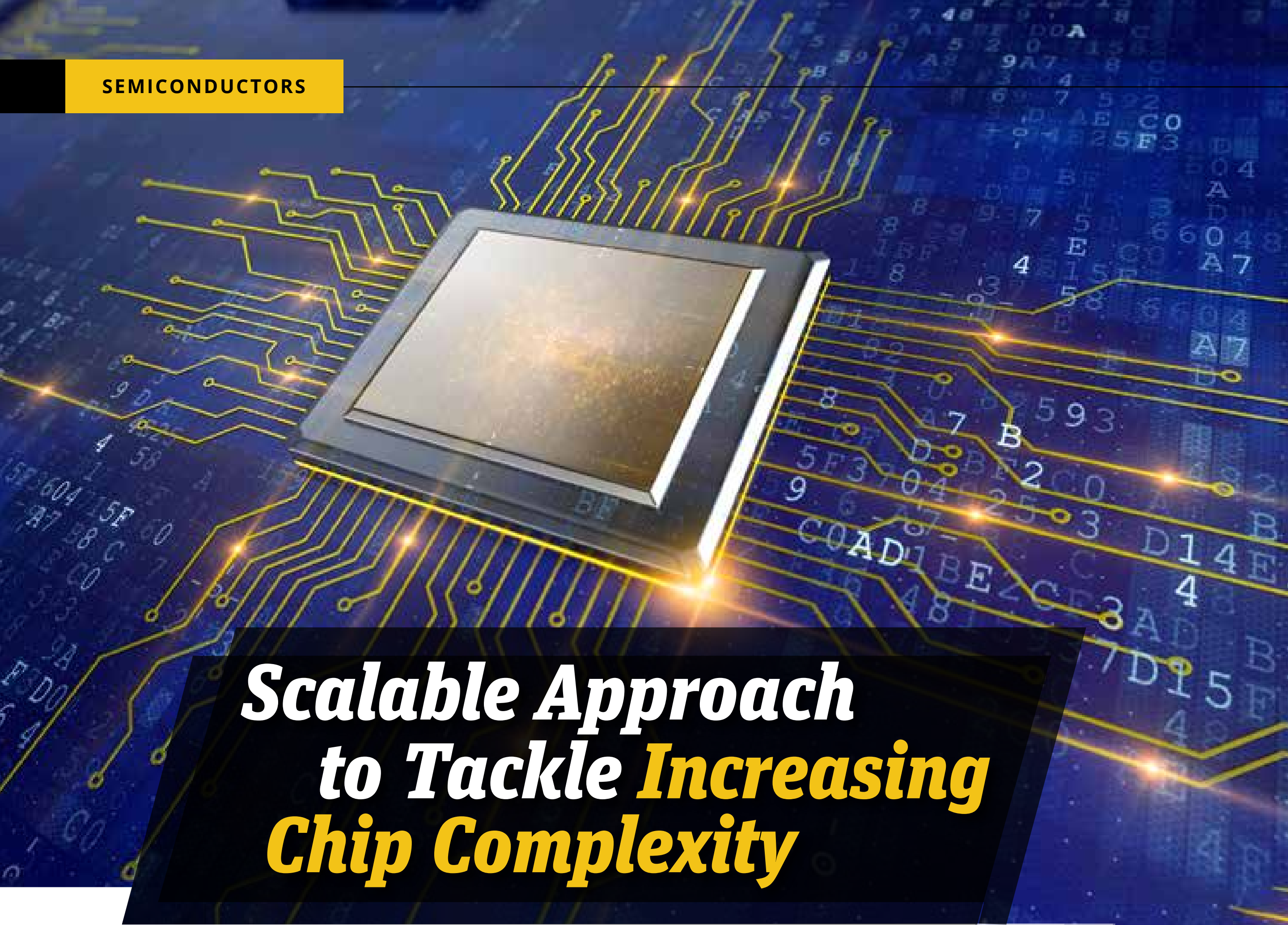
WINNING THE RACE VIA ACCELERATED MARKET LAUNCHES

"When Subaru engineers first started using ANSYS SCADE solutions, we were impressed by the initial improvements in speed and efficiency," stated Kawakami. "Over time, these improvements have only been amplified as the product development team has increased its ability to leverage SCADE's capacity for task automation.

"In developing the ECU for the e-BOXER, most steps were successfully automated, and the process included almost no human intervention," he continued. "Even non-dedicated engineers in other departments are able to convert control logic from Simulink to SCADE and generate accurate software code."

In the race to launch new hybrid and electric vehicle models, SCADE has emerged as a valuable strategic tool for Subaru over the past decade, supporting the automaker's commitment to uncompromising safety and quality. The time saved during the end-to-end development of the ECU — without sacrificing the accuracy of its control software — has been crucial to Subaru's ability to introduce innovative new technologies like the e-BOXER quickly, seizing a competitive advantage in an increasingly crowded industry segment.

Functional Safety Analysis and Support for ISO 26262 for Automotive Applications
ansys.com/iso26262



Scalable Approach to Tackle Increasing Chip Complexity

By **Anton Rozen**, Director of Backend Design, Mellanox Technologies, Tel Aviv, Israel

Increasing design complexity and multiphysics challenges hamper the productivity of system-on-chip (SoC) design teams. Engineers want electronic design automation tools that not only reduce runtime but also give them increased flexibility to critically examine and improve their designs. Mellanox engineers apply new solutions that leverage big data techniques and flexible computing resources to deliver this functionality.

High-speed networking is the backbone of connectivity in data centers. Extreme bandwidth and ultra-low-latency networking solutions are critical for the next era of data centers to efficiently process exponentially growing data from emerging AI, 5G and autonomous applications. Companies performing system-on-chip (SoC) designs for networking are challenged as chip size and complexity clash with ever-increasing time-to-market pressures. Grid complexity and the sheer number of gates increase dramatically each year, and network IC teams must design, analyze and tape out chips with dimensions of 400–500 mm or more.

Increased cross coupling of various multiphysics effects — including power and thermal reliability — pose significant challenges for FinFET design closure. Multiphysics analysis is critical to overcoming these challenges in order to design extremely large, complex and power-hungry chips, despite narrowing design margins and tighter project schedules.

Faced with this complexity, design teams must have software tools that deliver capacity, flexibility, speed and accuracy.

Mellanox, a leading supplier of end-to-end Ethernet and InfiniBand intelligent interconnect solutions and services for servers, storage and hyper-converged infrastructure, knows these challenges and trade-offs

ANSYS RedHawk-SC Introduction
ansys.com/redhawk-intro

Architecture (IR-drop methodology)	Design Complexity	IR-Drop Analysis Generation	Number of Machines	Max Node Count that Can Be Handled	Example
Monolithic	Up to 250 million nets	First 2003 onward	1	1B	ANSYS RedHawk and others
Distributed	Up to 350 million nets	Second 2013/14	32	4B	ANSYS RedHawk-DMP and others
Elastic	Greater than 350 million nets	Third 2016/18 onward	Scalable beyond 1,000 cores	Unlimited and scalable	ANSYS RedHawk-SC

As Mellanox has pushed designs into ultra-deep submicron nodes, the design complexity — and the need for more flexible and scalable design tool solutions — has increased.

firsthand. The design teams must manage and validate designs by making the most efficient use of computing resources and engineering time. To this end, the team relied on ANSYS RedHawk-SC software.

LOOKING FOR VISIBILITY

The Mellanox team needed fast turnaround time with pinpoint voltage drop accuracy to ensure power integrity and reliability for their highly complex network processors. But they also sought something that had eluded them in earlier years on other big, high-complexity designs: flexibility and speed of analysis. Because designs have evolved from a little more than 100 million nets at the 45nm node to nearly 350 million nets at 16nm, Mellanox estimates it will need to address nearly 450 million IC nets at 7nm.

This type of evolution requires tool capacity to match. A decade ago, in and around the 45nm process node, tool architectures were generally monolithic, and teams were restricted to a single machine that could handle up to 1 billion power and ground nodes at once. (A node is a connection point between any two elements in the power and ground network that are extracted. These elements could be parasitic resistance, inductance or capacitance of the wire or device instance pin connected to the wire. Node count is a metric commonly used in power integrity analysis to predict the design size; it directly impacts the runtime and memory requirements for the analysis.)

In those days, tool capacity was an issue. When conducting multiple analyses for power integrity and reliability signoff, each run (in serial rather than parallel) might take more than 24 hours. This required large servers and considerable resource allocation to complete the analysis. Worse, the system occasionally had trouble managing the complexity and would crash. The analysis then would have to be restarted from scratch.

A second generation emerged to keep up with complexity. This generation leveraged distributed compute, could scale to up to 32 machines and could handle a maximum of 4 billion nodes. This was satisfactory until ICs became even more complex.

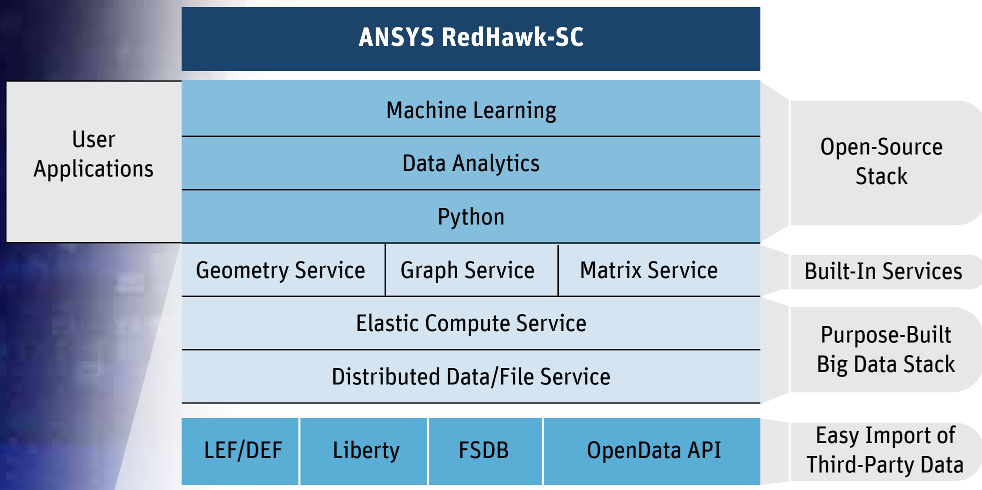
SCALING TO BIG DATA REQUIREMENTS

To deliver insights and enable the team to optimize its design, Mellanox needed a flexible, high-capacity solution that would scale for big data mining and analytics. Engineers began using ANSYS RedHawk-SC in 2018. RedHawk-SC is the latest SoC power integrity and reliability signoff platform built on ANSYS SeaScape — the world's first custom-designed big data architecture for electronic system design and simulation. SeaScape provides per-core scalability, flexible design data access, instantaneous design bring-up and many other capabilities.

	Monolithic	Distributed	Full-Chip Elastic Computing
Technology	28nm	16nm	16nm
Chip size (die area, inst. count, no. of gates)	1/4" Full Chip 96M nets	Full Chip 225M nets	Full Chip 340M nets
CPU core usage/ Machines	1 machine of 1 TB	4 machines of 1.4 TB	150 works of 72 GB
Runtime	60 hours	72 hours	24 hours

SCALABILITY COMPARISON

How the evolution of software has drastically reduced runtime for increasingly complex SoCs



ANSYS SeaScape big data elastic compute architecture

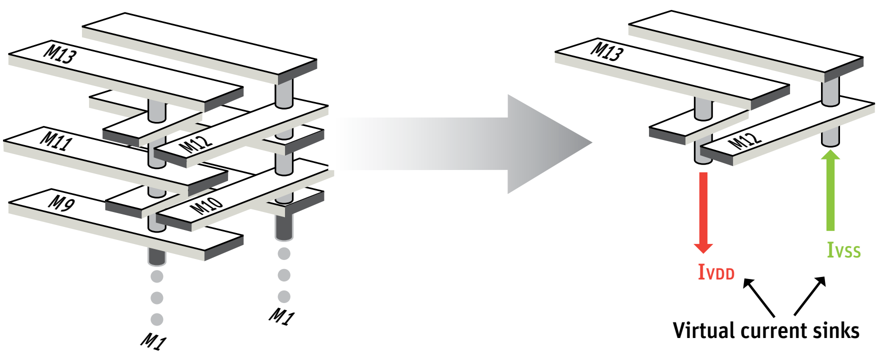
One of the keys to success lies in the elastic computing capabilities of RedHawk-SC. Elastic computing helps to process scenarios in parallel (or in serial), depending on the number of CPU cores available.

The SeaScape architecture is central to elastic computing. It rests on a distributed data/file service since data may be scattered around many locations. On top of this sits a distributed data analytics layer based on the MapReduce concept, which is fundamental to all big data analytics. This conceptually splits the data (mapping) into small chunks called shards and farms each shard for analysis. Processing can be distributed to servers as they become available, across as many servers as needed.

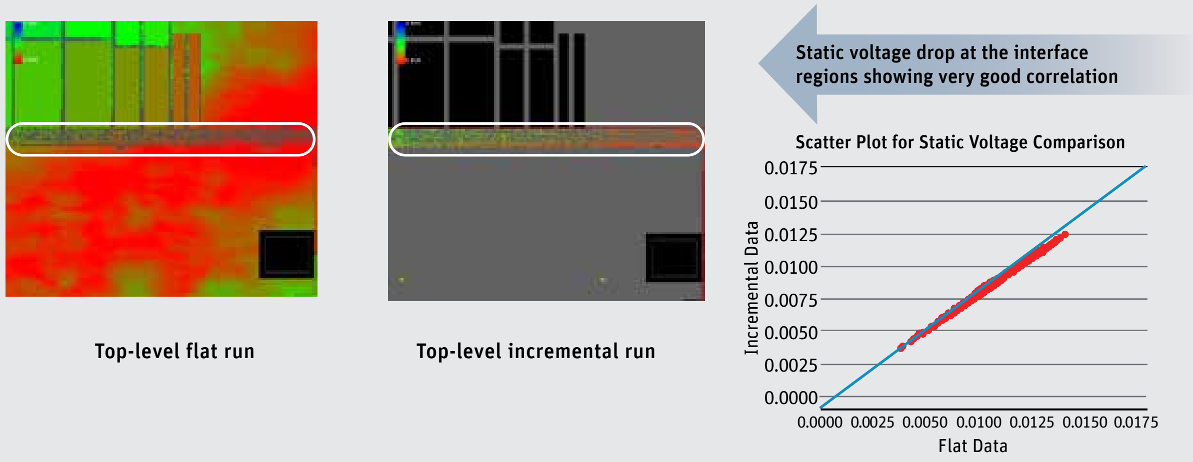
THE POWER PROBLEM

The challenge in these types of network processors is total power consumption and power dissipation. Unlike battery-powered designs, the types of designs that Mellanox works with can consume more than 200 W. So, engineers must achieve complete design analysis — accurate incremental power integrity and reliability analysis — while considering high power consumption without sacrificing accuracy or time to results.

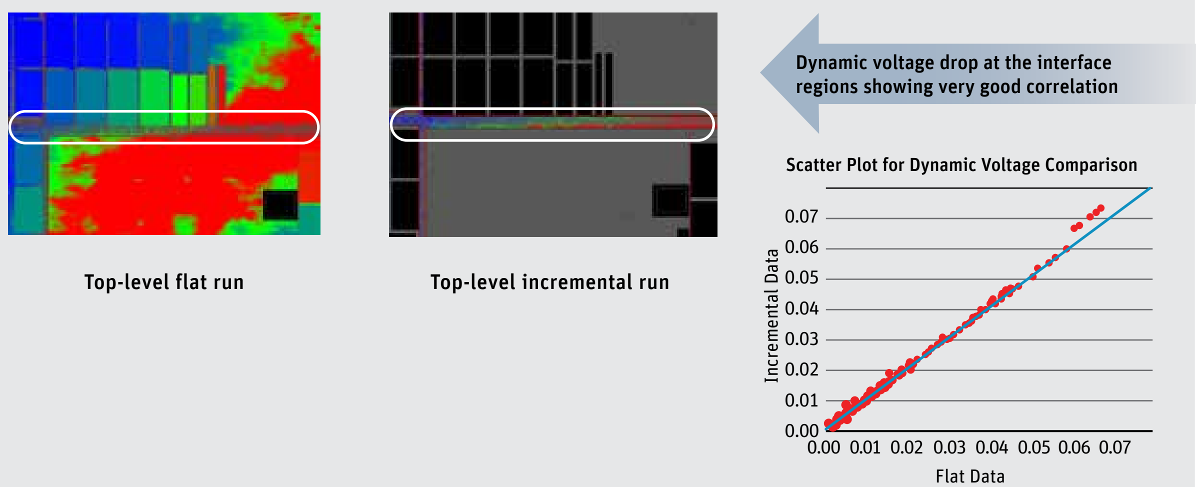
To speed up full-chip IR drop simulations, power grid roll-up methodology can be leveraged to abstract the low- and mid-level metals of the power and ground network. Such abstraction can be used in full-chip simulations. This allows teams to work at the unit level and then jump up to the top level for a comprehensive analysis of the full-chip design.



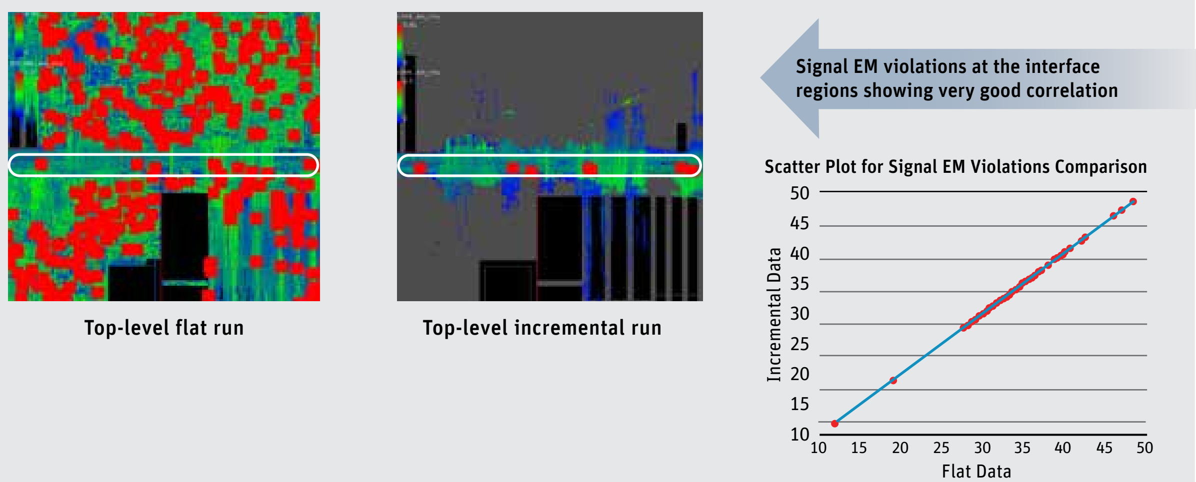
An example of a power integrity simulation using ANSYS RedHawk-SC roll-up methodology for abstracting low- and mid-level metal layers of the power grid for fast, incremental full-chip analysis



Full chip flat vs. incremental static voltage drop analysis shows very good correlation.



Full chip flat vs. incremental dynamic voltage drop analysis shows very good correlation.



Full chip flat vs. incremental signal EM analysis shows very good correlation for all top-level signal nets.

© 2019 ANSYS, INC.

Mellanox's experience reinforced the compelling features of ANSYS RedHawk-SC, including:

1. **CAPACITY** – Ability to run large designs and be able to query and visualize them in the GUI smoothly. With RedHawk-SC's elastic scalability, full-chip analysis was completed within 24 hours.
2. **FLEXIBILITY** – RedHawk-SC manages computing resources flexibly, setting a new level of effectiveness for EDA tool resources in the industry.
3. **SPEED AND ACCURACY** – Big data analytics techniques enable faster top-level runs, shorter ECO loops and faster ECO fixes with accuracy. With RedHawk-SC and actionable analytics, the productivity increased significantly by a factor of three because teams can parallelize runs and understand the impact of voltage drop in the blocks in the full chip context.

Doing a full-chip flat run is resource-intensive and time-consuming. By performing incremental analysis enabled by techniques using big data analytics, designers can create a detailed view of a specific block and abstract everything else. This enables them to perform faster analysis and conduct quicker engineering change order (ECO) fixes more easily with visibility.

ANSYS RedHawk-SC, with its elastic computing capabilities and big data–enabled analytics, gave engineers the visibility they needed to overcome some previous challenges. The team particularly appreciated RedHawk-SC's self-sustaining stability to monitor its own jobs and to renew the job if it fails.

The team also leveraged RedHawk-SC's elastic computing and its MapReduce-enabled analytics to gain key insights. MapReduce gives designers a bird's-eye view and zeros in on hotspots very smoothly. It provides powerful capabilities such as bringing up the GUI to view a full chip database in less than two minutes and navigating different areas easily, like Google Maps' functionality.

Additionally, it enables vastly more powerful compute flexibility. With RedHawk-SC's elastic scalability, large chip areas that once required huge computing resources can be broken into very small pieces for analysis. The nature of the architecture lets those elements be distributed through a company's computing resources. In this way, it maximizes hardware resource utilization and optimizes cost.

TACKLING CHIP COMPLEXITY

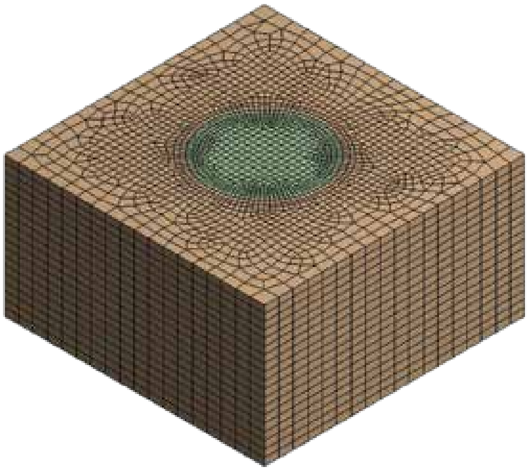
With the soaring complexity of networking IC designs, a new approach to full-chip power integrity and reliability signoff is required. This means leveraging highly parallel computing concepts to analyze large chunks of data to drive improvements in visibility, time to results, and overall design productivity and efficiency.

ANSYS RedHawk-SC's elastic compute scalability and big data analytics techniques for full chip power integrity analysis allowed Mellanox to run large designs with production-proven accuracy in less than 24 hours with pinpoint accuracy. Combining incremental power integrity/reliability and signal line electromigration analysis helped increase productivity by a factor of three.

RESOURCE

Cohen, R.; Rozen, A.; Abhijith, M.V.; Agarwal, R.; Ramachandran, S.; Johnson, S. "Fast and Accurate Incremental Power and Signal Integrity Analysis." www2.dac.com/56th/proceedings/posters/125_3.pdf (08/01/2019)

Stability Under Pressure



Finite element mesh of soil

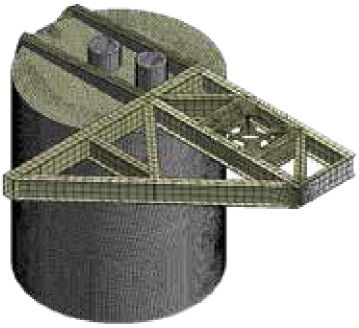
Suction piles provide a solid foundation for offshore energy structures, but overly conservative material specifications can increase project costs unnecessarily. ANSYS elite channel partner EDRMedeso used structural simulation to help oil and gas companies better understand how the sea floor interacts with pile structures to save costs.

By **Frode Halvorsen**, Head of Technology Innovation, EDRMedeso, Oslo, Norway

With winds, waves and ocean currents all capable of shifting offshore energy facilities off course, seabed anchoring is essential for safe, productive operations. Keeping large structures moored in place is no easy task, however. And that is especially true as the oil and gas industry explores water depths where conventional pile driving or gravity-based loading is not possible.

In those environments, suction piles are a proven alternative. They provide a reliable subsea anchor for production platforms, offshore drilling units, exploratory vessels, floating production and storage offloading (FPSO) units, and other facilities in water depths up to 3,000 m. They also secure fixed and floating wind turbine installations, even in shallower waters.

Described informally as resembling overturned buckets, suction piles are in fact large-



Finite element mesh of a suction pile

diameter, hollow steel cylinders with a sealed top and an open bottom. Embedding them into the sea floor is a multistep process: First, crews allow the suction pile to drive itself into the soil using its own weight, then they pump water out of the suction port. This creates differential underpressure that pulls the pile to its set depth, leaving the top flush with the seabed and providing significant foundation capacity.

Because suction piles rely on the soil around them for their holding capacity, understanding how they interact with the surrounding sea ground is a key design consideration. For more than a decade, TechnipFMC has used ANSYS structural simulation software to analyze suction pile/soil interaction. In this case, engineers used ANSYS Mechanical on a North Sea project, performing a 3D stability analysis that included a more physically correct simulation on suction pile nonlinear interaction with the sea ground. By better predicting soil behavior during suction pile installation, ANSYS Mechanical helped prove that thinner-walled suction piles could be deployed without buckling under pressure, enabling the development of more efficient, lighter and less costly technology.

SIMULATING SOIL BEHAVIOR DURING SUCTION PILE INSTALLATION

Soil is the stabilizing structure for suction piles, but offshore soil profiles, soil resistance and mechanical characteristics vary widely around the world and respond to loading conditions in unique ways. Depending on the seabed soil, it can take varying amounts of pressure to embed

© 2019 ANSYS, INC.

“The oil and gas industry explores water depths where conventional pile driving or gravity-based loading is not possible.”

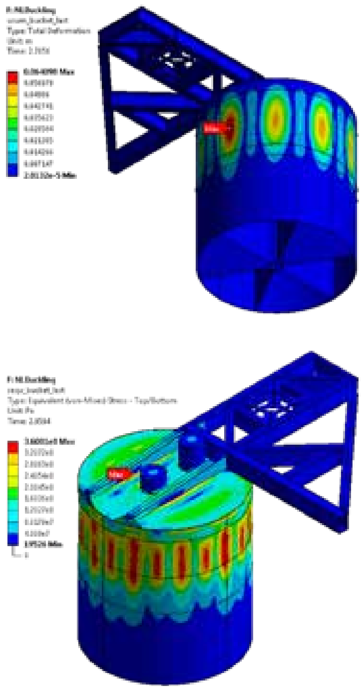
the suction pile, so it must be designed to withstand the pressure required to pull it down or it will collapse. In general, very stiff soil provides more support and reduces the likelihood of the suction pile buckling. On the other hand, very weak soil has a lower buckling threshold.

In the past, the interaction between the suction pile and the sea ground was often simulated by radial springs with nonlinear stiffness. However, this procedure can describe phenomena only in a simplified manner and results in very conservative — even uneconomic — design solutions incorporating thick-walled steel. Since the average suction pile is 10 to 12 m long and 6 to 7 m in diameter, and certain foundation bottom structures have as many as four suction anchors, the ability to reduce the thickness even a few centimeters can result in considerable material savings. In a North Sea project, using ANSYS Mechanical simulation reduced TechnipFMC’s steel cost by NOK 2.5M ($281,504 USD).

To verify the three-dimensional behavior of the soil during soil–pile interaction, EDRMedeso engineers used the finite element analysis tools and geomechanics library in ANSYS Mechanical, which analyzes rigid and flexible bodies within a single solver. ANSYS Mechanical leveraged Dynardo’s multiPlas software, which is now included in the ANSYS Mechanical geomechanics toolbox, to examine elastoplastic material behaviors.

Specifically, the simulation engineers:

- Modeled a quarter-scale integrated template structure with soil surrounding and filling the interior of the suction pile
- Generated a mesh of the entire assembly
- Performed nonlinear buckling analysis and gradually increased suction pressure



Buckling mode (top) and von Mises stresses (bottom) to determine the ultimate strength of a material and identify the point at which it will collapse

To validate the suction pressure capacity, engineers increased soil spring stiffness according to DNV GL specifications. The suction capacity obtained with various soil spring stiffnesses verified higher buckling resistance at increased working loads, providing confidence in the simulation and in the efficacy of thinner-walled suction piles.

MATERIALS CAN ACCOMMODATE INCREASED WORKING LOADS

Suction piles offer considerable advantages compared to other deep foundation technology. They are easier, less expensive and less time-consuming to install and require minimal seabed preparation. Suction piles are also environmentally sound — installation is so quiet it does not disturb marine life. Removing them after decommissioning is simple and leaves no metal behind.

Suction pile design is straightforward, but overdesign can be costly. Even small changes can generate significant savings. By showing a higher model-predicted failure load than previous calculations allowed, EDRMedeso and ANSYS Mechanical helped TechnipFMC eliminate conservatism in material selection while enabling significantly increased working loads and improved design and installation economics.

“ANSYS Mechanical has enabled TechnipFMC to optimize suction pile design toward a more realistic solution,” said Nuno Vaz, who works in Lead Structural Design Engineering, EMS Analysis Well Control at TechnipFMC. “Combined with the current Dynardo soil material model, it has given us more confidence about suction pressure limitation during installation.”

Advancing Concrete and Soil Structural Simulation
ansys.com/soil-simulation

Simulation in the News

ANSYS 2019 R3 UPDATES SPAN PRODUCT PORTFOLIO

***Digital Engineering*, September 2019**

ANSYS 2019 R3 includes ANSYS Autonomy, which enables engineers to develop safer autonomous vehicles (AVs) through advanced closed-loop scenario simulation, automated driving and control software development, functional safety analysis, and sensor, camera, lidar and radar simulation.

ANSYS 2019 R3 also includes multiple tool enhancements to ANSYS VRXPERIENCE Driving Simulator powered by SCANeR; updates ANSYS VRXPERIENCE; introduces ANSYS Minerva, powered by Aras; simplifies workflows across the company's product portfolio and more, according to *Digital Engineering*.

ANSYS 2019 R3 delivers significant simulation capabilities that enable vehicle-to-everything communication for autonomous vehicles.

ANSYS TO ACQUIRE LIVERMORE SOFTWARE TECHNOLOGY CORPORATION

HPCwire, September 2019

HPCwire reports that ANSYS has entered into a definitive agreement to acquire Livermore Software Technology Corporation (LSTC), the premier provider of explicit dynamics and other advanced finite element analysis technology. Once closed, the acquisition will empower ANSYS customers to solve a new class of engineering challenges, including developing safer automobiles, aircraft and trains while reducing or even eliminating the need for costly physical testing.

AUTODESK AND ANSYS ANNOUNCE AUTOMOTIVE ALLIANCE

Engineering.com, September 2019

Autodesk and ANSYS have announced a new collaboration aimed at the automotive industry, *Engineering.com* reports. The partnership will see an integration of Autodesk's VRED, an automotive visualization and prototyping application, and ANSYS tools for lighting simulation. The two companies expect that the integration will enable automotive designers to obtain photorealistic visual representations of their vehicles, providing workflows that more accurately reflect physical reality.

ANSYS NAMED TO FAST COMPANY'S LIST OF THE 50 BEST WORKPLACES FOR INNOVATORS

Fast Company, August 2019

Fast Company announced that ANSYS has been named to its Best Workplaces for Innovators list, which honors businesses and organizations that demonstrate a deep commitment to encouraging innovation at all levels. Developed in collaboration with Accenture, the 2019 Best Workplaces for Innovators showcases 50 winners from a variety of industries.

A mockup of the FCAS next-generation fighter jet was on display at the 2019 Paris Air Show.

AIRBUS TO USE AI TO DESIGN FLIGHT CONTROL SOFTWARE

Aviation Today, June 2019

ANSYS is partnering with Airbus Defense and Space to develop a new artificial intelligence (AI) design tool that can be used to create the embedded flight control software for Europe's Future Combat Air System (FCAS). According to *Aviation Today*, FCAS is a next-generation air combat development program involving France, Germany and Spain to develop a system of fully automated remote air platforms and sixth-generation fighters that will replace their current generation of Eurofighter and Rafale jets. The partnership between Airbus and ANSYS will engineer an advanced ANSYS SCADE tool that links traditional model-based software development with new AI-based development flow.

ANSYS SIGNS DEFINITIVE AGREEMENT TO ACQUIRE DYNARDO

HPC Wire, October 2019

ANSYS has entered into a definitive agreement to acquire Dynardo, a provider of simulation process integration and design optimization (PIDO) technology. The acquisition will give ANSYS customers access to a full suite of process integration and robust design tools — empowering users to identify optimal product designs faster and more economically, according to *HPC Wire*.

SUBARU USING ANSYS TO DEVELOP HEV CONTROL SYSTEMS

Green Car Congress, August 2019

Subaru Corporation is developing control systems for its next-generation hybrid electric vehicles (HEVs) using ANSYS embedded software solutions, according to *Green Car Congress*. ANSYS enables Subaru Corporation engineers to quickly and accurately generate code that ensures the operational reliability of key, interconnected HEV systems to help keep drivers safe on the road.

ANSYS CLOUD GAINS MOMENTUM

Scientific Computing World, August 2019

Engineers are unlocking increased compute capacity to advance 5G, autonomous systems, electric vehicles, and more thanks to ANSYS Cloud high-performance computing (HPC), powered by Microsoft Azure. Available within ANSYS engineering simulation software, ANSYS Cloud is helping organizations run high-fidelity simulations, shorten development cycles and improve time to market. Following its release in February, ANSYS Cloud has hundreds of customers taking advantage of its functionality, reports *Scientific Computing World*.

ANSYS AND BMW GROUP PARTNER ON SIMULATION TOOL CHAIN FOR AUTONOMOUS DRIVING

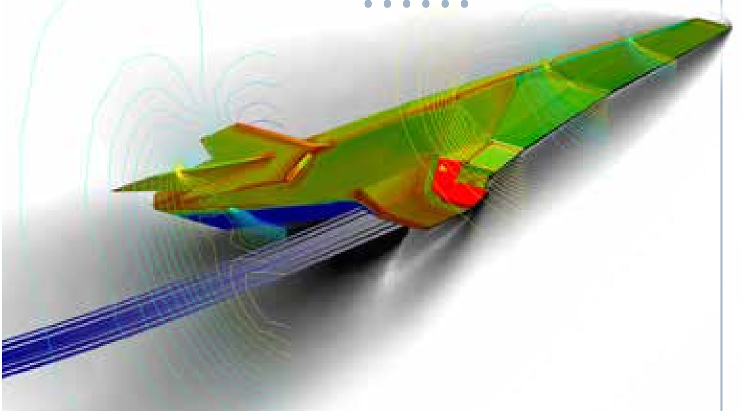
HPCwire, June 2019

The simulation tool chain will enable highly automated and autonomous driving. The multi-year agreement drives the development of BMW Group's Level 3 offering and Level 4–5 technology, delivering high/full automation for the BMW iNEXT, expected to launch in 2021.

SIMULATION PROVIDERS TEAM UP TO COMBINE AV TEST EXPERTISE

Automotive Testing Technology International, July 2019

AVSimulation and ANSY have entered a partnership that will integrate AVSimulation's high-performance simulation software, SCANeR Studio, with VRXperience, an immersive ANSYS solution that combines virtual reality (VR) capabilities with physics-based simulation. Embedded within VRXperience as its driving simulator module, AVSimulation's SCANeR Studio product creates an ultra-realistic virtual world in which an infinite number of driving scenarios can be recreated with numerous variabilities on high-performance clusters or on public cloud, according to *Automotive Testing Technology International.*



HYPERSONIC WEAPON DEFENSE SYSTEM PRIORITIZED BY DOD

Military Embedded Systems, August 2019

To engineer the U.S. military's next-generation missile defense system, Analytical Graphics, Inc. (AGI) and ANSYS are incorporating high-fidelity, multiphysics simulations with multi-domain mission-level modeling into early stages of missile defense system development. This could enable warfighters to combat high-speed, highly maneuverable hypersonic weapons, reports *Military Embedded Systems.*

CHEERS AND KEEP READING

Chris Reeves, who has edited *ANSYS Advantage* magazine since 2007, is retiring after more than 25 years in engineering communications at ANSYS and its acquisitions.

Over the years, Reeves ensured that the award-winning magazine was on the cutting edge of engineering technology, expanded its scope to reflect the company's growing portfolio of simulation solutions and successfully leveraged the magazine's content on ANSYS.com, social media and mobile platforms. In 2016, she led the launch of *Dimensions* magazine, which is targeted to executives facing the business challenges of digital transformation.

"It has been my privilege to work with a great team to produce more than 50 issues of *ANSYS Advantage* and *Dimensions* magazines over the past 12 years," she says. "As I head into retirement, I would like to thank the many dedicated people who have taken time to contribute to these publications. Thank you editorial and design teams, writers, channel partners, ANSYS partners, staff and especially ANSYS customers. It has been my honor to tell your stories. I am confidently leaving the magazines in the capable hands of **Jamie Gooch**."

Keep reading — there is much more to come.

MOTOR-CAD ADDED TO ELECTRIC MACHINE DESIGN WORKFLOW

ANSYS is creating a powerful design-to-validation workflow for electric machines through an agreement with Motor Design Limited (MDL) to distribute Motor-CAD. By combining the leading electric motor design software tool with ANSYS' multiphysics analysis capabilities, ANSYS is extending simulation into the design phase of the electric machine product lifecycle. Motor-CAD enables design engineers to evaluate motor topologies and concepts across the full operating range and to produce designs that are optimized for performance, efficiency and size.


PERVASIVE ENGINEERING SIMULATION MEANS
YOU CAN CUT THE SCRAP
Parts that won't print, fit or perform?
With ANSYS' complete simulation solution for metal AM, you'll eliminate trial-and-error waste for faster and flawless fabrication of 3D-printed parts.
SCRAP
Learn more at: ansys.com/additive
ANSYS