ANSYS

Excellence in Engineering Simulation

ADVANTAGE

ISSUE 2 | 2016

Engineering the Internet of Things

Wired Into Health
Safe design for implantable devices

Quantum Leap
Multiphysics enables quantum computing

The Backbone of the IoT
Ultrahigh-speed system-on-chip development

NEW PRODUCTS, SAME VALUES, **MORE DIFFICULT PROBLEMS**

Designing new products for today's connected world doesn't necessarily mean new engineering challenges, it just means that the engineering challenges we wrestle with on a daily basis have become a whole lot more difficult to overcome. While product values such as quality, innovation, time-to-market and cost control will continue to separate the winners from the also-rans, the winners will be those who can best adapt to the step changes in engineering challenges in this rapidly shifting landscape.

By **Rob Harwood**, Director, Industry Marketing, ANSYS

This edition of *ANSYS Advantage* is focused on the engineering opportunities and challenges introduced by the fast-growing Internet of Things, or IoT. But, what is the IoT?

Often, we think of the IoT in terms of things. Cell phones, tablets and other personal electronics garner much attention based on the world-wide explosion of personal technology devices — and with good reason. According to Gartner, a leading information technology research and advisory company, by 2020 more than 26 billion personal devices will be in use, generating over $300 billion in revenue and over $1.9 trillion in global economic value. But today, an exciting trend in IoT is the expansion of connected things into the industrial world—referred to as the Industrial Internet of Things (IIoT). This is mirrored in almost every industry sector, including connected cars, wearable medical devices, drones and even military applications.

But the IoT is more than things. It also includes the robust global network that coordinates and connects these things to one another, as well as the cloud, which consists of data centers that host the vast collection of software that is able to translate the data from the things into actionable intelligence.

Each of these components of the IoT must not only be engineered for optimal performance on its own, but must perform efficiently when integrated into an IoT solution.

Whatever your industry, the things, networks and cloud systems that make up the IoT will impact your company, creating radical changes in your business model and potentially your product line, as you install advanced communication, sensing and other capabilities. At ANSYS, many of our customers express concern about what that impact will be — and how engineers can lead change, instead of respond to it.

> "Whatever your industry, the IoT will impact your company."

While the IoT might seem like a brave new world, the core tenets of product development leadership still hold. Your customers are looking for robust products that perform as expected. These products allow your company to minimize warranty expenses and uphold your product promises. You must still focus on innovations that improve product performance and position you for a strategic edge—and you still need to launch products as quickly as possible to stay ahead of competitors. And, you must minimize the internal cost of product development to maximize profits.

Many fundamental engineering challenges have not changed in light of the IoT. These challenges just became more difficult to overcome. We have identified a number of common, traditional engineering challenges faced by our customers that are significantly more complex as a result of the pressure to develop IoT products. These include size, weight, power and cooling, sensing and connectivity, reliability and safety, and integration and durability.

Our research with the Aberdeen Group has shown that best-in-class companies address the increased difficulty in these engineering challenges by breaking down the silos in their engineering organization; simulation is a key part of this strategy. So whatever the next generation of products looks like, engineering simulation will remain a critical tool, becoming even more essential for success in the IoT economy. With its full range of simulation capabilities, ANSYS stands ready to support you as you embrace the opportunities presented by the IoT.

Why Engineering Simulation is Critical to Your Smart Product's Success in the Internet of Things
ansys.com/iot1

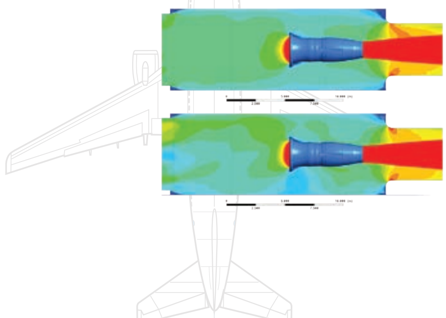
Table of Contents

Engineering the Internet of Things

ABOUT THE COVER

There are already more smart devices connected to the internet today than there are human beings in the world. Designing these Internet of Things (IoT) devices is creating massive opportunities for existing businesses and giving rise to brand-new markets and companies. ANSYS is uniquely positioned to offer the most complete line of engineering simulation solutions to meet the product design challenges of the IoT head-on.

Simulation@Work



Departments

Join the simulation conversation
ansys.com/Social@ANSYS

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

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

Executive & Managing Editor
Chris Reeves

Editorial Adviser
Tom Smithyman

Editorial Contributor
ANSYS Customer Excellence North America

Senior Editor
Tim Palucka

Editors
Erik Ferguson
Kara Gremillion
Thomas Matich
Mark Ravenstahl
Ravi Ravikumar
Walter Scott
Anatole Wilson

Art Directors
Ron Santillo
Dan Hart

Designer
Dan Hart

Cover Illustration
Gregg Weber

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

Subscribe at ansys.com/magazine

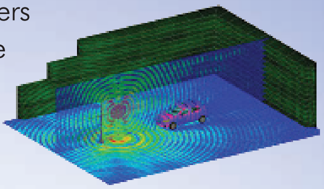
Realize Your Product Promise®
ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design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

ENSURING A WELL-CONNECTED FUTURE

All around us, electronic devices are proliferating — and old, familiar products have newer, smarter functionality. As the Internet of Things grows larger every day, ANSYS offers the full range of simulation capabilities to maximize product performance across a wide range of criteria.

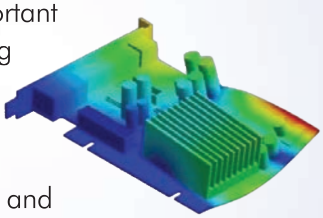
By **Sudhir Sharma**, Director, High-Tech Industry Marketing, ANSYS

The business media is filled with statistics about the continued, explosive growth of the worldwide Internet of Things, or IoT — and there is plenty of evidence to support these projections. All around us, phones, tablets and other devices keep us connected 24 hours a day, seven days a week. Smart functionality powers connected cars, drones, medical devices and industrial equipment. But the IoT is much more than just smart functionality in devices. The full value of the IoT will be realized through data analytics, powered by faster networks and quantum leaps in computing and data center technologies.

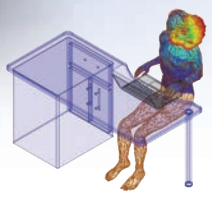


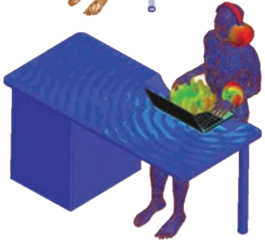
The IoT is clearly an exciting business opportunity for large and small companies. Yet success in the IoT economy depends on the ability of companies to constantly reinvent their offerings. Increasingly, product development and engineering teams need to address critical challenges related to communication system design, sensor design and product reliability to out-innovate the competition.

In this incredibly fast-paced environment, simulation software is an important strategic tool for creating a meaningful competitive advantage by getting the newest product model or next-generation features into customers' hands as fast as possible. Through engineering simulation, product designers can identify and address any functional flaws, such as impractical power demands or faulty antenna design, as quickly as possible — and as early as possible in the design cycle, when mistakes are less costly to address.



To help drive the exponential growth of the IoT, ANSYS has developed the industry's most comprehensive simulation solutions that improve the performance and reliability of electronic devices, as well as more traditional products that now include smart functionality. From confirming a product's structural integrity with ANSYS Mechanical to verifying the performance of embedded software with ANSYS SCADE, product developers can attack the full range of design issues associated with the IoT by relying on the proven power of ANSYS simulation software.





SMART DESIGN VIA SIMULATION

The multiphysics, multidomain capabilities of the ANSYS simulation software portfolio are especially critical in engineering today's smart, connected products. Because these products have complex functionality, smart product development teams demand an extremely high level of reliability, precision, robustness and innovation. At the same time, these teams face enormous pressures to keep costs low and accelerate launches. To achieve these goals, companies can neither design in silos nor rely on traditional build-and-test methods.

Not only is simulation a competitive requirement today, but it has significantly leveled the playing field, enabling smaller companies to compete with established market leaders. Using simulation, a few engineers can virtually prototype and refine many ideas quickly and cost-efficiently. Their ability to go beyond traditional engineering discipline boundaries — and instead leverage multidomain and multiphysics analyses — is consistent with their company's overall commitment to innovation.

Engineering the Internet of Things
ansys.com/ensuring

ENGINEERING THE IOT: FIVE KEY CHALLENGES

While every smart product has its own design challenges, the exponential growth of the Internet of Things means that some common design requirements are emerging. For example, each new generation of smart products typically must be smaller, lighter in weight and more power-efficient than the previous generation.

There are five key engineering challenges created by the rise of the IoT. The sheer size of the IoT opportunity, is creating new competitors for many established market leaders. Data from the Aberdeen Group highlights that winners and losers in the IoT economy will be separated by their ability to address these five design challenges successfully.

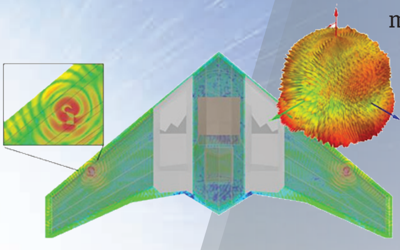
SIZE, WEIGHT, POWER AND COOLING (SWAP-C)

Whether designing planes, cars or smartphones, engineers typically need to optimize IoT products for size, weight, power and cooling — a set of design requirements popularly known as "SWAP-C." As consumers demand greater functionality, including pervasive connectivity and sensing, engineers are forced to add more electronic components. This high density of electronics brings new challenges in terms of product size, weight, energy demand and thermal build-up. Often, traditional products must be completely reimagined and redesigned for a new era. For example, as modern hearing aids transform into smartphone-connected devices with greater functionality, their design now includes a flexible printed circuit board (PCB), a battery, a receiver, an antenna and, in many cases, a telecoil. The flexible PCB alone incorporates more than 60 different components and integrated circuits. Engineers must manage all these components in a constrained space, while optimizing performance. This means relying on simulation to make design trade-offs quickly and cost-efficiently.

SENSING AND CONNECTIVITY

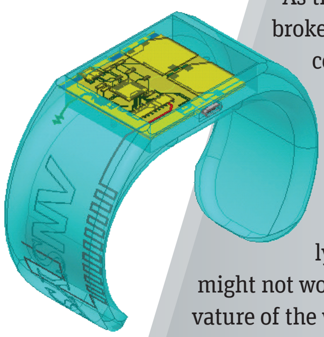
Today's products are considered "smart" because they can sense their environment, communicate with other electronics, and enable faster, more-informed decisions and outcomes. For example, modern cars are equipped with a host of sensing and communication technologies to make drivers safer, better connected and more informed. This is a huge and growing market; in fact, revenues are expected to grow from $8.4 billion today to $30 billion in 2020. The sheer amount of smart functionality in cars today is staggering. For example, adaptive cruise control technology utilizes radar sensors embedded in the bumper to keep cars at a safe distance from one another. Blind-spot monitors and lane-departure warning systems help drivers avoid collisions. Modern vehicles can even monitor and report traffic conditions, informing other drivers and suggesting alternate routes via global positioning system (GPS) capabilities. Unlike previous generations of automotive engineers, engineers designing today's cars need to consider electromagnetic interference, signal integrity, uninterrupted connectivity and other complex issues that may affect electronic performance. Simulation provides a means of maximizing reliability and ensuring a robust design from the earliest stages of product development.

RELIABILITY AND SAFETY



As smarter products proliferate and we increasingly rely on them for critical decision-making, safety and reliability become even more important product design considerations. While the IoT is driving enormous revenues, those revenues can't be outweighed by the cost of maintenance, repair, warranty charges or lack of uptake by the market. In addition, many products — such as those in the automotive, aerospace and healthcare industries — operate in safety-critical environments. Because lives are at stake, these products need to meet the highest standards for reliability and safety. An often-overlooked, yet mission-critical, aspect of the IoT is the embedded control and display software needed to operate the integrated mechatronic systems that guide connected cars and aircraft. Validating the tens of millions of lines of safety-critical embedded software code that underlie these systems is essential. Simulation and modeling enable the fast, automated production of flawless code that is needed whenever human safety is involved.

INTEGRATION

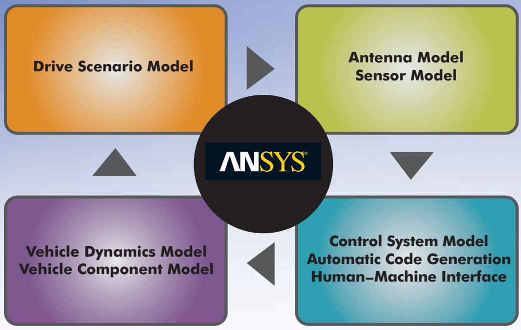


As the complexity of smart, connected products has increased over time, engineers have broken down the design process into smaller pieces to make it more manageable. While a component-level, bottom-up design methodology allows for very thorough verification of pieces and parts, significant late-stage design issues can arise when components are actually assembled to create a system. Finding system-level flaws late in the development cycle can lead to over-design, cost overruns and ill-considered design trade-offs as engineers scramble to meet product launch deadlines. For example, an antenna designed for a wireless fitness band may work perfectly when analyzed as a single component. But, once installed on the actual band, the antenna might not work as expected when adversely affected by such real-world factors as the curvature of the wristband, the presence of a biometric sensor antenna or even the metal clasp that fastens the wristband. Simulation can help predict system-level performance issues in a risk-free, low-cost virtual environment.

DURABILITY

While tiny and often unseen, trillions of sensors and microprocessors form the backbone of the IoT. These hardworking electronics collect and share useful information 24 hours a day, seven days a week. They need to perform reliably not just in optimal conditions, but must also withstand the rigors of harsh, unpredictable environments. For instance, consider a sensing system at the end of a drill bit in the oil and gas industry, in the highest-temperature regions of a jet engine, or in an unmanned military vehicle subjected to a hostile electromagnetic environment. As an extreme example, new solar-powered drones developed under Facebook's "Aquila" project will leverage lasers to provide internet access to remote parts of the developing world — flying for up to three months at a time. It is virtually impossible to explore these types of extreme operating scenarios using physical testing, so simulation plays an essential role in bringing these innovations to market with a promise of reliability. While not all products need to endure extreme conditions, every product must be verified for durability. Anyone who has dropped a smartphone understands the rigors of everyday usage.

THE IMPORTANCE OF A CONSOLIDATED SIMULATION PLATFORM



^ The ANSYS consolidated simulation platform provides all the capabilities required to model the connected car.

Just as the Internet of Things has changed our daily lives, smart devices have also revolutionized foundational product development processes. Because these products are multifunctional, their design requires the input of engineers from multiple disciplines. As cross-disciplinary teams are formed and the barriers between traditional engineering functions are broken down, engineers now need shared tools that can work across multiple departments and disciplines.

ANSYS has been a pioneer in providing versatile, broad-reaching engineering simulation tools that connect a series of discrete functional application areas in a common working environment, or simulation platform. Today, ANSYS provides both industry-leading discrete application simulation capabilities, as well as the consolidated platform needed to deliver an integrated IoT product development solution.

In addition to fostering cooperation and collaboration across different disciplines, a shared platform for simulation delivers a number of tangible benefits. Research has shown that product development teams that consolidate their simulation-driven product development capabilities on a single platform are 33 percent more likely to meet their new product introduction targets. Additionally, they reduce product development time by 7 times and costs by 2.5 times, compared to teams that use siloed development methods. These critical metrics can mean the difference between success and failure in today's fast-paced, highly disruptive and competitive business environment.

Holistic Development vs. Siloed Development

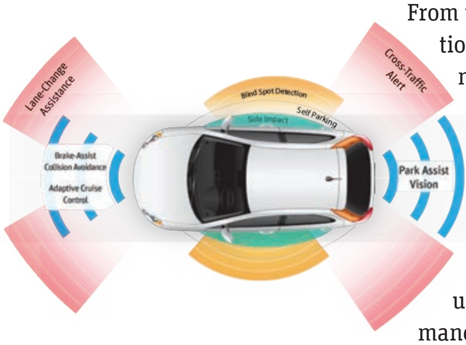
Holistic Development Experiences:

Length of Development Time	▼	**7x** reduction
Overall Product Cost	▼	**2.5x** reduction
Warranty Costs	▼	**53%** more likely to decrease
Number of Change Orders After Release to Manufacturing	▼	**123%** more likely to decrease
Percent Successful New Product Introduction (NPI) Rate	▲	**33%** more likely to achieve

Aberdeen Group, 2016

CASE IN POINT: ADVANCED DRIVER ASSISTANCE SYSTEMS

For applications such as connected cars, a consolidated simulation platform is an absolute necessity. An advanced driver assistance system (ADAS) is a classic example of a large, complex system encompassing the entire vehicle — and incorporating an enormous range of product functionality. From traditional features like cruise control and automatic airbags to newer functionality such as parking-assistance and lane-departure warning systems, the modern car equipped with ADAS is a multidisciplinary engineering marvel. To virtually validate ADAS design, all other major vehicle systems — including control systems and human–machine components such as brakes and vehicle dynamics — need to be modeled in a comprehensive system-level simulation. Next, the performance of that comprehensive vehicle and ADAS model must be tested in a simulated model of roads, buildings and pedestrians under diverse driving scenarios. The ANSYS consolidated simulation platform provides all the capabilities required to model these performance aspects, providing a one-stop resource for automotive product development teams. The ANSYS platform is also open and collaborative, enabling the participation and input of suppliers in today's fast-paced, highly disruptive and competitive business environment.



ANSYS: YOUR TRUSTED PARTNER

ANSYS does not manufacture electronics or devices, but today the Internet of Things is absolutely critical to our product offerings and our customer value proposition. Whatever your industry or product focus, the IoT is poised to impact your business in significant and often unexpected ways.

At ANSYS, we've developed an expanded range of capabilities, including electronic and embedded software modeling, to help you anticipate and prepare for that impact with innovative new products and smart functionality that enable you to thrive in the IoT era. In fact, the world's leading companies are already using ANSYS solutions to deliver the most innovative smart products — from smartphones and spacecraft to autonomous vehicles, drones, robots and wind turbines.

As the IoT continues to evolve, ANSYS will remain your trusted partner — delivering the proven simulation capabilities you've come to rely on, along with new capabilities that support your continued product development success in a transformed world. We can help you engineer, design and test the best possible products for the Internet of Things.

In this issue of *ANSYS Advantage*, we invite you to read how ANSYS simulation is not only applied to the development of IoT technologies for smarter products and faster networking equipment, but also for quantum computing data centers, which require advanced energy and heat management solutions enabled by ANSYS simulation.

> "Simulation software is an important strategic tool for creating a meaningful competitive advantage."

SMART PRODUCTS REQUIRE COLLABORATIVE DESIGN

Independent research has shown that smart product design requires an increase in communication and collaboration among functional engineering teams. A product designed without collaboration can lead to integration issues, especially when subsystems are built and over-designed as each team adds its own safety margins. Launch delays, reliability issues and cost overruns are other risks.

Engineering simulation is an important vehicle for fostering collaboration. Today, best-in-class companies use a consolidated simulation platform to analyze component- and system-level behavior, as well as subsystem interactions, before building physical prototypes. Designers at these companies are able to quickly explore the performance of numerous design alternatives at a rapid pace. This ability to analyze multiple alternatives enables designs to be optimized for cost, quality and/or performance. The metrics highlight just some of the benefits of a simulation-based design approach executed on a consolidated platform that enables cross-functional engineering interaction. ∧

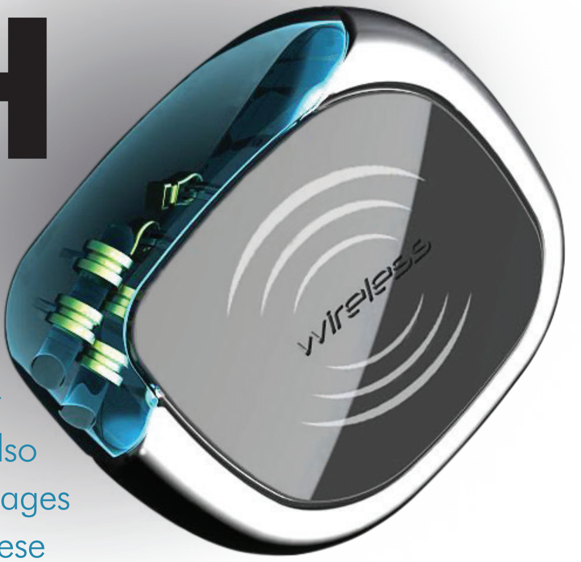
Simulation vs. No Simulation
Simulated Environments Experiences:

Length of Development Time	▼ **9x** reduction
Overall Product Cost	▼ **4x** reduction
Warranty Costs	▼ **89%** more likely to decrease
Number of Change Orders After Release to Manufacturing	▼ **2.5%** more likely to decrease
Percent Successful New Product Introduction (NPI) Rate	▲ **67%** more likely to achieve

Aberdeen Group, 2016

Engineering the Internet of Things
ansys.com/future

WIRED INTO HEALTH



The Internet of Things for healthcare requires antennas in implantable medical devices to operate safely within the human body, over longer distances than before and at more than one frequency. These devices must also be reliable in the wide range of body types and ages that comprise the human population. To take these many factors into account, Cambridge Consultants uses ANSYS software to model body variations and simulate antenna performance.

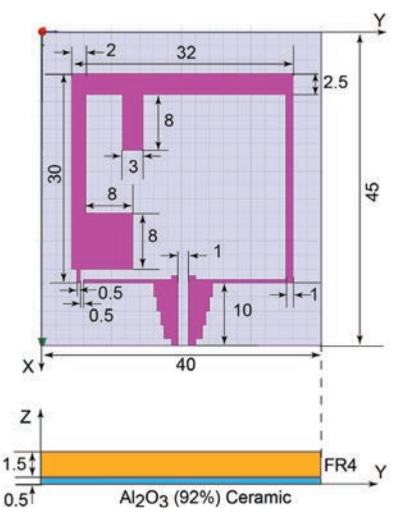
By **Arun Venkatasubramanian**, Associate Director, Cambridge Consultants, Boston, USA

Wireless technology adds a new dimension to medical implants, allowing remote monitoring and treatment optimization. However, to design a successful wireless implant, designers must address many different use cases and regulatory requirements, each of which poses its own unique challenges.

Typically, a smart medical implant must communicate wirelessly with an external handheld device in at least three different environments:

- The operating room, where the implant is programmed before being inserted into the patient.
- The medical office, where a clinician needs to carry out follow-up monitoring by wirelessly communicating with the implant using the external programmer device.
- The home, often using a bedside wireless box that talks to the implant to relay diagnostic information, as well as any alarm conditions, immediately to the doctor/caregiver.

Body tissue affects wireless radio performance by causing reflections and absorbing some of the wireless signal, as well as affecting the operating frequency and bandwidth of the antenna. The patient's body type has a significant effect on the communication distance between the implant and the external device. In recent years, Bluetooth® Smart communication to smartphones has emerged as a popular choice for connectivity. Implantable device manufacturers will no doubt want to explore this option. Bluetooth operates at much higher frequencies than current wireless technologies used in medical devices,

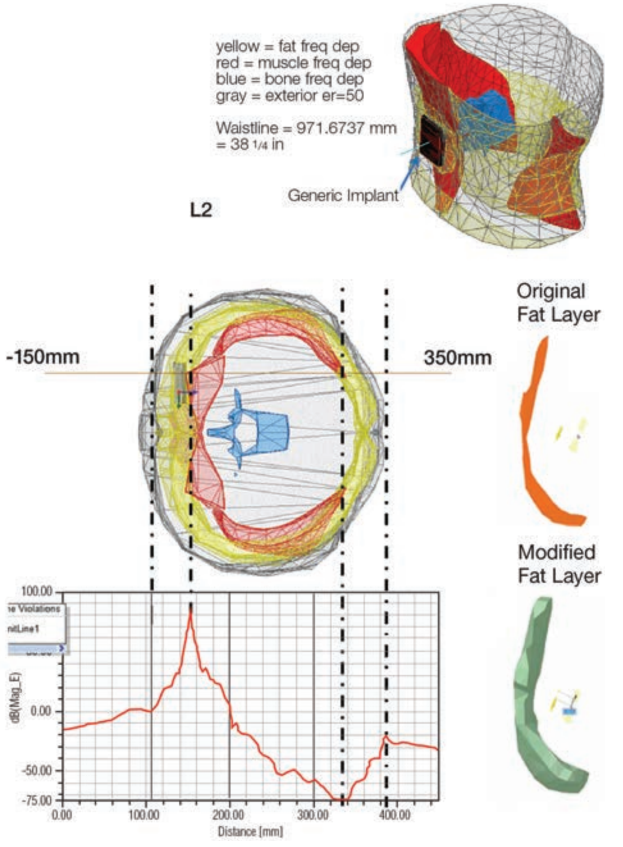


^ Model of the antenna for an implantable device

“Engineering simulation tools from ANSYS enable Cambridge Consultants to design innovative medical devices faster.”

which means that the body absorbs an even higher proportion of this energy, making the range problem even more difficult. The antenna may need tuning from time to time to accommodate patient physiology changes — for example, if the patient gains or loses weight. Finally, regulatory bodies place stringent restrictions on the radiated power, the specific absorption rate, and the rate and amount of data that can be transmitted over the air.

Cambridge Consultants, a world-class supplier of innovative product development engineering and technology consulting, uses ANSYS simulation tools to overcome these challenges. Simulation allows engineers to optimize the design of implanted device antennas to increase their range, enable them to operate at desired frequencies, and validate their performance in advance for a wide range of body types.



^ The ANSYS HFSS human body model was modified with ANSYS SpaceClaim Direct Modeler to represent different body types. ANSYS SpaceClaim can easily change the geometry of an object.

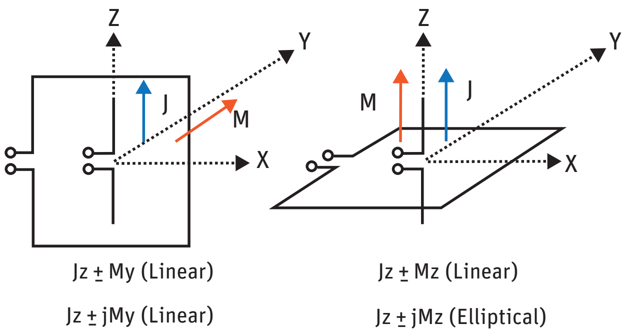
DESIGNING AN IMPLANTABLE ANTENNA

Cambridge Consultants’ engineers recently designed a small antenna that works on both the 402 to 405 MHz (medical implant communications service [MICS]) and 2.4 to 2.5 GHz (industrial, scientific and medical [ISM]) bands, and enables wireless communication at a range of 2 meters or more so that it can be used outside the sterile zone in the operating room. The capacitive nature of human tissue, along with the large capacitive reactance of traditional electric dipole antennas, produces a residual negative reactance that must be compensated with a lumped inductive load to match the microchip impedance. So engineers used a relatively new antenna design approach — a compound field antenna that employs both magnetic loop radiators and co-located electric field radiators. This approach provides an intrinsic inductive reactance that enables engineers to match impedance to the implanted electronics much more easily, and better supports miniaturization and biocompatibility.

Fat, muscle, various types of bone, skin and blood all have different dielectric properties. The dielectric properties of the surrounding tissue strongly affect the behavior of the antenna, for example, lowering the resonant frequency compared with the free-space performance of an antenna with the same dimensions. But the effect of the body on the antenna differs depending on the location of the antenna in the body and on the patient’s body type. Nearly all engineers who design antennas for implantable devices perform electromagnetic simulation using a human body model with elements designed to match the relative permittivity and conductivity of various body materials such as skin, fat, compact bone, spongy bone, muscle and blood. The problem with many of these models is that they are difficult to change to match different body types. So engineers usually optimize the antenna for one average body type, which often leads to antenna performance issues when the device is implanted into a patient with an atypical body type.

Internet of Things: Wearables and Medical Devices
ansys.com/wearables

> “Cambridge Consultants’ engineers use ANSYS SpaceClaim Direct Modeler software to rapidly modify the ANSYS HFSS human body model to represent changes in body morphology.”



Cambridge Consultants developed a compound field antenna that uses both magnetic loop radiators and co-located electric field radiators.

ANTENNA PERFORMANCE AND BODY WEIGHT CHANGE

Cambridge Consultants designs its antennas by simulating performance using ANSYS HFSS electromagnetic software with the HFSS human body model to represent the antenna’s use environment. Recognizing the critical importance of developing an antenna design that is robust to changing body morphology (weight), Cambridge Consultants’ engineers use ANSYS SpaceClaim Direct Modeler software to rapidly modify the HFSS human body model to represent changes in body morphology.

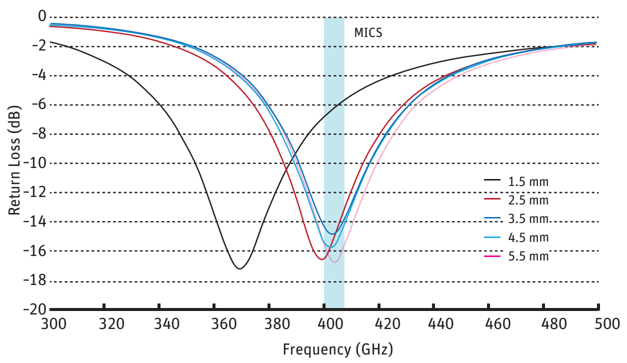
SpaceClaim enables users to create, edit and repair geometry without worrying about underlying technology, thereby speeding up time to analysis. Users can pull, move, fill and combine features of a model to, for example, create rounds, move a feature to another face or change the size of a face. If they prefer, users can enter explicit body dimensions.

This allows Cambridge Consultants to alter fat layer thickness and surrounding skin and muscle layer contours to scale a single body model. This capability is currently not available in other software packages in which a family of body types is provided rather than a single scalable body model.

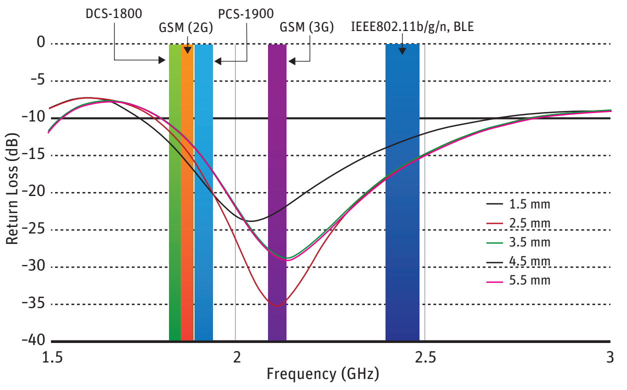
OPERATING IN TWO FREQUENCY BANDS

Nearly all modern wireless implants operate in the MICS band, but recently manufacturers are interested in developing devices that can operate in the ISM band. Using the ISM band can enable devices to communicate with smartphones, eliminating the need for a custom external communications component, and making it possible to take advantage of the powerful capabilities of smartphone technology. Cambridge Consultants engineers designed a new antenna from the ground up to work with both bands.

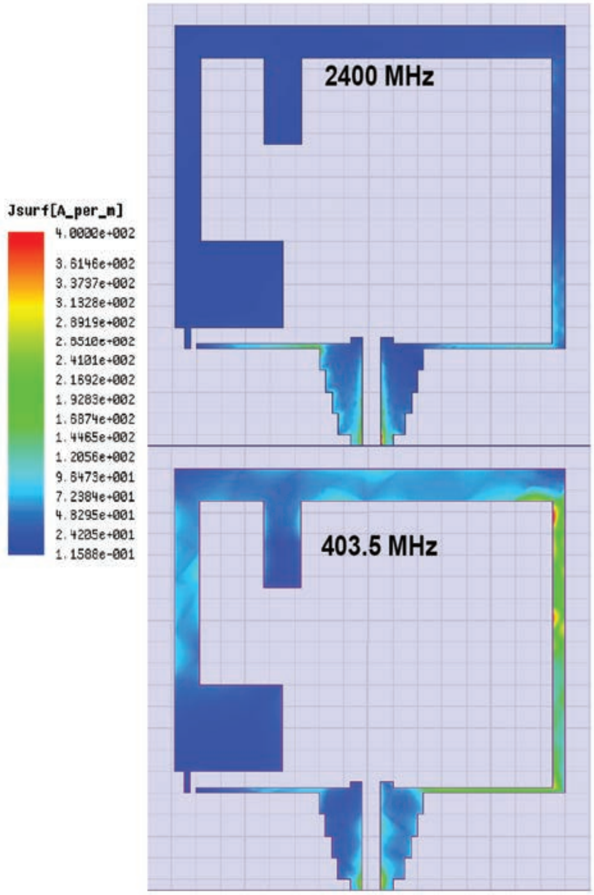
The dual band compound antenna topology was simulated on a 1.5 mm FR4 (glass-reinforced epoxy laminate printed circuit board) substrate with a 0.5 mm Al_2O_3 (aluminum oxide) substrate backing. The fat thickness in the HFSS human body model was varied between 1.5 mm and 5.5 mm in 0.5 mm steps. The antenna return loss was



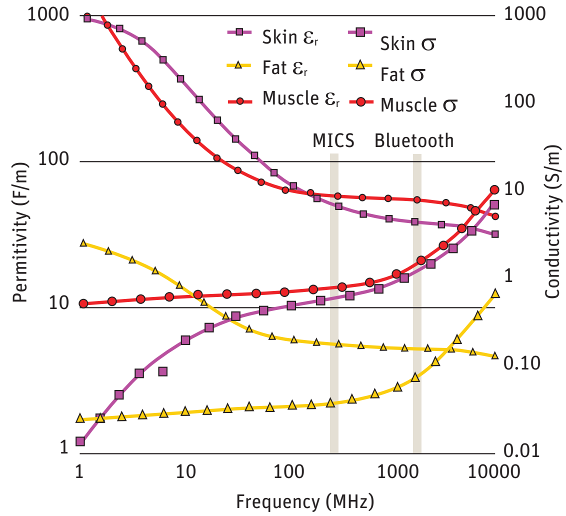
Frequency response of antenna in MICS band for different amounts of body fat



Frequency response of antenna in ISM band for different amounts of body fat



^ Surface current density for 2.4 GHz and 403.5 MHz, two of the bands in which the antenna must operate



^ Dielectric properties of fat, muscle and skin

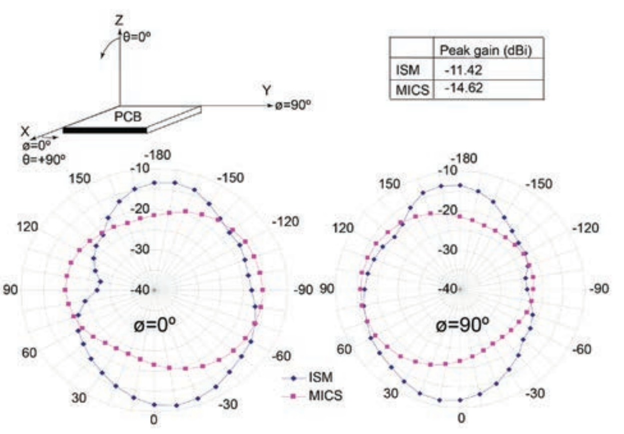
optimized for both frequency bands. The surface current density for the antenna structure was plotted for excitations at the two center design frequencies. The antenna demonstrated a peak gain of –11.42 dBi in the ISM band and –14.62 dBi in the MICS band.

The coupled electric and magnetic dipole antenna provides sufficient gain, radiation efficiency and broadband response in both the 402 to 405 MHz and 2.4 to 2.5 GHz bands in a wide range of body types and dimensions to enable the external communications component to operate outside the sterilized zone. The single planar structure is easily fabricated on a single 40 mm by 45 mm bilayer substrate (FR4/Al_2O_3). This antenna and others developed using similar simulation methods will help to improve healthcare by enabling the design of a new generation of medical devices that operate at a longer range to collect patient data for many body types.

DELIVERING INNOVATION FASTER

Modern implantable devices are very complex and require engineers to balance performance, safety, reliability, cost and time-to-market constraints. Engineering simulation tools from ANSYS enable Cambridge Consultants to design innovative medical devices faster.

Developing a scalable human body model helps Cambridge Consultant engineers to perform regression analysis on antenna designs right from the beginning of the design process. This halves the number of iterations needed and reduces the design time by 25 percent. The company has been able to increase the radio range for its novel antenna designs by 45 percent compared with traditional PIFA and loop antennas. Field data shows a very close correlation between the simulated and finished product results.



	Peak gain (dBi)
ISM	-11.42
MICS	-14.62

^ Antenna radiation patterns showing gain vs. radiation angle

Quantum Leap

By **Mark Nissen**,
Operations Manager, and
Sergey Uchaikin,
Senior Scientist,
D-Wave Systems,
Vancouver, Canada

Quantum computers take advantage of quantum mechanics to speed up certain important types of computing problems by many orders of magnitude. Harnessing quantum effects requires reducing the temperature of the processor to near absolute zero and providing shielding that reduces stray magnetic fields to 50,000 times less than Earth's magnetic field. D-Wave uses ANSYS electromagnetic and thermal simulation tools to achieve these goals in less time and with less physical testing.

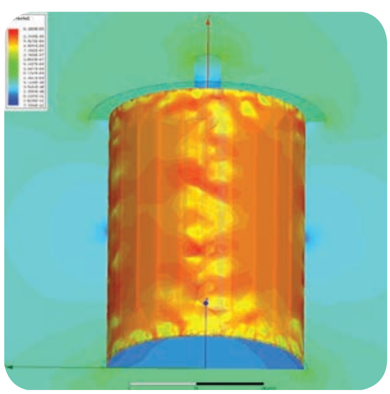
Market forecasts for the Internet of Things (IoT) predict that over a trillion sensors will soon be deployed around the world, collecting data. This data could be used to locate objects, understand and improve the performance of industrial assets, or support crucial research to prevent and cure diseases. For example, an Airbus 380-1000 series aircraft is expected to have more than 20,000 sensors generate more than 7.5 terabytes of data per day. By analyzing this data from hundreds of planes, airlines and their suppliers could improve the reliability and performance of the fleet.

> "Engineers use ***multiphysics simulation*** employing ANSYS Maxwell and ANSYS Mechanical within the ANSYS Workbench environment."

With the volume, velocity and variety of data rapidly increasing, quantum computing is a crucial technology for gleaning insights and driving outcomes. While conventional computers store information using bits representing 0s and 1s, in a quantum computer, the unit of information is called a qubit, which can be a 0 or a 1 or both at the same time. This enables quantum computers to consider and manipulate all combinations of bits simultaneously, making quantum computation extremely powerful. For example, the D-Wave 2X™ processor with 1,000 qubits can evaluate 21,000 possible solutions to a problem at the same time.

D-Wave quantum computers implement a quantum annealing algorithm that searches for the global minimum of a problem, and is designed to address a difficult class of computing problems, such as controlling risk in a financial portfolio, minimizing error in a voice recognition system, or reducing energy loss in an electrical grid. This type of problem can be thought of as trying to find the lowest point in a complex landscape consisting of many peaks and valleys. Every possible solution is mapped to coordinates on the landscape, and the altitude of the landscape is the energy or cost of the solution at that point. The aim is to find the lowest point or points on the map and read the coordinates, as this gives the lowest energy, or optimal solution to the problem. A classical computer is like a single traveler exploring the surface of a landscape one point at a time. A quantum computer performs in a way that is analogous to covering the entire landscape with a layer of water. The more water pooled in a particular valley, the higher the probability that the solution lies in that valley.

^ ANSYS Maxwell 2-D simulation identifies small amounts of leakage through shielding.



^ ANSYS Maxwell 3-D simulation of shielding shown in previous image

CHALLENGE OF CREATING ISOLATED ENVIRONMENT

For a quantum computer to function properly, the quantum processor must operate in an extremely cold and electromagnetically isolated environment. The D-Wave processor uses qubits based on the quantum unit of magnetic field, so eliminating all sources of magnetic fields is especially critical. Reducing the temperature of the quantum processor to near absolute zero is required to isolate it from its surroundings so that it can behave in a quantum manner. D-Wave quantum computers use a refrigerator and many layers of shielding to create an internal environment with a temperature close to absolute zero that is isolated from external magnetic fields, vibration and external RF signals of any kind. The D-Wave 2X processor operates at a temperature of 15 millikelvin, which is approximately 180 times colder than interstellar space. The quantum processor is adversely affected by stray magnetic fields, so extreme care must be taken to exclude them. The magnetic shielding subsystem achieves fields of less than 1 nanotesla across the processor in each axis. This is approximately 50,000 times less than the Earth's magnetic field.

The conditions required for optimal quantum computing performance are greater than those that have been required by previous generations of supercomputers, and they increase with each new product generation. Because of this, D-Wave continually pushes the state of the art in both low-temperature and electromagnetic

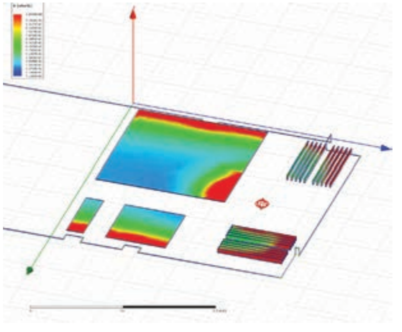
> **"For a quantum computer to *function properly*, the quantum processor must operate in an extremely cold and *electromagnetically isolated* environment."**

isolation technology. This task is complicated by the fact that changes in temperature change the properties of the shield, while the minute magnetic fields that penetrate the shielding produce heat that in turn further changes the properties of the shield. In the past, D-Wave used handbook calculations as a starting point in the design of magnetic shielding for quantum computers. The calculations are only accurate for very simple geometries, so engineers primarily relied on physical experiments to design shielding. These experiments were expensive and time-consuming because most of them needed to be performed in cryogenic conditions. Another limitation of the physical experiments is that only a few sensors fit inside the shielding, limiting the amount of information that can be obtained.

SIMULATION TOOLS EXCHANGE DATA SEAMLESSLY

D-Wave evaluated a number of different simulation tools that don't communicate well with each other. Today, engineers use ANSYS Maxwell and ANSYS Mechanical within the ANSYS Workbench environment to automate the process of exchanging data between the two software packages for multiphysics simulations. D-Wave scientists employ Maxwell to model the operation of the company's shields and the tiny residual magnetic fields to which the company's chips are exposed. This information is passed to ANSYS Mechanical to simulate the effects of these fields on the shielding, chips and other components; to calculate the heat that is generated by the magnetic fields; and to determine the effect of the resulting temperature change on the properties of the materials being simulated. ANSYS Mechanical passes this information to Maxwell, which updates the electromagnetic simulation to take the materials' property changes into account.

ANSYS Maxwell
ansys.com/maxwell



^ ANSYS Maxwell simulation of a magnetic impurity located on a D-Wave DW2X chip near the active area of the processor

OPTIMIZING THE MAGNETIC SHIELDING

In the design of a typical shield, D-Wave engineers evaluate the performance of the shielding with respect to distant magnetic fields, such as the Earth's magnetic field, and fields nearby, which could be caused by very small amounts of magnetic material. Engineers often create their own materials by entering magnetic hysteresis curves, also known as B-H curves, obtained from physical testing data. Generating the mesh for electromagnetic simulation of the shielding is challenging because the difference in scale between the large features and the small features of the shield typically is five orders of magnitude. D-Wave scientists have had difficulty in the past obtaining convergence with other electromagnetic solvers. The Maxwell mesher automatically increases the density of the mesh in areas with high gradients while reducing density in areas with low gradients so convergence is easily achieved.

The Maxwell simulation identifies any weak points of the shield design, such as areas where it is penetrated by an external magnetic field. The simulation also shows the effects of the shielding on the external magnetic field. Based on the simulation results, D-Wave engineers typically change the shield design, for example by increasing its thickness in areas where penetration is seen. Engineers also try to minimize the mass of the shield because extra mass increases the cost and time required to cool the shield to cryogenic temperatures. They are frequently able to reduce the mass by removing material from areas of the shield where performance is better than necessary.

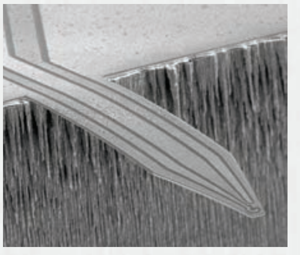
Simulation helps engineers and scientists diagnose proposed isolation technology designs and evaluate ideas for improvements without the need for expensive physical prototyping and testing. Simulation also provides more comprehensive measurement of shield leakage to simplify the process of developing compensating devices. ANSYS solutions help D-Wave optimize today's quantum computers at a faster pace while providing insight that will make it possible to achieve huge leaps in performance in the next generation. And this next generation of computing is vital to enable organizations to glean value from the vast amounts of data generated by the IoT. ^

Making Sensors for the IOT

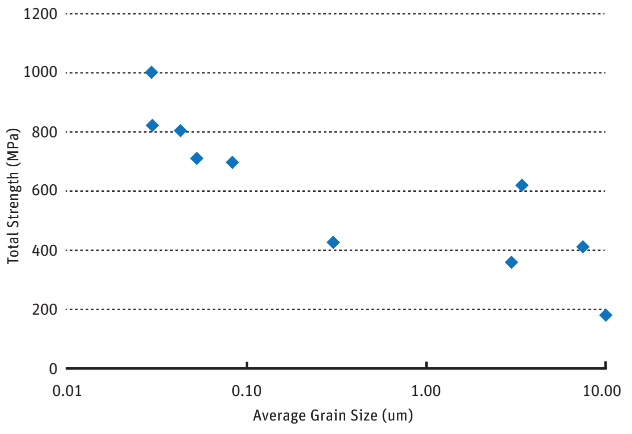
MEMS technology is a critical element in many of the sensors that will generate the data to drive value from the IoT. An experienced MEMS developer describes some of the issues involved in creating reliable MEMS and provides some best simulation practices to assist in their design.

By **Alissa Fitzgerald**, Founder and Managing Member, A.M. Fitzgerald & Associates, LLC, Burlingame, USA

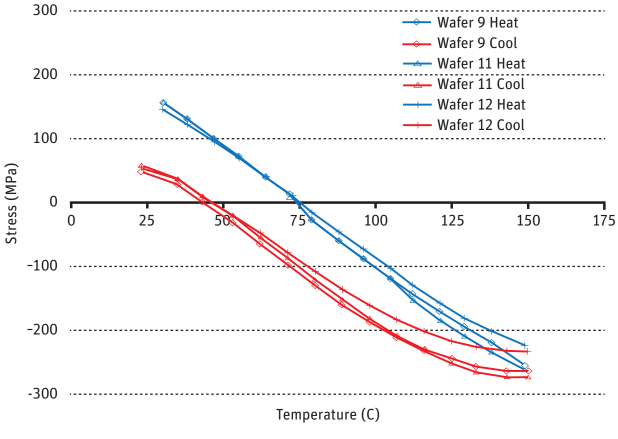
Microelectromechanical systems (MEMS) is a manufacturing technology that can be used to create many different types of sensors — temperature, motion, pressure, sound, etc.— on silicon wafers. MEMS sensors serve as the eyes and ears of today's smart connected products by acquiring information from the environment, such as the air pressure of an automobile tire or the motion of your body to record your steps. It is anticipated that MEMS sensors will experience a rapid growth curve as the Internet of Things (IoT) makes it possible to capture the information from billions of MEMS sensors and



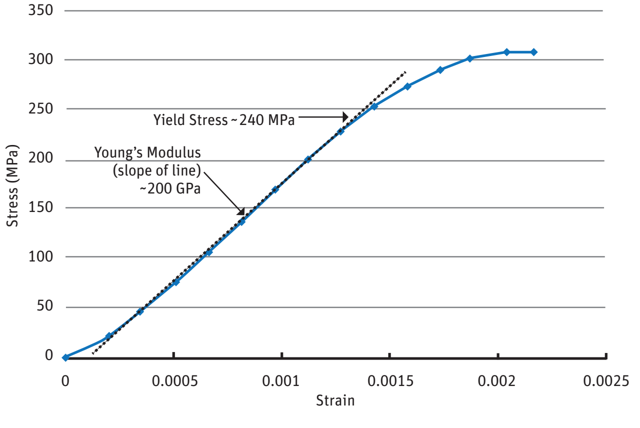
^ A MEMS cantilever sensor designed and prototyped by AMFitzgerald



^ MEMS material properties are recipe- and tool-dependent.



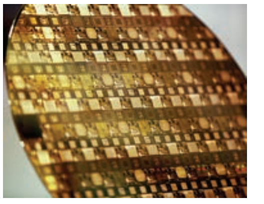
^ Thermal cycling caused a 100 MPa change in residual stress due to plastic deformation of the metal film.



^ Modulus can be estimated from wafer-level film stress data over a temperature range.

utilize this data to intelligently control devices to improve efficiency, quality, health, safety and the environment.

But despite the fact that MEMS are made on a silicon wafer, MEMS development is considerably less rapid and more difficult than conventional integrated circuits (ICs), largely due to a lack of established design practices and the limitations of some current simulation methods. Fortunately, it's possible to work around these challenges using practices such as parametric analysis to understand the impact of uncertain material properties, and to integrate test results with simulation to calibrate boundary conditions. Using these and other techniques makes it possible to achieve useful simulation results that can reduce the number of fabrication (fab) rounds and bring MEMS devices to market in considerably less time and expense than is required using the traditional build and test approach.



MEMS DEVELOPMENT CHALLENGES

The semiconductor manufacturing industry has developed process design kits (PDKs) consisting of standard cell libraries, design rules, simulation models and layout information for producing ICs with a specific technology in a particular foundry. The industry has also benefited from a wide range of well-validated design and process simulation tools. As a result, a company bringing an IC to market can be fairly certain that, if they follow the PDK guidelines and simulate early and often, the resulting device will perform as expected and be manufactured without major difficulty.

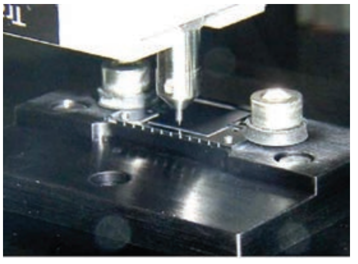
But the wide variation in geometry and materials among MEMS devices has, so far at least, forestalled the development of PDKs and prevented simulation from reaching the same level of maturity as it has for ICs. The result is that developers of MEMS devices are never quite sure what they are going to get until they receive samples from the foundry. Unfortunately, the minimum cost for a development batch of ten 150 mm wafers, including recurring costs only, is well over $100,000, and the lead time is a minimum of eight to 12 weeks. Escalating, unknowable development costs have led to the premature closure of some MEMS companies, and to investor caution about the entire industry.

One of the biggest obstacles in simulating MEMS is determining the material properties of thin films. The tensile strength of a thin film is highly dependent on the deposition recipe, and can even vary widely among different manufacturing tools using the same recipe. Another challenge with thin films is that metals and dielectrics are generally deposited onto silicon at a very high temperature. In regions where the deposition layer and substrate have different coefficients of thermal expansion, residual stresses are generated in the film when the wafer cools back to ambient temperature. This residual stress often varies at different steps in the process as the wafer undergoes temperature changes. It can result in bowing, buckling or cracking of the MEMS structures, so this stress needs to be accommodated within the simulation model.

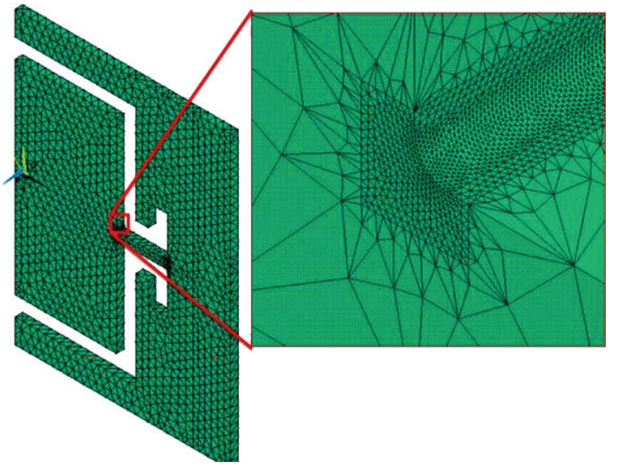
Simulation Reduces Uncertainty and Risks in MEMS Design and Manufacturing
ansys.com/sensor

SIMULATION OVERCOMES MATERIAL PROPERTIES UNCERTAINTIES

Uncertainties in material properties can be addressed by using parametric simulation to determine the effects of material property variability on device performance. Tools within ANSYS Mechanical software allow engineers to quickly evaluate the effects of varying material properties, along with design parameters, to save spins in the fab. A typical approach is to start with the textbook material properties, then vary the geometry of the structure while holding the elastic modulus constant. The next step is to hold the geometry constant and vary the modulus. Simulation helps you to make smart design choices that reduce sensitivity to material properties. If you find that your design is overly sensitive to material properties, you can perform wafer-level and device-level measurements to gather empirical data to improve simulation accuracy. For example, a KLA-Tencor Flexus stress measurement system can scan the wafer to determine thin film stress as a function of temperature, which in turn can be used to estimate modulus.

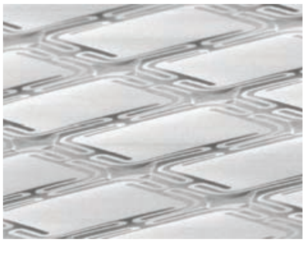


^ Measurements showed that the original MEMS model's boundary conditions were too stiff.



^ Force-displacement measurements were used to correct the finite element model.

While ductile materials such as metals fail at a well-defined limit, predicting the load limit of brittle materials used in MEMS is more difficult. Crystalline microstructures fail at surfaces, so their strength is a function of the size and location of the surface flaws that are created during the etching process. Details such as etch tool type and operating parameters can lead to significant differences in surface strength and device fracture probability. AMFitzgerald invented a method to estimate device fracture probability that uses empirical data and custom ANSYS APDL scripts. The solution is to first build test structures using different process parameters to statistically characterize the influence of the manufacturing process on surface strength. Next, create a finite element model of the proposed or existing microstructure in ANSYS Mechanical and simulate the stresses in the microstructure device under applied load. Finally, in the post-processor, combine the results of the load simulation with surface strength information to predict failure locations and load limits. Based on this information, engineers can revise the device design or process, or proceed directly to manufacturing.



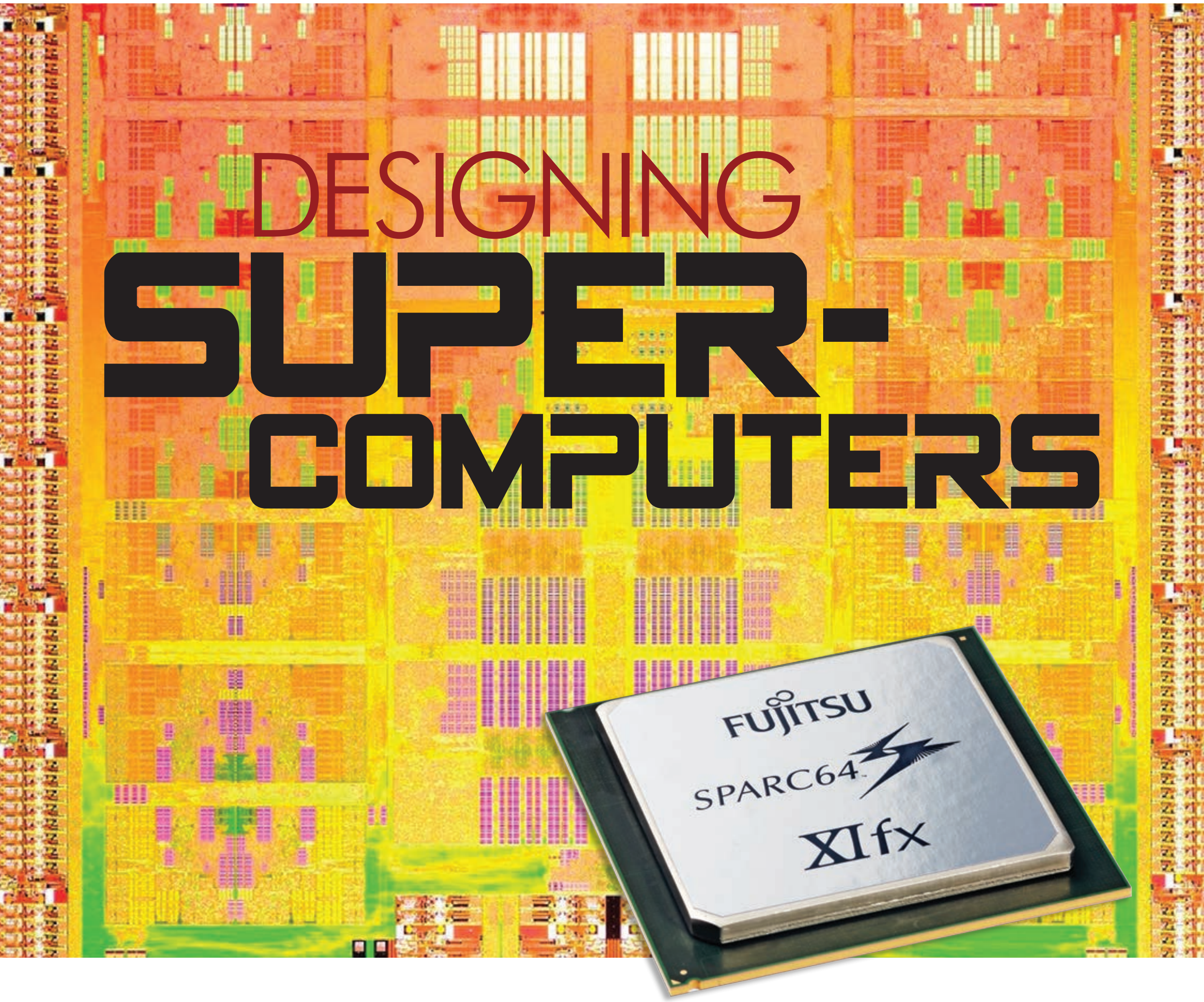
> "Simulation can *save* ***hundreds of thousands*** **of dollars in fabrication and testing and months of** ***redesign time.***"

SIMULATION AND PHYSICAL TEST INTEGRATION HELPS UNDERSTAND BOUNDARY CONDITIONS

Obtaining accurate results from simulation requires using accurate model boundary conditions. For a thin film structure deposited onto a silicon substrate, an analogous macro-level structure would use a fixed boundary condition at the film–substrate interface. However, thin film structures are not as stiff as macro structures. The solution is to perform force versus displacement measurements to determine the stiffness of the structure, which can then be fed back into the model.

Simulation of MEMS is much more challenging than ICs because their material properties and boundary conditions are not nearly as well defined. With some effort, you can work around these challenges and reap the benefits of simulation. Simulation is much less expensive than processing test wafers in the fab, so you will be able to evaluate many more design and manufacturing options early in the design process and bring better products to market in less time. Simulation can easily save hundreds of thousands of dollars in fabrication and testing and months of redesign time. Investing in the development of accurate models for your technology will always benefit your future products.

DESIGNING SUPER-COMPUTERS



One of the greatest challenges in designing fast, data-intensive supercomputers is providing power to and removing heat from hundreds of thousands of computing cores. Fujitsu uses the ANSYS power and thermal toolset to simulate its next-generation 3-D IC semiconductor designs, starting before the systems have been fully defined until final signoff. The result is higher performance and fewer design turns.

By **Koichi Yoshimi** and **Hironori Kawaminami**, Fujitsu Limited, Japan, and **Norman Chang,** Vice President and Senior Product Strategist, Semiconductor Business Unit, ANSYS

Positioned at the forefront of the supercomputing space with 30 years' experience in the successful development of high-performance systems, Fujitsu cooperates with leading organizations to apply supercomputing to address increasingly complex social, environmental and business challenges. Fujitsu co-developed the K supercomputer with RIKEN, which was first on the TOP500 ranking of supercomputers in 2011.

The K supercomputer again took first place in the 2015 Graph 500 supercomputer ranking, which gauges the ability of supercomputers to handle complex data problems in areas such as cybersecurity, medical informatics, data enrichment, social networks, symbolic networks and modeling neuronal circuits in the brain. The same technology used in the K supercomputer is used in Fujitsu's commercial supercomputer products PRIMEHPC FX10™ and FX100™.

Fujitsu supercomputers pack an enormous number of compute cores—the FX100 scales to over 100,000 nodes (a node consists of all cores that are connected to a common memory source). But to meet the ever-increasing demand for computing power, Fujitsu needs to pack more processors into a smaller footprint with lower power consumption. Traditionally, Fujitsu uses bleeding-edge semiconductor processes to gain more performance, but the end of semiconductor scaling is on the horizon.

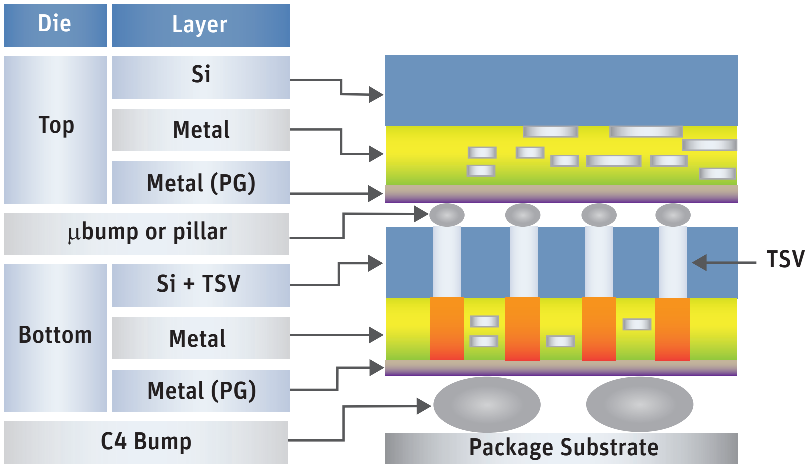
Several promising technologies have the potential to overcome this barrier, including non-Si materials, non-Neumann architecture and 3-D IC structure. Fujitsu is currently studying the use of a 3-D IC structure as a good candidate for achieving the required power, performance and footprint goals. A 3-D IC structure will provide faster cycle time and lower power consumption by improving circuit density and minimizing wire length without requiring further process scaling. But the use of a 3-D IC structure also comes with big challenges, including power integrity, cooling, signal integrity and the most important: cost.

PRIMEHPC FX100™

3-D IC DESIGN CHALLENGES

In the past, engineers used system-level thermal analysis to predict a uniform temperature for the entire chip. They were forced to use a high margin of safety to account for the thermal gradients on the real chip, which limited the performance improvements that could be achieved. Lack of knowledge of thermal gradients also made it impossible to accurately determine the resistance (R) and electromigration (EM) limits of individual wires in the chip because R and EM limits are highly dependent on temperature. Thus, engineers could not accurately calculate the IR (voltage) drop and EM in the voltage flowing over each wire. IR/EM is a key factor in determining power integrity — the ability to supply power to each complementary metal oxide semiconductor (CMOS) device on the chip. Engineers also had no way to predict the effects of other chip structures, such as through-silicon vias (TSVs) and µBumps, which also have a major impact on power, thermal and signal integrity.

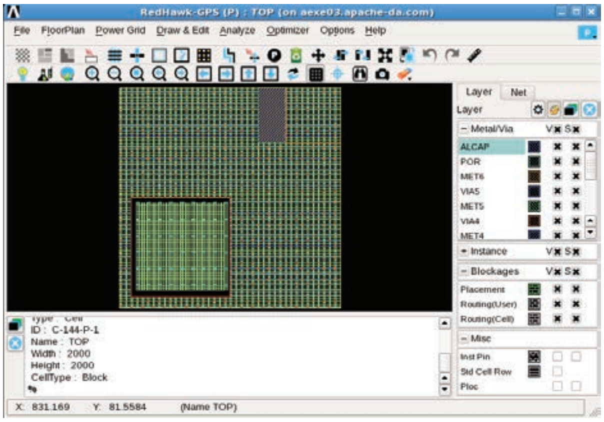
Now, Fujitsu engineers use ANSYS RedHawk to simplify chip design by dividing each layer into rectangular elements comprising a chip thermal model (CTM) that contains information about the temperature dependent power and metal layer density based on the detailed design (if it is available) or a previous design (if it is not). The CTM also contains information about thermal conductivity between layers. The TSV layout is defined in RedHawk-GPS for early power and ground network construction, including TSV placement. This model calculates the power distribution network and temperature profile across each individual chip.



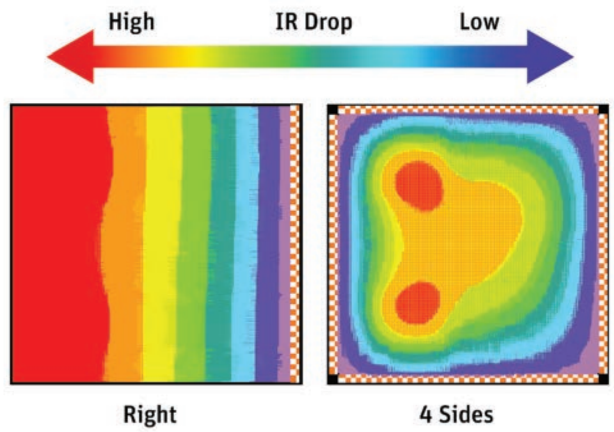
3-D model of typical 3-D IC

DETERMINING EFFECTS OF NUMBER AND PLACEMENT OF TSVS ON IR/EM

TSV positioning also has a major impact on power and thermal integrity, both of which need to be considered in the early stages of the design process. Fujitsu engineers use RedHawk, which allows them to explore a large design space — possible placements of TSVs, silicon interposer structure, effective power delivery networking, redistribution layer wirings and others — in the early stages of the design process. The packaging details are usually not available at this early stage, but ANSYS Sentinel-TI empowers engineers to generate a package thermal model based on a few parameters, such as the package size, CTMs of each die, position of the µbumps and C4 bumps, etc. Sentinel-TI performs a package-scale



^ P/G/TSV construction and chip thermal model generation in ANSYS RedHawk-GPS



^ IR drop for two different TSV placement cases: (left) TSVs all on right side, (right) TSVs on all four sides. IR drop is substantially lower when TSVs are placed on four sides.

power and thermal analysis, and generates power and thermal maps of the entire system. System power and thermal boundary conditions are fed back to RedHawk, and the thermal-aware IR/EM analysis for each chip is performed again to take the effects of the other chips and package into account.

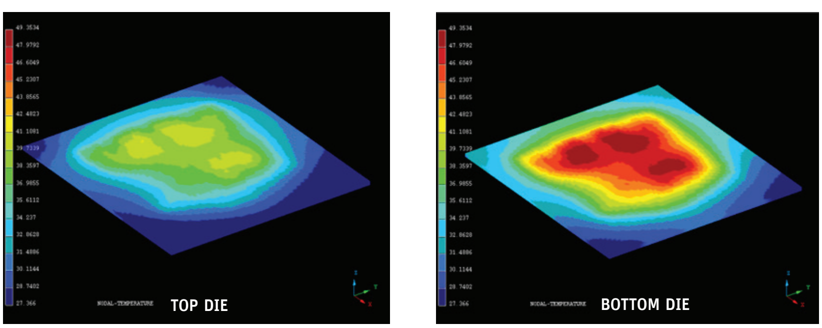
EVALUATING EARLY TSV PLACEMENT

Fujitsu evaluated early TSV placement for a next-generation design for which detailed design information was not readily available. Therefore, the previous generation of the design was used for the evaluation. They assigned power targets to each logic region to account for the higher density of the next-generation design. In one of the studies, two different TSV placements were considered — one solely on the right side of the logic area and the other throughout the logic area. In addition to the analyses described above, an IR drop analysis was performed using total power dissipation to calculate a constant current draw, which was then multiplied by the equivalent resistance of the power distribution network to arrive at the voltage drop. The results showed that IR drop and temperature were much higher on the left side of the chip that had the TSVs only on the right side of the logic area. Then more TSVs were added to the design on four sides, and the corresponding reduction in static IR drop and temperature across the chip were tracked.

The same model was updated throughout the design process and used to investigate effects of changes in chip design, TSV placement, bump placement and packaging design. This allowed engineers to maintain power-, thermal- and signal-integrity not only during the early stages of the design process but all the way through to sign-off. The ability to evaluate thermal-, as well as power- and signal-integrity, made it possible to improve static IR drops by 62 percent and dynamic drops by 15 percent while checking thermal integrity at the same time. This workflow, which ensures that engineers get power-, thermal- and signal-integrity right the first time, reduces expensive design turns.

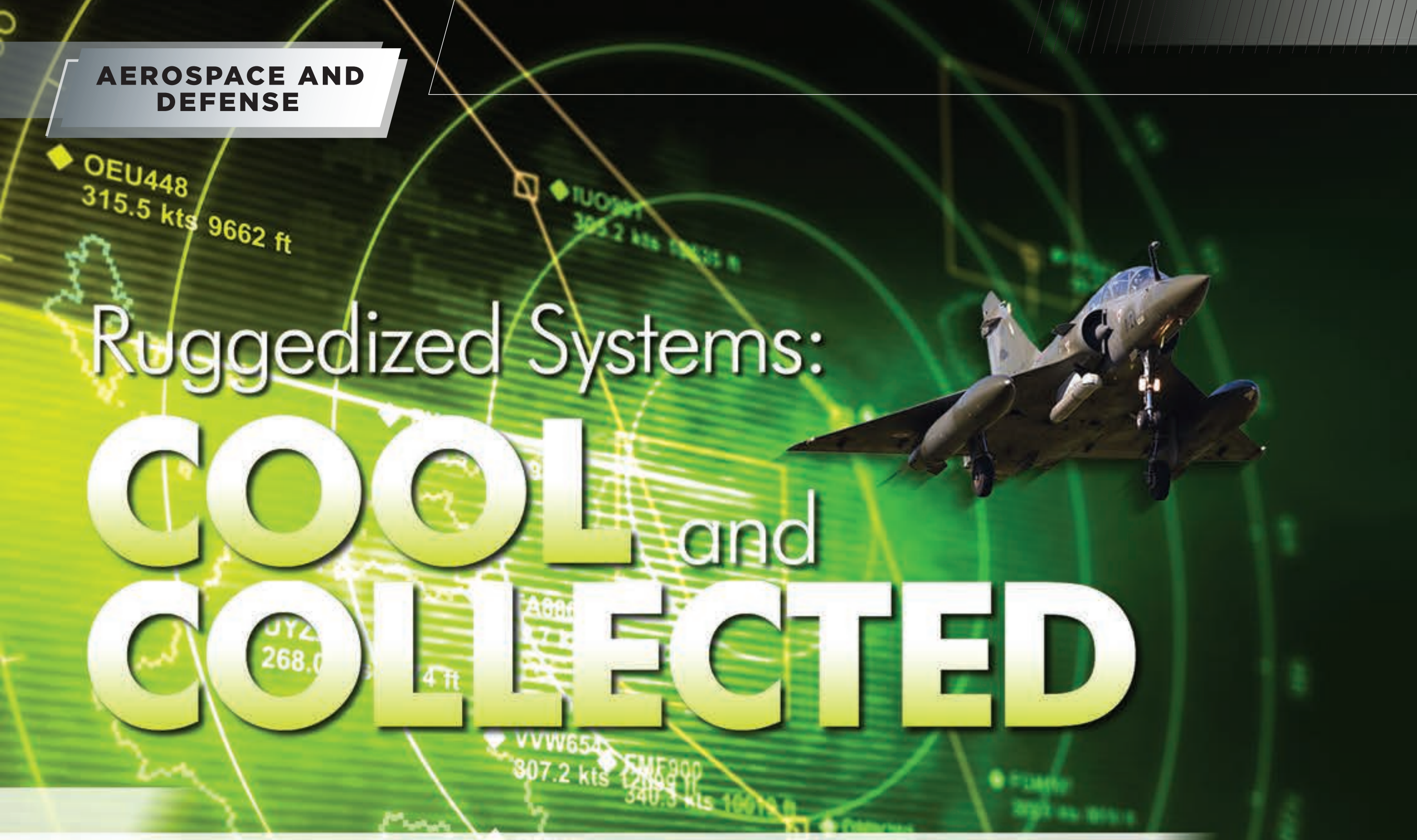
FOR FUTURE 3-D IC DEVELOPMENT

Going forward, for 3-D ICs it will be even more important to verify the integrity of power, thermal, timing and cost than in the past. ANSYS RedHawk and Sentinel enable Fujitsu engineers to perform many design-space explorations of a 3-D IC from the early design stages. This will significantly shift the design effort to the left, effectively reducing the non-recursive design cost for future 3-D IC development.



^ Temperature of top and bottom die in a case in which TSVs are placed on four sides. The bottom die is hotter because the heat sink is on top, but the bottom die is still within the acceptable range.

ANSYS RedHawk-GPS
ansys.com/redhawk



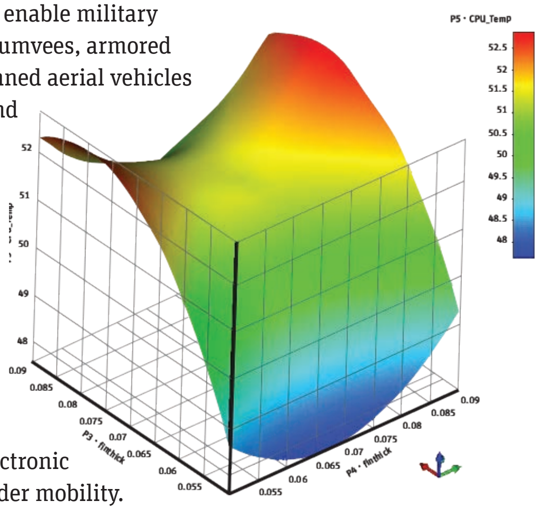
Ruggedized Systems: COOL and COLLECTED

To meet demanding military specifications for mobile and interconnected surveillance, communication and operational devices, Kontron uses sophisticated thermal simulation to balance size, weight, power and cooling (SWAP-C) trade-offs for “ruggedized” modular chassis that support customized solutions for mission-critical operations.

By **Simon Parrett**, Conceptual/Structural/Thermal Engineer, Kontron, Poway, USA

Today’s military vehicles depend on state-of-the-art visualization, imaging and networking technologies to improve situational awareness and enable military leaders to make the best possible decisions. Vehicles such as Humvees, armored mine-resistant ambush protected vehicles (MRAPs) and unmanned aerial vehicles (UAVs) increasingly rely on advanced electronics, such as processors and circuitry, in compact systems to support their missions.

To satisfy the military’s demand for these electronic systems that can be adapted to a range of uses, defense contractors must meet a host of requirements and specifications. The devices placed on vehicles, such as battlefield sensor systems, military GPS and next-generation communications equipment, must be able to communicate and interact in extreme physical environments where they might be exposed to severe electromagnetic conditions. Military standards require that these devices withstand specified extremes of temperature, vibration, shock, salt spray, sand and chemical exposure. Size, weight, power and cooling (SWAP-C) requirements demand that the electronic systems that power these devices be small enough that they do not hinder mobility.



^ Parametric optimization of enclosure cooling fins

The approach that has proven most effective is to contain the electronic system functionality in a chassis that has been precertified for “ruggedized” operation. Using this chassis, designers can ensure that the system is maintained in a sealed and temperature-controlled environment. To design these ruggedized systems, Kontron, a global leader in embedded computer technology and an IoT leader, uses sophisticated computational

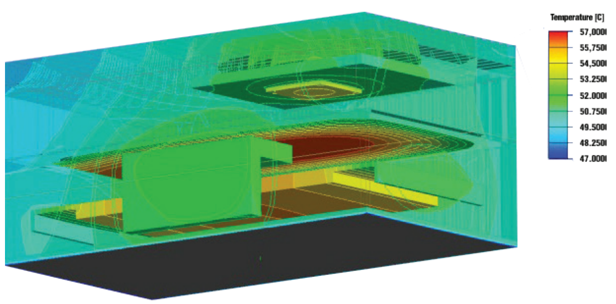
"Defense contractors must meet a host of requirements and specifications to satisfy the military's demand for flexible electronic systems."

fluid dynamics (CFD) analysis to accurately manage thermal reliability for components and ultimately the complete integrated system. The chassis they provide enables original equipment manufacturers to build customized solutions for mission-critical applications.

RUGGEDIZED SYSTEMS

Kontron's COBALT line of computing platforms uses a modular approach to deliver a rugged, sealed computing system with a specialized carrier board and configurable front panel that can be integrated into the electronics bay of a Humvee, MRAP-type vehicle or UAV. The box-level system provides processing power to enable third-party developers to maintain flexibility, compatibility and interoperability for many types of rugged applications. Using standard interfaces reduces long-term costs and makes it easy to upgrade, replace and reuse capabilities across systems. Hundreds of these systems can be fitted onto a single aircraft or ground vehicle.

To develop a truly flexible system, Kontron must take many variables into account and identify trade-offs. Surveillance applications, for example, require high I/O and fast processing speeds. They also require low signal bandwidth for communications efficiency, and reliable wireless communication to send information back to data centers. For these applications, customers want a chassis that can ensure that powerful processors or other components do not impair the radio signal. Power consumption and thermal management are also important; the heat from a processor can impede the performance of other components, and thermal cycling stresses as the processor heats and cools, especially in conditions such as extreme desert heat or the cold of high altitudes, can cause fatigue in components and the chassis. As systems become more complex and are required to incorporate more capabilities, managing SWAP-C requirements is even more critical, and design priorities depend upon the size of the vehicle, the nature of the applications, and the missions for which the vehicles are employed.



^ System-level thermal trade-off analysis, used to build the Excel product thermal configurator

CFD ANALYSIS FOR THERMAL MANAGEMENT AND RELIABILITY

To develop these chassis, designers of the COBALT product line have adopted a "five-gate" process of sign-off procedures, from loose specification (Gate 1), through various iterations, to a finished product (Gate 5). Typically, they introduce ANSYS analysis at Gate 1 to anticipate problems and trade-offs early in the design phase, leading to more complex products in a shorter design frame. The team uses ANSYS DesignModeler to import geometries, ANSYS Icepak to determine temperatures, ANSYS DesignXplorer for design exploration, and ANSYS HPC for faster results. ANSYS Workbench provides the common environment to integrate the simulation process.

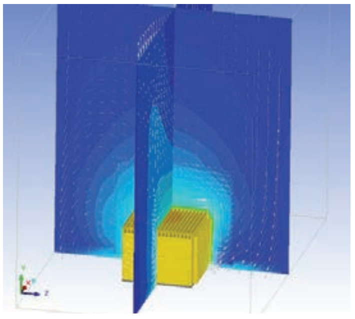
The Kontron design team uses CFD analysis to evaluate and optimize chassis thermal performance.
Some key activities are:

- Designing the enclosure to draw as much heat as possible from the circuit board and processor. The team uses ANSYS Icepak to streamline CFD analysis to design finned surfaces and heat sinks, and arrive at an optimal design.

Conceptual CAD rendering of the Kontron COBALT (computer brick alternative)

- Determining placement of electronic components and subsystems within the chassis and balancing the trade-offs necessary to meet SWAP-C requirements. For example, engineers analyze the power dissipated by an expansion board and its effect on the temperature of a nearby processor.
- Reviewing internal thermal conduction paths from high-power components to ensure that there are efficient paths to the enclosure walls.
- Exploring external environmental factors in situations where the full system will be deployed. If the system chassis is deployed in a UAV, for example, the cooler temperatures and thinner air in high altitudes will affect thermal management. Another factor might be the location of the chassis in the vehicle. If additional chassis are located nearby, heat and radiation exchange need to be considered.

Besides the early focus on optimal design for SWAP-C considerations, Kontron designers are also concerned about longevity. When the chassis is added to a ground vehicle or plane, it's expected to last three to five years, plus another two years with maintenance. The mean time between failures (MTBF) is very important to their customers.



Initial natural convection cooling assessment

EVALUATING DESIGN TRADE-OFFS

Recently, the design team introduced a new gate, Gate Zero, wherein they talk to customers and work with product managers to get new ideas for their products. This enables the team to create "what-if" scenarios even before they write the specifications. To test the Gate Zero concept, Kontron engineers modeled a sample heatsink using rough designs in Icepak and tested various configurations to determine what trade-offs would be required.

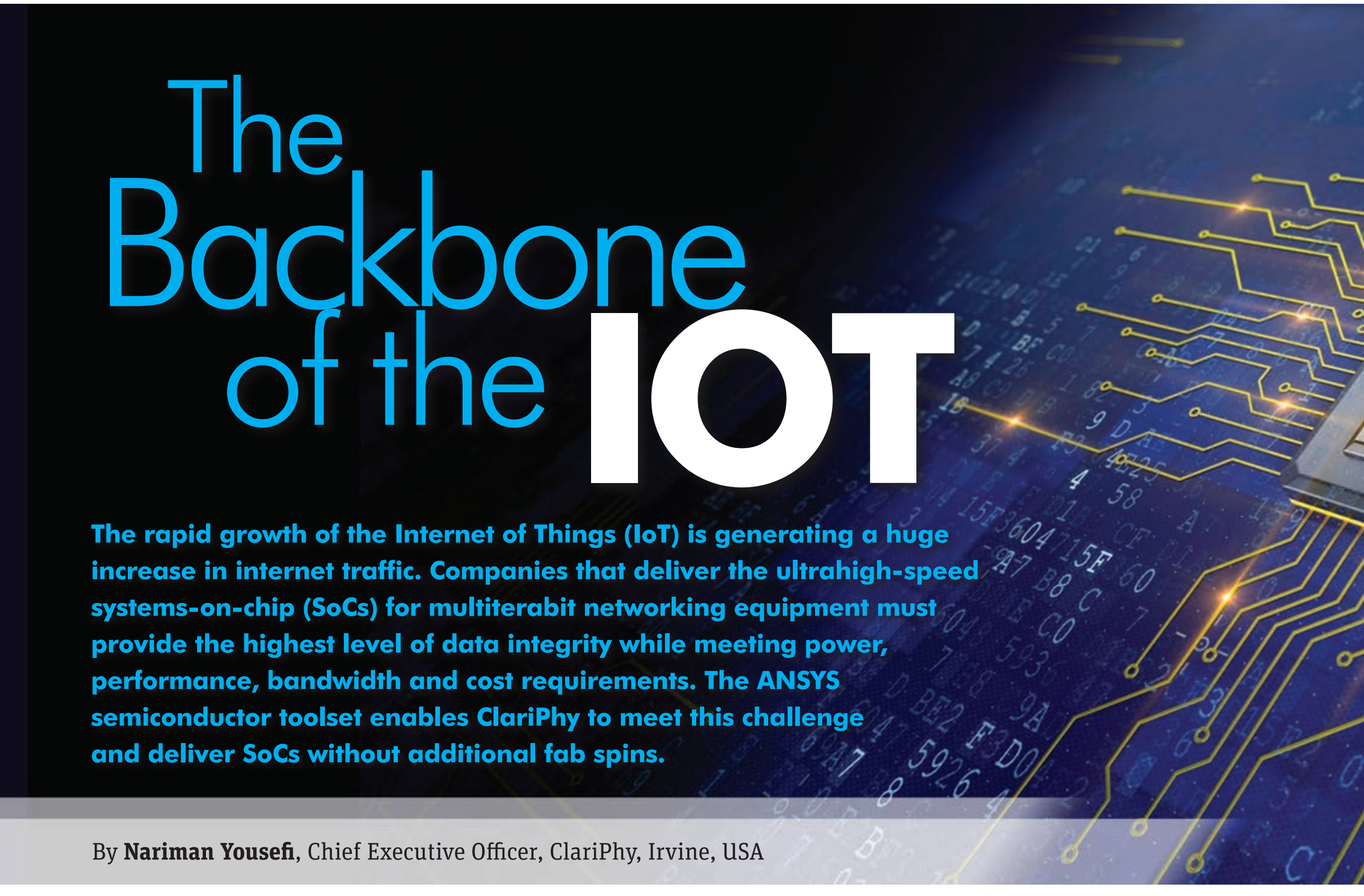
In the past, they would analyze thermal problems by running an initial analysis, trying some manual design variations, and after seven or eight design iterations that included physical mockups, perform a final analysis and publish their results. Using DesignXplorer to drive Icepak, the team was able to exceed those limitations, identifying 240 potential design variations to test. The software then used mathematical models to narrow down the list to just 70 essential variations for further study. By running 70 intelligent design iterations over a weekend, engineers were able to evaluate 10x more design variables than was possible with the old methodology in the same amount of time. The designers were presented with three optimal design candidates to choose from.

From the large design space that was explored using simulation and driven by DesignXplorer, the Kontron team developed a chassis configuration tool with an Excel® interface that their sales team can use in customer meetings to rapidly design a chassis customized to client requirements.

Starting with a baseline configuration with the desired maximum ambient temperature, application engineers add design variables, such as CPU max power or electronic expansion trays; operating parameters, such as the orientation and position of the device; and the altitude where it will be used. The spreadsheet shows the power consumption of each component in the box and how their interaction affects the temperature within the box. They can also plot out remediation options, such as extending the size of the heat sinks, to calculate their effect on the temperature. The spreadsheet can also be used to factor in the cost of changes, for example, the cost of adding a heatsink based on the number of fins and their thickness. Using the inputs and relationships they have learned using ANSYS software enables them to better inform their customers so that they can find the best configuration together.

With the ability to increase virtual tests by a full order of magnitude in less time, Kontron can avoid potential problems, adapt to their customers' needs, and provide rugged, reliable systems for the connected army of the present and the future.

ANSYS Icepak
ansys.com/icepak



The Backbone of the IOT

The rapid growth of the Internet of Things (IoT) is generating a huge increase in internet traffic. Companies that deliver the ultrahigh-speed systems-on-chip (SoCs) for multiterabit networking equipment must provide the highest level of data integrity while meeting power, performance, bandwidth and cost requirements. The ANSYS semiconductor toolset enables ClariPhy to meet this challenge and deliver SoCs without additional fab spins.

By **Nariman Yousefi**, Chief Executive Officer, ClariPhy, Irvine, USA

As the Internet of Things (IoT) grows from six billion connected devices today to an estimated 20 billion by 2020, global data traffic is expected to increase nearly five-fold, passing a new milestone figure of two zettabytes by 2019. [1] (One zettabyte is equal to a billion terabytes.) ClariPhy engineers are delivering next-generation communication architectures fabricated on the most advanced technology process nodes to enable the fabric of networks across the globe to handle rapid increases in data traffic without compromising quality of service. For these state-of-the-art designs to meet stringent performance and quality (reliability) requirements, the power delivered to the transistors inside these ultralarge system-on-chip (SoC) designs has to be robust across the chip and across all operating conditions. To meet these specifications, the global power distribution network (PDN) must be constructed with awareness of requirements on the chip (e.g., voltage drop, routing resources), the package (e.g., package plane, decaps) and the PCB (e.g., voltage regulator module placement).

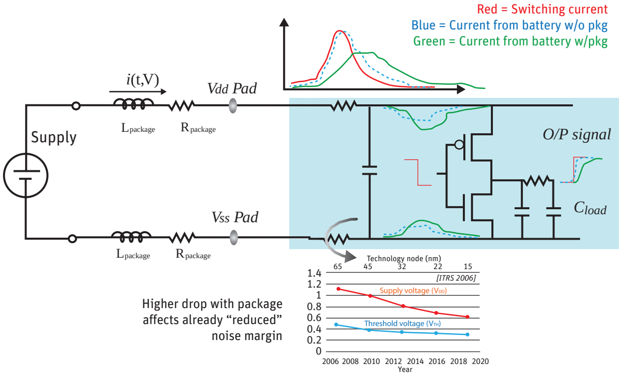
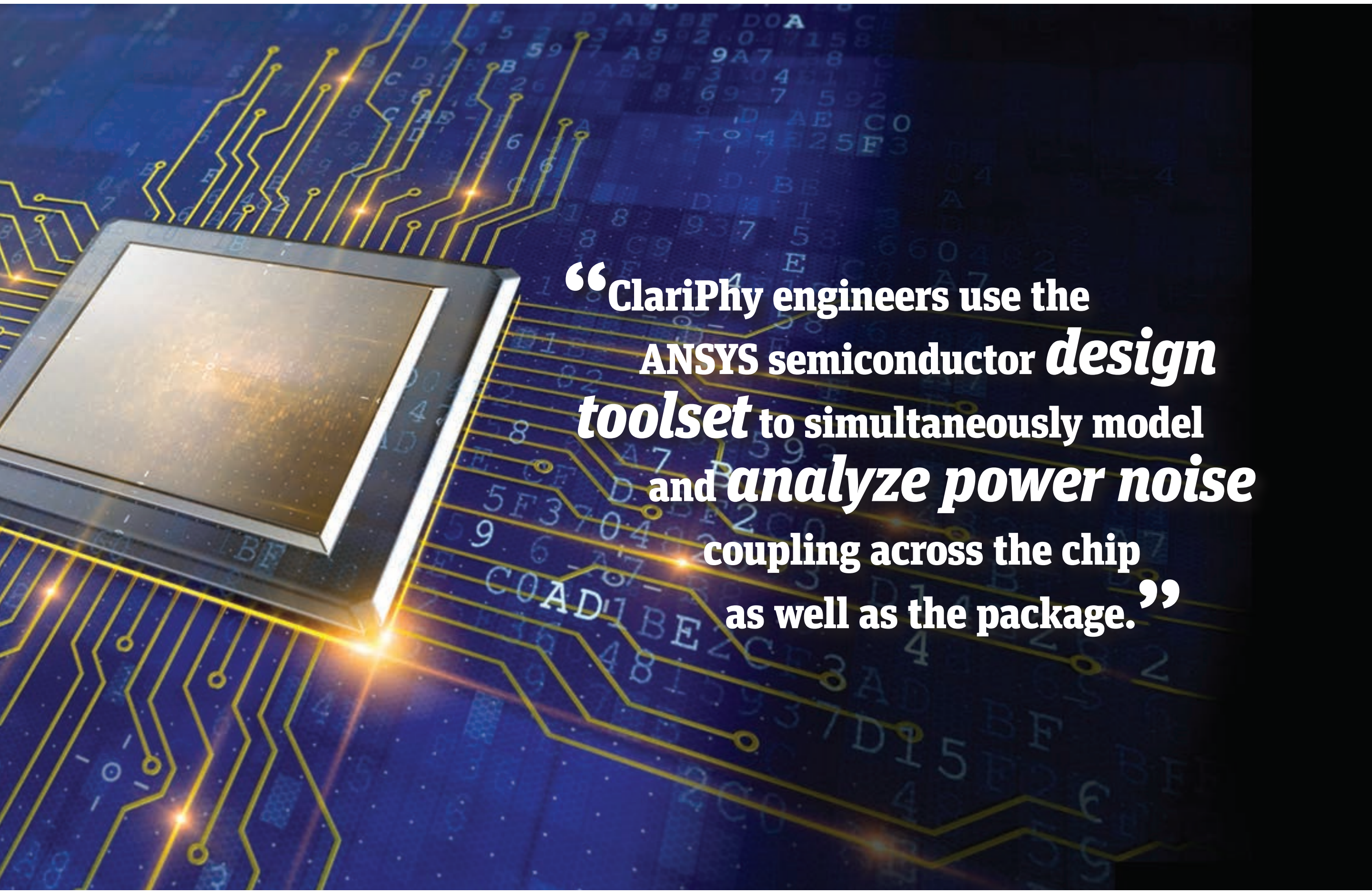


Diagram shows important impact of package on PDN.

In the past, ClariPhy engineers were able to simulate the PDN on the chip but could not accurately account for package effects or analyze the segments and components that comprise the package. With the reduced margins of the most advanced process nodes used today, PDN problems rooted in the package could necessitate an extra foundry spin, which could potentially delay delivery of the chip by a year.

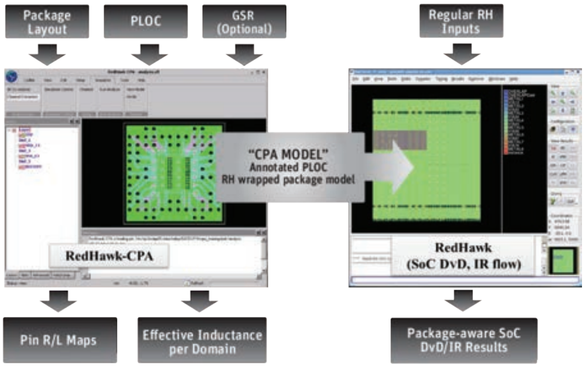
Today, ClariPhy engineers use the ANSYS semiconductor design toolset to simultaneously model and analyze power–noise coupling across the chip as well as the package. They can now make intelligent trade-offs to manage the noise at



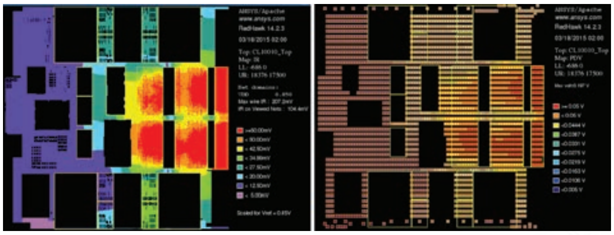
the chip and package levels from the early stages of the design process. This makes it possible to deliver new products within a time window that enables success in the market.

POWER DISTRIBUTION NETWORK DESIGN CHALLENGES

Each chip typically has a maximum dynamic voltage drop that must be maintained through the circuit in all modes of operation. With the continuing adoption of lower on-chip supply voltages in modern semiconductor devices, circuits are becoming very sensitive to power supply noise. As margins for proper functionality decrease, just several millivolts of noise on the power rail can make the difference between a logic 1 and a logic 0, and potentially corrupt the data transmission. But, before the power even reaches the die, the system experiences a voltage drop at the chip–package interface. Static IR drop plays a role, but simulating time-dependent dynamic voltage drop, impacted by passive elements such as package inductance and decoupling capacitors, is even more important. With the number of transistors per square millimeter doubling with each product generation, the amount of current and number of signals in this small area also tends to double. This makes it increasingly difficult to supply sufficient voltage to every transistor on the chip under all possible operating conditions to avoid a dropped bit that could translate to a data transmission error.



^ Package extraction flow in ANSYS RedHawk-CPA



In a chip–package co-simulation, voltage drop on the metal 1 layer on die (left) closely matched voltage drop on bumps (right), indicating that the package caused the PDN problem.

Previously, separate chip and package teams worked in silos, relying upon upfront, project-wide decisions on specifications to guide their design on the chip. ClariPhy chip engineers simulated the voltage delivered across the chip and the resulting drop in voltage. But they had no way to accurately determine the value of voltage that was conveyed to the chip from the package, which in many cases had already been reduced, primarily due to the inductive effects of the package. This approach sufficed for previous technologies with generous design margins, but it will simply not work for the latest generation of products. The new designs are so intricate that if the first sample comes back from the foundry with PDN noise problems, it could easily take a year to fix, drastically reducing the revenue generated by the product.

CO-SIMULATION OF CHIP AND PACKAGE

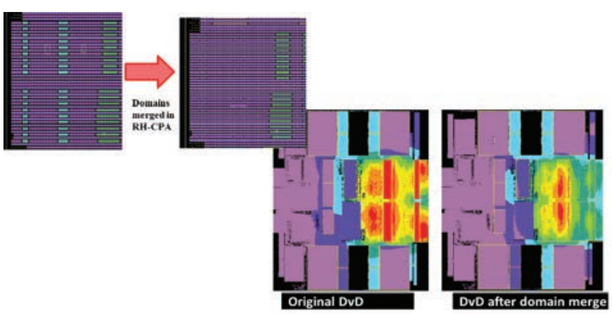
ClariPhy switched to ANSYS tools a few years ago because of the software's unique ability to incorporate the chip and package into a single simulation, enabling engineers to accurately determine dynamic voltage drop with package effects included, and to troubleshoot PDN noise problems all the way upstream to the source. Engineers use ANSYS RedHawk chip package analysis (CPA) to import the package layout and bump location file and automatically connect the bump locations on the chip to the package pins on the layout. RedHawk-CPA generates a 3-D finite element model to extract the high resolution (per-bump) physical RLCk parasitics of the package. Finally, the voltage sources are assigned and a package model is generated for use in RedHawk.

"Engineers can better understand the *complex interactions* between the chip, package and PCB, and *address these issues* early in the design cycle."

A recent chip–package co-simulation showed that voltage drop on the metal 1 layer (the lowest metal layer that interfaces with the silicon) closely matched the bump voltage map (at the interface between the package and die), indicating that the majority of the voltage drop occurred within the package. ClariPhy engineers added probe points across three different package locations on all 10 package layers. The voltage probes showed a large drop across the package core. Engineers then reduced the core height and re-ran the simulation. They discovered that effective inductance was reduced between 20 percent and 30 percent in seven key domains. This reduction in effective inductance manifested itself in improved chip power integrity, with lower dynamic voltage drops seen across the chip.

In another example, ClariPhy engineers evaluated the impact of merging power domains that were isolated at the chip as well as the package to supply adjacent blocks. Edits were quickly made in RedHawk-CPA to merge the domains and re-extract the package models. The dynamic voltage drops were substantially lower in the merged domains.

In a third case, ClariPhy engineers noticed that multiple instantiations of the same block, each of which was sinking the same power through the same power grid architecture, had very different voltage drops. They looked at the pin RL maps in RedHawk-CPA and determined that the packaging accounted for nearly the entire difference in voltage drops among identical blocks. They proceeded to set probes within the package to zero in on the root cause.

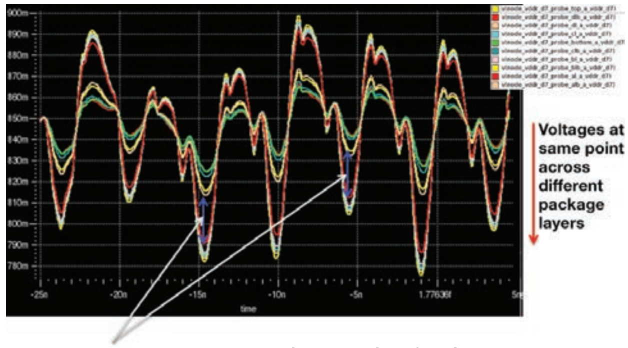


Merging power domains in another study provided a substantial reduction in dynamic voltage drop.

GETTING THE DESIGN RIGHT THE FIRST TIME

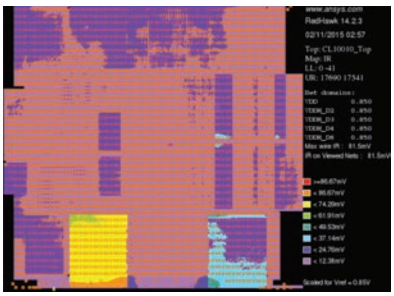
RedHawk-CPA enabled collaboration across ClariPhy chip, package and system teams throughout the design of the CL20010 LightSpeed-II™ 200G coherent optical transport solution, which has 5 billion transistors and over 200 million gates. ClariPhy engineers used chip–package co-analysis to identify problems and provide timely information to the package team during the course of the project to help them make necessary edits. These edits were then used to update the chip simulation to ultimately mitigate package-induced voltage drops on the chip. Furthermore, co-designing the two interdependent components prevented overdesign and opened up previously unforeseen opportunities for cost savings.

The new chip was awarded a perfect 5.0 score in the 2016 Lightwave Innovation Reviews program. One of the judges commented: “This 200G Coherent SoC [system on chip], is truly groundbreaking, breathtaking and revolutionary, and ushers in an entire new era in optical communications.”[2]



Voltage probes for the same chip–package co-simulation showed a large drop across the package core (as indicated by arrows).

Equally as important as enabling the package and system effects when analyzing the chip, it is critical to take the chip and package effects into account when analyzing the system. The ability to generate a chip power model (CPM) from RedHawk with a direct link into ANSYS SIwave enables ClariPhy PCB/system engineers to analyze the PDN of their reference boards, which in turn helps customers design their systems using ClariPhy chips.



In a third case, multiple instantiations of the same block had different voltage drops.

Chip development can be a two-year process, and in the highly competitive semiconductor industry, reducing time-to-market is crucial for success. A spin could take a year, and during that time the market window could close. Issues with bugs and noise could destroy a return on a research-and-development investment. ANSYS tools allow early trade-offs to avoid bugs and noise long before fabrication. Simulation helps ClariPhy to ensure first-pass success and avoid prohibitive fab costs.

Using the ANSYS chip–package–system workflow enabled ClariPhy engineers to better understand the complex interactions between the chip, package and PCB, and to address these issues early in the design cycle to reduce development cost and deliver better products to market quickly.

CL20010 LightSpeed-II coherent optical transport solution

References

[1] Cisco Visual Networking Index: Forecast and Methodology, 2014-2019 White Paper, www.cisco.com/c/en/us/solutions/collateral/service-provider/ip-ngn-ip-next-generation-network/white_paper_c11-481360.html

[2] ClariPhy LightSpeed-II CL20010 Coherent SoC, www.lightwaveonline.com/articles/2016-innovation-reviews/clariphy-lightspeed-ii-cl20010-coherent-soc.html

The Big Data CHILL

To help reduce massive power demands, many data centers are increasingly using liquid cooling to complement or even replace air cooling systems. The leading provider of closed-loop liquid cooling systems for data centers, Asetek, uses thermal simulation with ANSYS Icepak to optimize cooling system components.

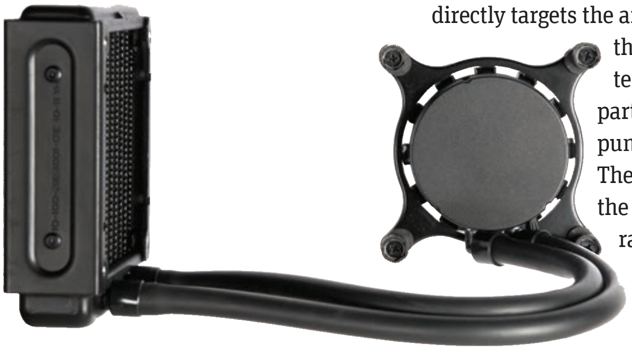
By **Kåre Elgaard Buskov**,
Thermal Engineer, and
Mette Nørmølle,
Vice President of Engineering,
Asetek A/S, Aalborg, Denmark

Overheating equipment is an archenemy in the big data industry. Producing, storing and analyzing huge amounts of information requires ever faster and denser processing power. The requisite compute cycles — which enable the Internet of Things (IoT) — in turn produce massive quantities of heat that must be removed. In 2013, U.S. data centers alone consumed 86 billion kWh just cooling their servers [1] — the equivalent of powering 7.6 million homes. Worldwide, data center servers demand roughly 1.5 percent of the electricity generated annually.

To help reduce the industry's massive power demands, many data centers are increasingly using liquid cooling to complement or even replace air cooling systems. The Danish company Asetek is the leading provider of closed-loop liquid cooling systems for data center computing. In response to market demands for both highly dense and highly customized server racks, Asetek is continually innovating to develop cooling systems with constituent building blocks that are configurable for a wide range of data center needs. As part of the company's overall design process, thermal simulation with ANSYS Icepak computational fluid dynamics software is an important tool for optimizing cooling system components.

LIQUID LOOP COOLING SYSTEM

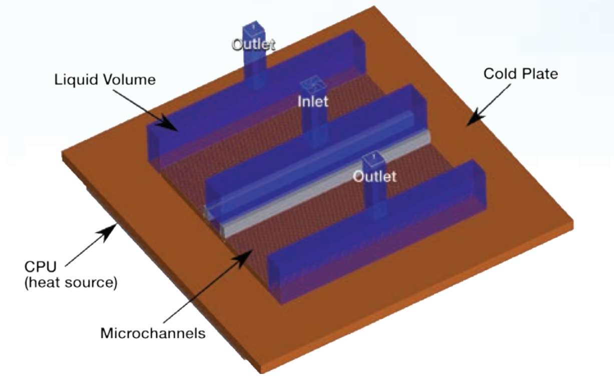
In an Asetek liquid loop system, the cooling liquid is primarily water. The system directly targets the areas of highest heat flux within a server. These are primarily the CPUs, GPUs and memory modules that can have operating temperature ranges of 70 C to 95 C (158 F to 203 F). The main parts of an individual rack cooling unit are a metal cold plate, a pump, dripless quick connectors, and liquid entry and exit tubes. The cold plate, installed above a targeted area, transfers heat from the chip to the cooling liquid, which is pumped out of the server rack and into the coolant distribution unit for the rack tower. The hot coolant passes through a liquid–liquid heat exchanger where it is cooled by facilities water before re-entering the server rack.



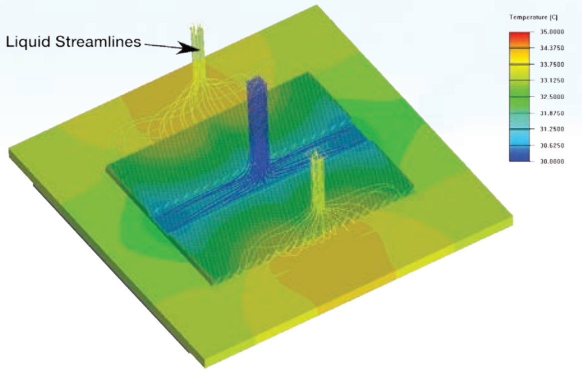
The process of directly cooling hot spots in this manner requires maximizing the heat transfer of the system to cool a particular processor or other module. For Asetek's engineers, this means optimizing the surface area of the cold plate that can transfer heat to the liquid. The cold plate contains a series of microchannels through which cooling liquid is pumped. The Asetek team uses Icepak to aid in design and configuration of these microchannels to analyze the cooling needs of different chips.

supply temperature specified for the servers. Additionally, engineers must create a more compact design to increase the density of processors inside the server racks. Further, the overall thermal impedance of the system must maintain the coolant temperature below a maximum value of 60 C (140 F). Some chip designs will thus require higher coolant flow rates to dissipate more heat.

Typically, the Asetek team will run 30 to 40 simulations for a new cold plate design as part of a parametric study.



^ Icepak model of a CPU cold plate with the solid (brown) and fluid (blue) zones. Cooling liquid enters through the tube in the center, passes through the microchannels in the copper cold plate, and exits through the two outlet tubes on the ends.



^ Contours of temperature on both the copper cold plate and the liquid stream lines showing the fluid heating up over its path through the cooling unit

COLD PLATE SIMULATION

For Asetek to begin a cold plate design, the chip manufacturer first provides data about the geometry and heat dissipation of the particular card or module that needs to be cooled. Depending on the complexity of the components, Asetek engineers will then either import 3-D CAD geometry or build the card or module geometry directly inside Icepak. The team also builds the cold plate geometry before performing the overall thermal analysis using Icepak. A usual mesh size for the fluid zone ranges from one to three million cells. A simulation of this size range converges in 10 to 30 minutes running on four processors.

The systems are designed to handle the case in which a continuously overclocked chip transfers the maximum possible heat to the cooling liquid, at a maximum liquid

They will vary such parameters as the width, height and spacing of the microchannel fins, the plate thickness, and the flow rate and inlet temperature of the cooling liquid. Icepak has been shown to be a good tool to do these kinds of parametric studies rather quickly. The entire process, starting from the chip geometry to the final cold plate design, is usually completed within two weeks.

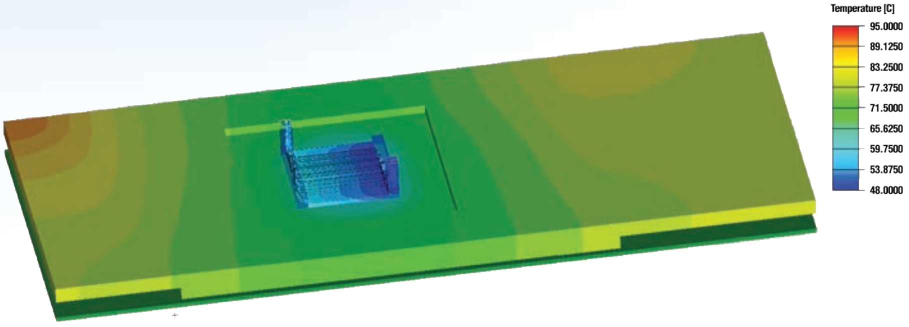
ENABLING MATERIAL AND ENERGY SAVINGS

In addition to speeding up design cycles, optimizing cold plate designs with Icepak has enabled Asetek engineers to

ANSYS Icepak
ansys.com/icepak

switch from copper to aluminum depending on the particular cooling needs. Although copper has superior thermal conductivity, new microchannel designs with an aluminum plate can outperform legacy copper pin fin designs for specific applications that have lower heat dissipation density. Substituting aluminum for copper can reduce the raw materials cost of a cold plate by about 40 percent, resulting in lower-cost cooling solutions.

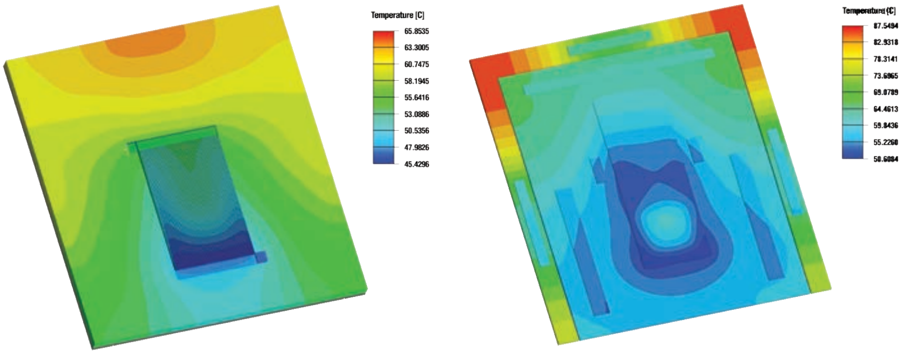
Once a liquid cooling system is implemented in a data center, it can reduce the need for expensive blowers and air-conditioning systems, and cut the overall power needed for cooling by up to 50 percent. Such substantial savings from power consumption enable companies to recover the capital cost of a liquid cooling system within a year. Using the system's heat exchanger, cooling liquid exiting server racks at 60 C (140 F) can be recycled to provide heat for water and residential and commercial buildings. Reusing heat that would otherwise be vented to the atmosphere can prove beneficial, especially in cold climates. At Tromso University, which is located above the Arctic Circle in Norway, the facilities that house 12,000 students and staff are kept warm year-round by the heat that Asetek liquid loop systems capture from the university's data center.



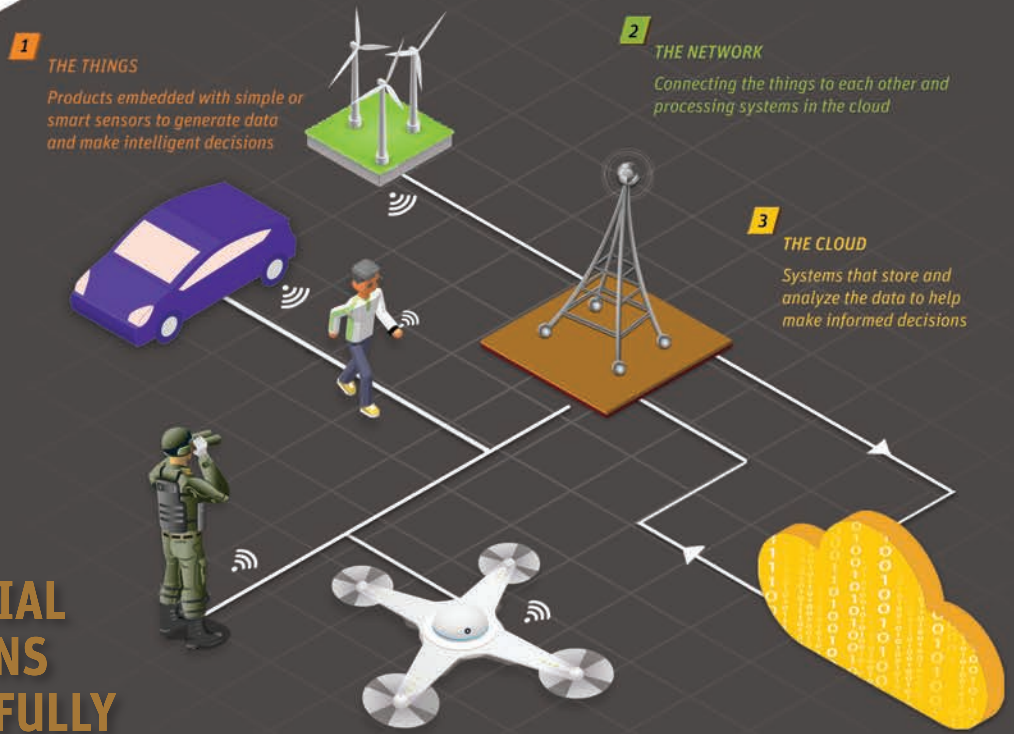
^ **Temperature contours on the graphics card cold plate. Away from the GPU, the plate does not have microchannels, so the temperature of the plate varies depending on the location of other power-dissipating components, such as GDDR memory modules, field-effect transistors or inductors.**

> **"Thermal analysis with *Icepak* enables Asetek to continue delivering market-leading *cooling solutions* to IoT companies."**

At data centers like Tromso and around the world, Asetek's products are at the forefront of enabling large-scale energy savings. Thermal analysis with Icepak enables Asetek to continue delivering market-leading cooling capabilities to the companies designing IoT solutions that keep billions of people and machines connected to the internet. Going forward, the design team is investigating the use of simulation to optimize the pumps that drive the cooling liquid, and also the air flow through complete server racks to ensure sufficient cooling of all components. All of this is leading the future of big data looking very "chill" indeed.



^ **Temperature contours on the cold plate and microchannels (left) and on the graphics card and thermal gap pad strips (right)**



SEVEN CRUCIAL APPLICATIONS TO SUCCESSFULLY ENGINEER THE INTERNET OF THINGS

THE INTERNET OF THINGS AND THE PRODUCTS AND DEVICES THAT MAKE IT WORK ARE INCREDIBLY COMPLEX, CONSISTING OF EVERYTHING FROM MAMMOTH DATA CENTERS TO TINY SOLDER BALLS ON A CHIP. EACH COMPONENT AND SUBCOMPONENT IS AFFECTED BY THE OTHER COMPONENTS AND THE ENVIRONMENT IN WHICH EACH OPERATES. SEVEN APPLICATIONS, ENABLED BY SIMULATION, ARE CRITICAL TO COST-EFFECTIVE AND TIMELY DEVELOPMENT OF IOT PRODUCTS.

BY ANSYS

In its simplest form, the Internet of Things (IoT) comprises three elements: the things, the network or gateway, and the cloud. Things, such as cars, phones, robots, industrial equipment and even homes, are becoming smart and connected. More and more processing power is being added to these things. The network is integral to the IoT infrastructure; without it there are no connected devices. A robust and reliable network includes high-speed routers, switches and gateway technology. The cloud consists of data centers and the software that runs much of the business logic of the IoT.

Each product or device that makes up these three elements is in itself a collection of components that must work together reliably. Components, both electronic and structural, include integrated circuits, switches, antennas, sensors, batteries, cases and much more that in turn make up larger components and complete systems. The complexity is astounding. Each component must work as intended by itself, as part of the complete system of components and within operating environments that are often harsh and sometimes unexpected.

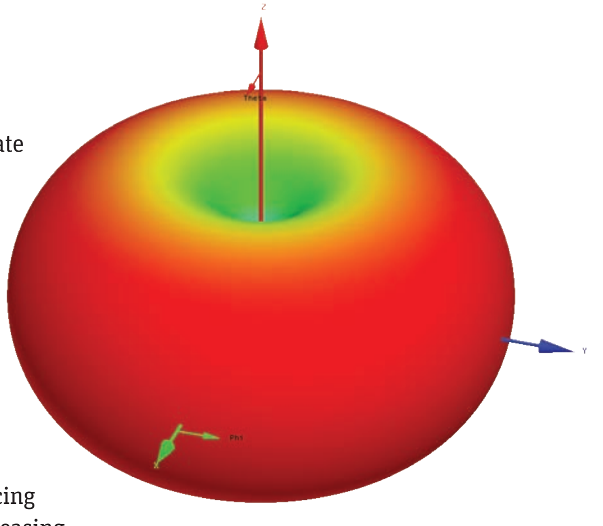
> "Synapse uses ANSYS HFSS and the ANSYS human body model to evaluate performance of various antenna designs by modeling the complete system, including the wireless device and antenna and their interactions with the human body."
>
> — SYNAPSE PRODUCT DEVELOPMENT

Studies have shown that designing products of this complexity through an integrated simulation platform is much superior to physical testing or performing simulation in silos. This integrated platform enables seven applications in which simulation is essential to designing successful products for the IoT. ANSYS provides not only a best-in-class platform but proven tools that reduce costs, increase reliability, and speed time-to-market for each of these applications.

ANTENNA DESIGN AND PLACEMENT

Antennas are essential for the things within the IoT to communicate with each other reliably. Without antennas, the IoT would not exist. Wireless systems that are prototyped and tested in anechoic chambers can experience issues with multipath signal propagation and fading once integrated into a device. The structure of the device itself — be it a manufactured object or a human body — as well as motion and other environmental factors can cause the antenna to fail or not perform optimally. To provide rich functionality, modern devices may use a combination of wireless technologies — Bluetooth®, Wi-Fi, LTE — that require multiple antennas. Antenna coupling and co-site issues can degrade performance. For example, a wireless sensor network deployed in a factory contains sensors with dipole antennas to communicate with other sensors. The ideal radiation pattern of a dipole antenna resembles a donut, but, when deployed in the industrial setting, the complex structures and interference from other antennas distort the radiation pattern, reducing antenna efficiency, increasing power consumption and leading to unreliable performance and failure.

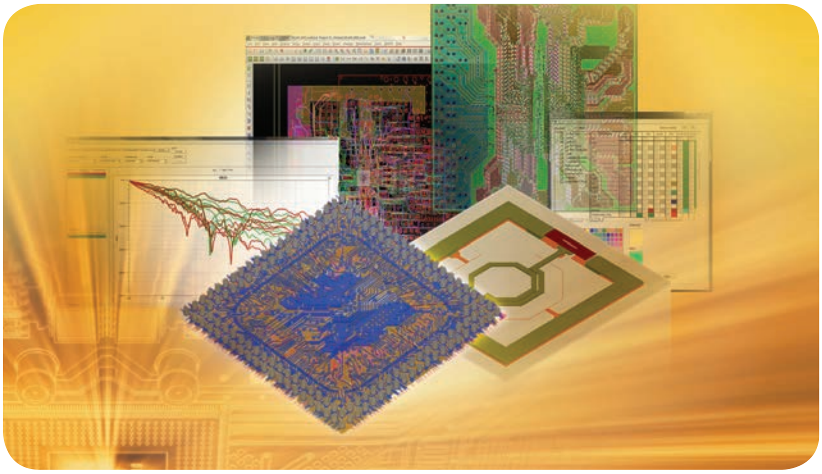
ANSYS provides solutions to predict the effects of the entire industrial environment on the performance of antennas and wireless devices. This approach provides greater insight, improves accuracy and increases reliability.



Simulation of donut-shaped radiation pattern of a dipole antenna

Wearing a Wire
ansys.com/wire

CHIP–PACKAGE–SYSTEM DESIGN



^ Chip–package co-analysis

Designing high-speed printed circuit boards (PCBs) and semiconductor integrated circuits (ICs) that make up devices, networks and data management for the IoT poses significant challenges. Engineers must address the complexity of designing for lower operating voltages, increasing circuit density and faster data rates. Engineers must balance the requirements of three broad areas that affect product reliability — electrical, thermal and mechanical performance. They also need to simulate the interactions between the IC, the IC package and the PCB.

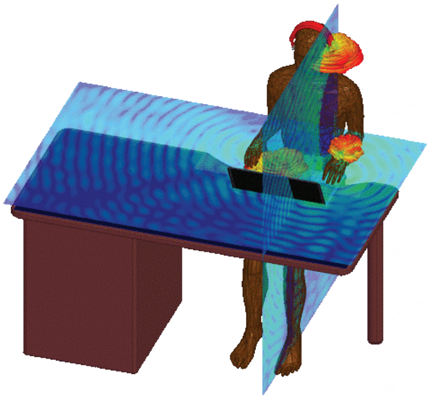
Electrical reliability requires engineers to perform signal- and power-integrity analysis. Power integrity simulations ensure that power delivery networks are robust, and signal integrity analysis minimizes crosstalk and electromagnetic interference (EMI). Addressing thermal reliability entails simulation to evaluate the impact of board temperatures and associated components so that devices operate reliably in the specified operating range. Mechanical reliability requires a thermal stress simulation to evaluate thermal and mechanical stresses in the board, as well as solder joints between board and components.

In addition to individual physics simulations, engineers must consider the interaction between physics disciplines, coupling electromagnetic field simulation with thermal simulations and connecting thermal simulations with structural analysis. This method provides a holistic view of the overall reliability of the PCB design.

The chip–package–system workflow, unique to ANSYS, enables engineers to improve electronic system performance. With all relevant systems-level considerations modeled and simulated, engineers can reduce electromagnetic interference, develop robust electrostatic discharge (ESD) protection, and improve electronic systems to power the IoT economy.

Overcoming Uncertainties in High-Speed Communication Channels
ansys.com/uncertainties

POWER MANAGEMENT



^ Wireless communication

Anyone whose smartphone battery has run out understands the essential role of power management. But power management isn't just about smartphones or Wi-Fi. Energy harvesting, wireless power transfer and low power IC design are the foundations on which many IoT devices will be built.

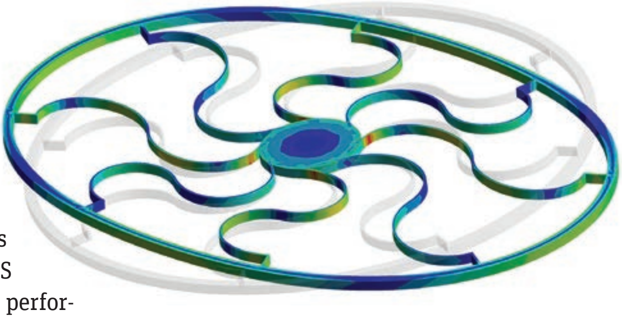
Energy from mechanical motion, heat, piezoelectric material and electromagnetic emissions can be captured and converted directly into electricity. When designing energy harvesting systems, engineers need to consider several parameters, including the energy source, transducer type, power efficiency, required power levels and energy storage. ANSYS provides a wide range of simulation tools that work seamlessly together to take all these factors into consideration.

When designing wireless systems, safety is a key consideration. Standards and regulatory agencies limit the amount of electromagnetic energy that can be delivered to living tissue. ANSYS simulation tools, including human body models, can be used to design and analyze a variety of power delivery systems and their impact on the human body.

SENSORS AND MEMS DESIGN

Sensors, actuators and other MEMS (microelectromechanical systems) devices are essential to the IoT. They gather the information from the environment necessary for human–machine interfaces, device management and product feedback; trigger action based on that feedback, such as starting or stopping heating systems, showing a warning, or opening a valve; and perform a whole host of other functions. Designers of sensor and other MEMS devices face business and technology challenges when designing, prototyping and creating compelling products that can mean the difference between success and failure. To gain a competitive advantage, MEMS manufacturers need to develop their products as fast and efficiently as possible.

MEMS and sensors are complex because of their special functions, challenging manufacturing processes and often tiny size. MEMS devices are so small that performance measurement equipment can impact device function, making it difficult to obtain reliable performance data. Simulation provides accurate insight into the performance of these devices beyond what physical prototyping affords.



^ Contours of z-axis deformation on a gyroscope

ANSYS solutions enable simulation of a wide range of sensors, actuators and other MEMS devices, from RF sensors dependent on electromagnetic fields to gyroscopes dependent on mechanical motion to piezoelectric devices that depend on both mechanical and electromagnetic components. Proven solvers and coupling solutions enable high-fidelity analysis of device designs. Once an initial design is created and simulated, ANSYS tools allow the entire device to be optimized before physical prototyping, including how the components will work together and transmit their information.

EMBEDDED SOFTWARE DEVELOPMENT

Many modern cars contain 50 to 100 million lines of code. With autonomous vehicles on the way, software content will rapidly increase. But it is not just cars: Software is essential to adding richness and smart functionality to many IoT devices, including industrial equipment, robots, planes and drones. Because many of these products and systems are safety- or mission-critical — for example, braking systems on cars and planes — the control software must operate flawlessly. When systems fail, they must fail in a predictable way to minimize damage.

Industry regulations, certifications and qualifications often govern the reliability and performance of software. Software development is no longer just about writing the code: It is about verification and validation. For each line of implementation code, software engineers often find themselves writing many more lines of verification code. Despite the amount of effort expended, software code bugs continue to persist, leading to safety recalls, security breaches and sometimes tragic outcomes.

ANSYS has created a model-based embedded software development and simulation environment with a built-in automatic code generator that significantly accelerates the pace of embedded software development projects. Engineers can use ANSYS solutions to model complex systems, understand the interaction of various subsystems, and generate high-integrity software code that complies with industry standards. The ability to generate millions of lines of code at the push of a button not only removes human coding errors, but also increases productivity, quality and traceability of code.

Automatic code generation using ANSYS software modeling tools enables engineers to express the design specification in a formal manner. As a result, companies can produce software with a significantly reduced certification cost and reduce the number of very expensive test demonstrations. Software modeling and simulation can reduce software generation time, providing a time-to-market advantage.

DESIGNING FOR HARSH ENVIRONMENTS

Whether they are employed in industrial, aerospace or consumer applications, IoT devices can be subjected to harsh environments including vibrations, impact and fatigue. Despite these conditions, IoT devices must be robust and remain active for extended periods and across great distances without maintenance. In these extremes, a malfunction can result in significant investment to repair or replace the system, mission failure, and even risk to human lives. NASA has shown that 45 percent of first-day spacecraft electronics failures were due to damage caused by vibrations during launch.

Engineers must consider these potential harsh environments very early in the development process when design choices can be made at the lowest cost — and with the least impact on the project schedule. Physical prototyping is simply not a viable option for many obvious reasons. Not only is it difficult to create all the possible test scenarios given the constraints of time, budget, location and resources, but also the measurement results can vary greatly and lack the fidelity needed for IoT and many other critical applications. One leading aerospace and defense firm simulated harsh environmental conditions, including vibration, to eliminate expensive destructive testing. As a result, they were able to save over $1 million by reducing the development time, eliminating consultant fees, reducing physical testing, improving product capabilities for accuracy and maintaining safety.

^ Designers used ANSYS simulation tools to optimize this wearable device belt clip for strength, fatigue life and thickness.

"Using design and simulation tools from ANSYS, Astrobotic quickly designed and refined a lightweight aluminum and composite spacecraft able to withstand static acceleration and dynamic random vibration loads of launch while maintaining an acceptable level of safety. Simulation helps reduce costs related to prototypes and physical testing."

— JOHN THORNTON, CHIEF ENGINEER, ASTROBOTIC TECHNOLOGY, INC.

Astrobotic Case Study
ansys.com/astrobotic

VIRTUAL SYSTEM PROTOTYPING

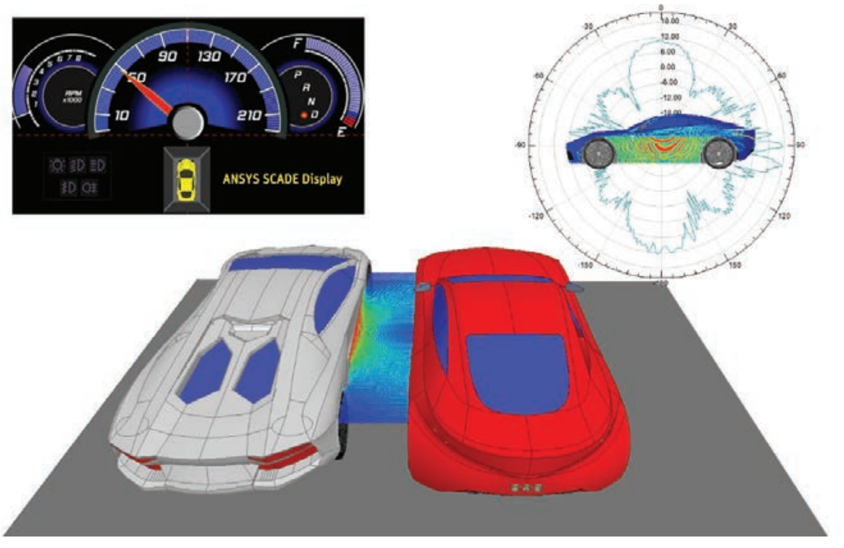
As product complexity grows, so does the need for enhanced simulation capabilities. The complexity within systems arises from the challenges of connecting the individual pieces to ensure that they work together as specified and expected. Coupling physical attributes of a product with systems and embedded software enables companies to greatly minimize integration issues, reduce costs, increase the likelihood of first-pass success and ensure that products perform as expected.

While it is easier to visualize the IoT in terms of individual devices or components — a smartphone, a thermostat or a wind turbine — the complex and invisible networks that connect them, as well as the cloud that stores and delivers data on demand, require sophisticated modeling and simulation. The smart wind turbine, for example, needs to adjust its behavior according to wind patterns, the amount of energy on the grid and the behavior of other smart wind turbines. To model the complete operation of this wind turbine and its interactions with the real

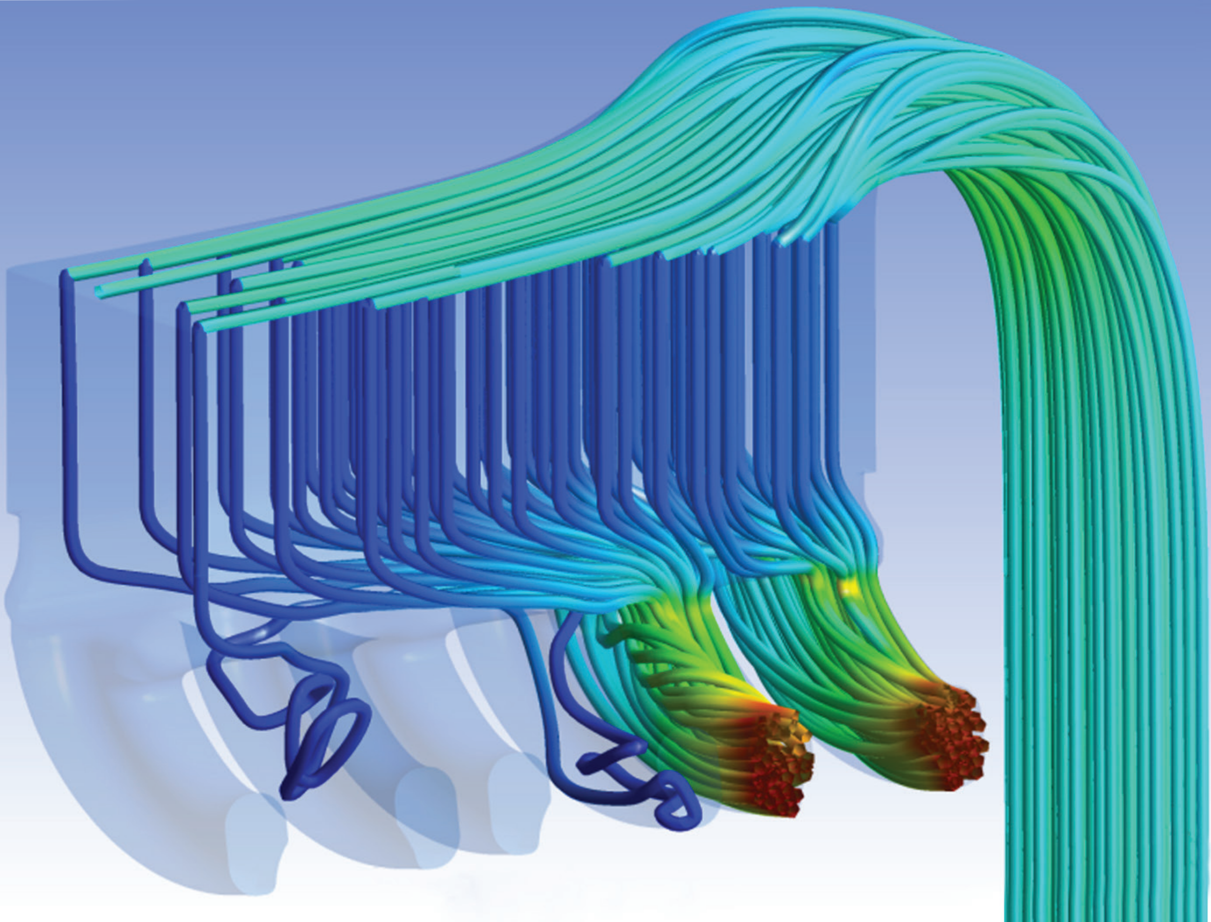
Mastering Complexity
ansys.com/complexity

world, engineers need to use simulation tools that model fluid dynamics, embedded software, and structural and electronic functions.

The interactions of the software, the electronics hardware and the multidomain nature of the problems significantly increase the complexity of the engineering challenge. Simulation software from ANSYS can help by providing validation results that include systems-level qualities, properties, characteristics, functions, behavior and performance insight. Based on this high-level perspective, system designers can make informed design choices that optimize the performance of not only each individual component, but also the entire system.



An advanced driver assistance system (ADAS), common in today's cars, requires both electromagnetic and systems simulation to develop.
Image courtesy ESSS.



CHANGING CHANNELS

Modeling the full geometry of the complex wave channels in a new, integrated intake-manifold–intercooler design for fuel-efficient cars previously required lengthy solution times. Magneti Marelli engineers used the directional loss model in ANSYS CFD software to simulate the channels as a porous medium in one-third of the time.

By **Massimiliano Di Paola**,
Senior CAE Analyst, and
Nazario Bellato, Simulation Manager,
Magneti Marelli Powertrain S.p.A.,
Bologna, Italy
Bhartendu Tavri,
Simulation Engineer, Magneti Marelli
India Pvt. Ltd., Gurgaon, India

^ The directional loss model produced the velocity plot shown here in one-third of the time required in the past.

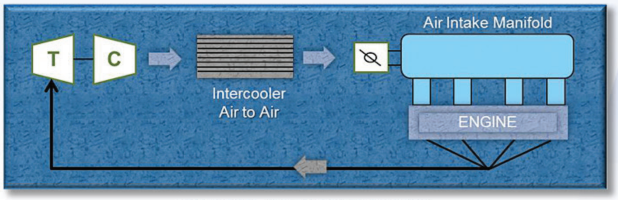
Based on government regulations and customer demand, automobile manufacturers need to improve fuel economy and reduce emissions for vehicles. For this reason, many are adding turbochargers to their cars. However, to ensure reliable operation and performance of these turbocharged vehicles, other changes to the engines are required.

TURBOCHARGERS AND INTERCOOLERS

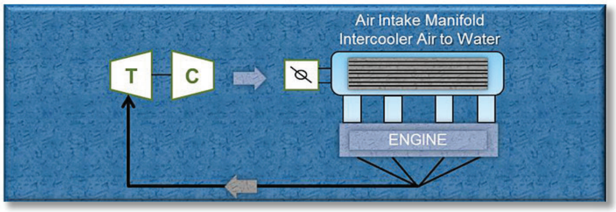
In most piston engines, intake gases are pulled into the cylinder by the pressure reduction caused by the downward piston stroke. Turbochargers and superchargers increase the performance and efficiency of internal combustion engines by compressing the air prior to entry into the intake manifold, so that more air is forced into the cylinder and each engine cycle generates more power. However, as turbochargers compress air, the temperature

> **"The current design delivers a *substantial increase* in heat exchange through the intercooler, *reducing outlet temperature* by 8 percent to improve engine *performance*."**

of the air increases, leading to reduced density and limiting the air mass that can be forced into the cylinder. This in turn affects combustion ability. High intake air temperatures also increase the risk of pre-ignition or knocking, which can cause serious damage to engines. For many years, the most advanced turbocharged and supercharged engines, such as those used in racing, have used an intercooler to remove heat after the air has been compressed



CURRENT CONFIGURATION



NEW CONFIGURATION

^ Integrating the air-to-water intercooler with the air intake manifold provides a substantial reduction in cost and weight.

to increase air density. The most common type of intercoolers, called air-to-air intercoolers, use atmospheric air to cool the intake air. This type is relatively simple and inexpensive, but its efficiency is limited by the amount of contact with and the temperature of ambient air.

Air-to-water intercoolers provide considerably higher performance by using water to extract heat from the air. But air-to-water intercoolers have rarely been used in production vehicles in the past because they require a pump, radiator, fluid and plumbing that add considerable cost and weight to the vehicle. A significant recent trend in automotive design is the use of air-to-water intercoolers that are lighter, more compact and less expensive because they are integrated into the intake manifold.

But efficiently integrating the intercooler and intake manifold is challenging. The intake air and cooling water must be channeled through the integrated intake-manifold–intercooler in such a way that heat transfer is maintained at high levels to keep intake air temperature low while minimizing pressure losses that reduce engine efficiency. Magneti Marelli engineers use ANSYS computational fluid dynamics (CFD) software to optimize the performance of a new, integrated intake-manifold–intercooler in a fraction of the time that would have been required using previous simulation methods.

MODELING A COMPLEX SYSTEM

Designing this complicated system using conventional build-and-test iterations would be very expensive and time-consuming because of the large number of design parameters and the limited amount of information that could be collected during physical testing. On the other hand, the product presents a difficult simulation challenge because of its complex internal geometry, which includes tiny wave channels that move air through the intercooler on a tortuous path to transfer as much heat as possible to the surrounding liquid. Accurately capturing flow behavior in regions such as these, where abrupt changes are expected in key variables such as velocity, pressure and temperature, normally requires that the mesh be refined by generating an inflation (boundary) layer. In the past, Magneti Marelli engineers used a hybrid mesh in these applications, with hexahedral elements in the boundary layer and less computationally intensive tetrahedral elements in the rest of the flow volume.

But in this case the geometric complexity of the wave channels was so great that a good quality hybrid mesh would yield long solution times, even with large computing resources.

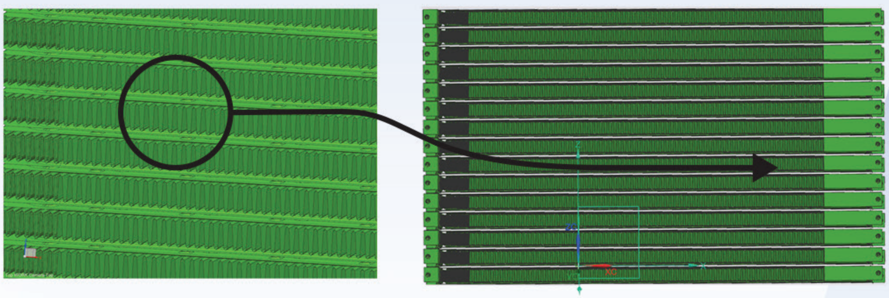
Steering Toward the Future
ansys.com/steering

Thermal Management with ANSYS Multiphysics
ansys.com/thermal101

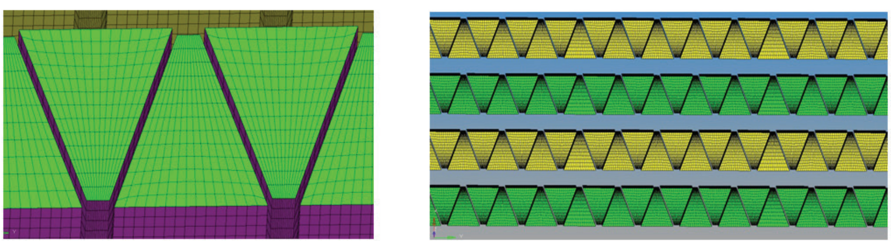
A hex-dominant mesh was not practical because of the presence of flow obstructions that were designed to improve heat transfer by generating turbulence. Even by ignoring the flow obstructions, and meshing the micro-channels as regular fluid passages with an assumption of trapezoidal cross-sections, there would have been a very high element count and nonuniform mesh density, resulting in long solution times.

MULTIPHYSICS SIMULATION

Magneti Marelli engineers overcame this problem by using the directional loss model in ANSYS CFX, which is based on Darcy's momentum loss equation for fluids flowing in a porous medium. Using this model, a generalized form of the Navier-Stokes equation together with Darcy's law were solved over a given domain in a form that accounts for the volume porosity of the media and expresses pressure drop in terms of Darcy's law. Engineers ran the simulation and compared the results to physical testing. Then they adjusted the linear and quadratic loss coefficients that control the intercooler's porosity levels in the simulation to match the air pressure loss across the intercooler, as measured by physical experiments. After calibrating the model, it correlated well with physical measurements, while taking a fraction of the solution time that would have been required to model the complete geometry of the intake-manifold–intercooler. The simulation provided far more accurate performance estimates of design alternatives than were obtained in the past with hand calculations. The CFD model based on trapezoidal elements took three days to solve, while the model based on Darcy's law had 12 million fewer elements and took only one day to solve.



^ Tiny wave channels in the intercooler provide a difficult analysis challenge.



^ A more traditional approach involving meshing the wave channels with a trapezoidal cross-section yielded excessive solution times.

With the model validated, Magneti Marelli engineers used it to evaluate the impact of many design parameters on heat transfer and pressure drop through the intercooler. They also exported the pressure fields to ANSYS Mechanical and performed structural simulations to evaluate the stability of the structure. At various stages of the design process, engineers used ANSYS DesignXplorer to rapidly iterate through the design space and identify design parameter values that best met the specified design objectives. The team is now using simulation to study several other complex phenomena, such as the condensation effect inside the air intake manifold and the water hammer effect in the intake system while the engine is being cranked by the starter motor.

Using simulation early in the design process saved time and money by making it possible to optimize the design at an earlier stage than was possible in the past. The company expects to achieve higher performance while creating fewer prototype iterations than would have been necessary using previous design methods. The current design delivers a substantial increase in heat exchange through the intercooler, reducing outlet temperature by 8 percent compared to the previous intercooler design, which, in turn, improves engine performance. The new design also reduces overall pressure loss to improve fuel economy by 5 percent. The project is continuing as engineers currently focus on optimizing the structural and thermal stability of the system. Magneti Marelli has received a very favorable response from major automotive manufacturers on integrated intake-manifold–intercooler technology, and is working to implement the new design in upcoming vehicle models. ^

Improving Turbocharger Design
ansys.com/turbocharger

TO THE TEST

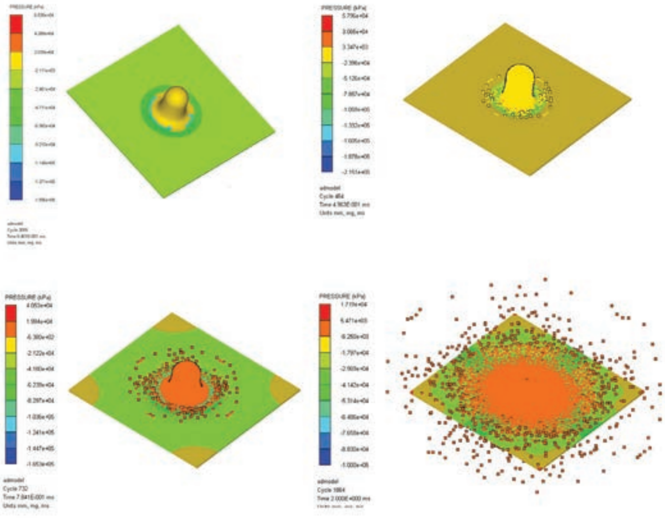
In the past, the only way to determine whether composite aircraft components could withstand bird strikes was with time-consuming physical tests. Now, Hindustan Aeronautics Limited engineers use simulation to get the design right the first time. Bird strike simulation saves the company design time and thousands of dollars per test of composite helicopter components.

By **Vijaykumar Rayavarapu**, R&D Manager, Hindustan Aeronautics Limited, Bangalore, India

In 2014, four U.S. Air Force personnel were killed when their HH-60G Pave Hawk helicopter crashed during a training mission in Norfolk, England. The U.S. accident investigation board found that the accident was caused by geese flying through the aircraft's windshield, knocking the pilot and co-pilot unconscious. They were unable to react when another bird struck the helicopter's nose and disabled the craft's stabilization system. The result was an uncontrolled roll to the ground, destruction of a US$40 million helicopter and loss of life. This is not an isolated incident. According to the United States Department of Agriculture's Animal and Plant Health Inspection Service (APHIS), bird strikes to civilian and military helicopters have resulted in 11 human deaths and 61 injuries since 1990. [1]

^ SPH bird model with Lagrange model of cowling

In an effort to protect crew and passengers from the dangers of bird strikes, regulatory authorities, including the Federal Aviation Administration (FAA) and the European Aviation Safety Agency (EASA), have issued regulations regarding the ability of helicopters to survive bird strikes. For example, the FAA's 14 CFR 29.631 regulation now demands that category A rotorcraft (the highest certification standard, which requires, among other things, assurance of continued flight in the event of failure) be capable of continued safe flight and landing after bird impact. Bird strike certification has been a time-consuming and expensive process because the only way to determine whether a component could survive a bird strike was physical testing. Tests usually needed to be repeated several times because components often failed and replacements were required for each new design. Hindustan Aeronautics Limited (HAL) has substantially reduced the time and cost of certification by using ANSYS Composite PrepPost and ANSYS Autodyn to accurately simulate bird strikes. Simulation makes it possible to efficiently determine a suitable design so that only one test is required per component.



^ Simplified simulation of bird model into flat plate

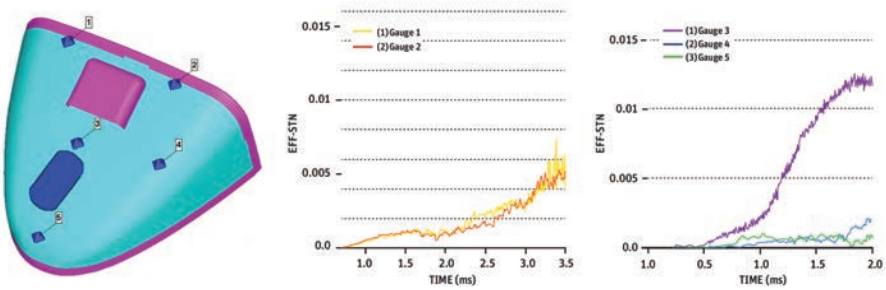
SIMULATION CHALLENGE

The components that require certification on modern helicopters, such as cowling, horizontal stabilizers and end plates, are typically made of fiber-reinforced composites. Cowling refers to detachable panels covering those areas to which access must be provided, such as the engine, transmission and other vital systems. Bird strike simulations are challenging because they are of short duration, cause large material deformation, and involve interactions between bodies with rapidly changing surfaces. The difficulty is increased by the need to model composite materials that include numerous layers, each with its own material, footprint, thickness and orientation.

As a first step to determine the validity of the model used, HAL simulated a simplified case that could easily be done experimentally. The results of physical testing were correlated with the calculations, which confirmed the viability of models used with the aircraft. The bird strike simulation consisted of an idealized geometry striking a flat plate. The bird was modeled as a cylinder with flat ends, and as a cylinder with hemispherical ends. A bird undergoing impact at high velocity behaves as a highly deformable projectile with a yield stress much lower than the sustained stress. Based on this, and also because the density of flesh is close to the density of water, it is possible to approximate the bird as a lump of water hitting the target. The analysis was carried out with the Autodyn solver using the smoothed particle hydrodynamics (SPH) method to avoid numerical difficulties associated with extensive mesh distortion. The results correlated well with the analysis of shock pressures calculated using hydrodynamic theory.

DEFINING COMPOSITE GEOMETRY

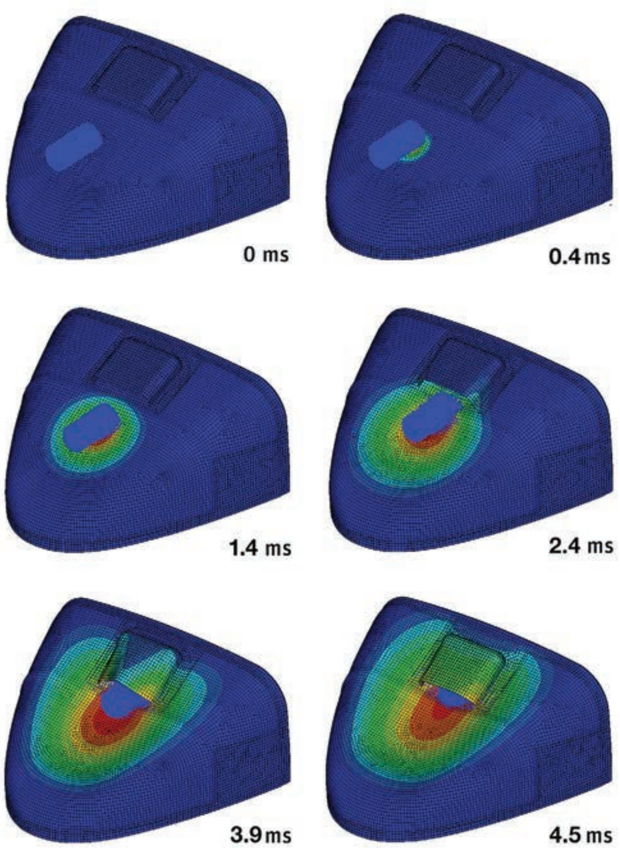
Realistically simulating certification tests requires modeling complex composite structures. HAL imported the geometry of a cowling into the ANSYS Workbench environment. The cowling comprises a Kevlar® fiber skin and a honeycomb core. ANSYS Composite PrepPost was used to define the number of layers and the shape, thickness and orientation of each layer. Compression tests on square specimens were performed according to ASTM standards to determine the properties of the core. The composite



^ Effective strain plot predicted by simulation definitions were then transferred to the finite element model and the solver input file. The material properties for each composite layer were defined with a constitutive material model inside ANSYS Composite PrepPost, with appropriate damage initiation criteria and damage evolution. Further preprocessing was done in ANSYS Explicit STR. The composite definitions from ANSYS Composite PrepPost were seamlessly transferred to Autodyn through ANSYS Workbench.

> **“*Bird strikes* to civilian and military helicopters *have resulted* in 11 human deaths and 61 *injuries* since 1990.”**

A key advantage of ANSYS Autodyn explicit solver is its ability to combine Lagrange, Euler, arbitrary Lagrange-Euler (ALE) and SPH methods in a single problem to produce results with the highest accuracy possible within a reasonable computational time. In this case, the SPH bird model was used to model the bird, while the Lagrange model, with its high computational speed, was used to represent the cowling structure. The model was set up to match the test conditions of a bird strike test conducted at a research facility, including the application of aerodynamic loading to the cowling. Virtual strain gauges were defined within Autodyn at the same positions on the cowling as those used in the physical test.



^ Cowling deformation at various time intervals

CORRELATION WITH PHYSICAL TESTING

Within each element, the Lagrange solver captured the material location of the discretized model and followed its deformation as forces were applied. The solution time was under one hour for a simulation time of 4,000 microseconds. The simulation accurately predicted the basic parameters of the test as well as the damage location and failure size.

The failure mode at different time intervals also matched well with the test results. At the early stages of impact, the mechanical response of the composite structure is controlled by the fiber–matrix interface. At the intermediate stages of impact, when the shock wave reaches the face-sheet–core interface, a negative pressure region begins to develop on the back of the face sheet, giving rise to tensile failures of fibers in this region. At later stages of impact, a substantially larger region of outer face sheet is subjected to negative pressures, causing it to fail structurally. Meanwhile, high strains are observed in the cowling surrounding the top of the projectile.

The correlation study provided a high level of confidence in the ability of the simulation to predict dynamic responses and structural failures subjected to high-energy bird impacts. With the model validated, HAL now uses it to design new exterior structural components that can pass bird strike certification tests the first time. In obtaining EASA certification for a civilian version of the HAL Dhruv Advanced Light Helicopter, simulation eliminated the need for one or two additional tests that were nearly always required in the past, saving time and thousands of dollars in testing for each component that was certified. ^

References

[1] Keirn, G. Helicopters and Bird Strikes; Results from First Analysis Available Online. blogs.usda.gov/2013/06/06/helicopters-and-bird-strikes-results-from-first-analysis-available-online (12/17/2015).

Impact
ansys.com/impact

LONG SHOT

Designing drivers for golf involves trade-offs. Drivers with large clubfaces and deep center-of-gravity (CG) positions have a high moment of inertia and are thus more forgiving of mishits, while smaller clubfaces and geometries can lower drag for faster club speed and longer shots. PING used ANSYS fluid simulation to design unique aerodynamic features in its new drivers that reduce drag and enable the large clubface and deep CG to deliver longer shots while remaining stable on off-center hits.

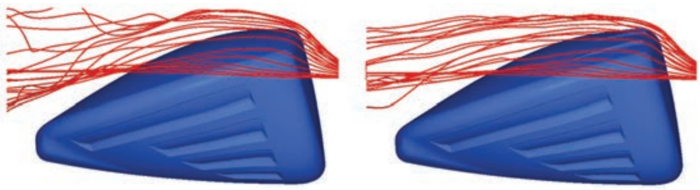
By **Erik Henrikson**,
Innovation and Fitting Science Manager,
PING Golf, Phoenix, USA

For over 50 years, PING, Inc. has been developing golf clubs that assist both amateur and professional golfers in reaching their potential. Because of constant innovation and design expertise, many champion golfers have used PING clubs. Maintaining this pace of innovation requires engineers to use every tool available to achieve design goals. Engineers at PING employ ANSYS computational fluid dynamics (CFD) solutions to improve the aerodynamics of clubs.

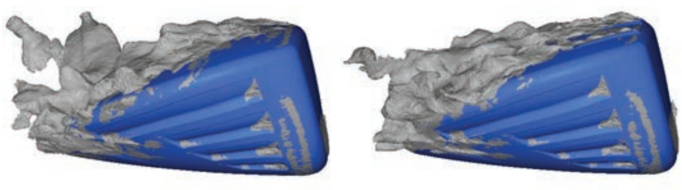
The clubface of a golf driver is usually designed to be as large as the rules allow, with its mass distributed away from its center as much as possible. This increases the moment of inertia of the club, which is a measure of the club's resistance to angular acceleration. So when the clubface contacts the ball away from the center of the clubface, it resists twisting more than a club with a lower moment of inertia. Big clubfaces enlarge the effective sweet spot of the club in all directions, which makes it easier to hit the ball straight.

But large clubfaces and mass distributions centered far from the face typically generate high levels of drag that are mostly caused by pressure differences in the front and rear of the club. This is because the flow around

< Turbulators are shown on top of clubhead of new PING G driver.



^ CFD simulation of clubheads without (left) and with (right) turbulators shows how the new design delays flow separation and reduces drag.



^ Side view of simulation without (left) and with (right) turbulators shows that wake is smaller with turbulators.

a blunt body separates from the body and forms a separated or recirculating region in the rear of the body. The flow exerts more pressure on the front of the body than the separated region exerts on the rear of the body. This difference in pressure generates drag that reduces the speed at which the golfer can deliver the clubhead to the ball, which in turn reduces the ball speed and the distance of the drive.

PING used ANSYS CFD software to create and optimize the design of aerodynamic features on the clubhead that excite the flow near the surface of the club and promote a turbulent boundary layer transition that delays the formation of the separated region toward the rear of the clubhead. This reduces the size of the turbulent wake, which in turn increases the pressure on the rear of the clubhead and thus reduces the drag. The resulting PING G30, and other new G drivers, enable top professionals like Bubba Watson to generate up to 10 additional yards in driving distance while providing smaller but still significant gains to average golfers.

SIMULATING A GOLF SWING

PING engineers set out to study clubface aerodynamics to reduce drag without sacrificing the moment of inertia and other mass properties that also have a great impact on driver performance. They began by importing a PTC Creo® computer aided design (CAD) model of their existing clubhead design into ANSYS Workbench. They used ANSYS Meshing to automatically generate a grid consisting of a mixture of tetrahedral and hexahedral elements. Hexahedral elements are preferred for higher accuracy around the boundary layer between the clubhead and the airstream, so engineers added 10 inflation layers consisting of hexahedral elements around the clubhead to accurately capture the high velocity gradients in these areas. Moving away from the clubface, each inflation layer was sized at 1.2 times the thickness of the adjacent layer.

The velocity and angle of attack of the clubhead with respect to air flow are constantly changing during the downswing, so PING engineers developed a series of models with different velocities and angles of attack to separately address each segment of the downswing.

UNDERSTANDING THE PHYSICS

The simulation provided a fundamental understanding of the physics that occurs when air flows over the previous-model driver. It showed that the flow separates at the leading edge of the clubhead, resulting in a large wake in the rear of the clubhead. This insight gave the engineers an idea. Would adding ridges (which PING calls turbulators) to the top surface of the clubhead generate turbulence to delay flow separation toward the rear of the clubhead and reduce the size of the wake area and drag? A small ridge size would have only a very small effect on the mass properties of the club.

PING engineers modified the CFD model to add the ridges, and, when they performed the simulation again, they observed a significant reduction in drag. The team ran a series of simulations with different numbers of

On the Ball
ansys.com/ball

> “PING engineers modified the CFD model to add the ridges, and, when they performed the simulation again, they observed a significant reduction in drag.”

turbulators, each having different widths, lengths and angles with respect to the clubhead. Each of these clubhead designs was created in the Creo CAD program and then exported to ANSYS Workbench. Engineers simulated these models at a number of different speeds to evaluate how each design would perform for players of different levels. They converged on a turbulator configuration that provides the greatest possible delay in flow separation and the lowest possible drag.

PHYSICAL TESTING VALIDATES SIMULATION

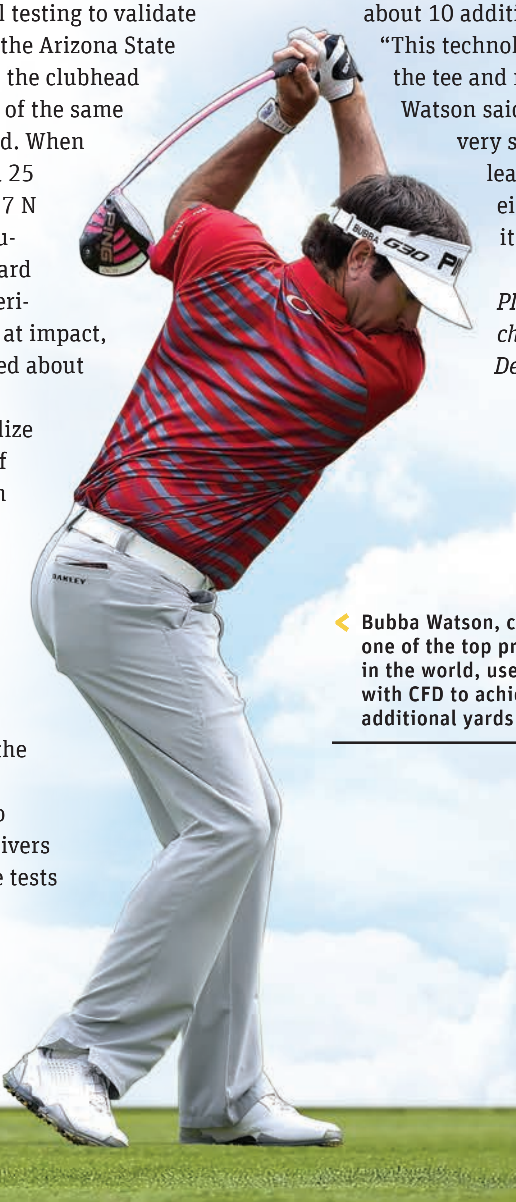
The next step was to perform physical testing to validate the CFD results. PING engineers used the Arizona State University wind tunnel facility to test the clubhead with and without turbulators at some of the same angles and speeds that were simulated. When the driver was square to the airflow, a 25 percent reduction in drag from 9 to 6.7 N was observed for the driver with turbulators when compared with the standard driver. While the standard driver experienced a little over 1.5 N of downforce at impact, the driver with turbulators experienced about 0.5 N of lift.

A smoke stream was used to visualize the flow in the tunnel. Photographs of the smoke showed laminar separation of flow over the standard head at the leading edge of the crown in the driver head without turbulators. In the driver head with the turbulators, flow separation was significantly delayed. The reduction in drag force and delay in flow separation seen in physical testing correlated well with the flow simulations.

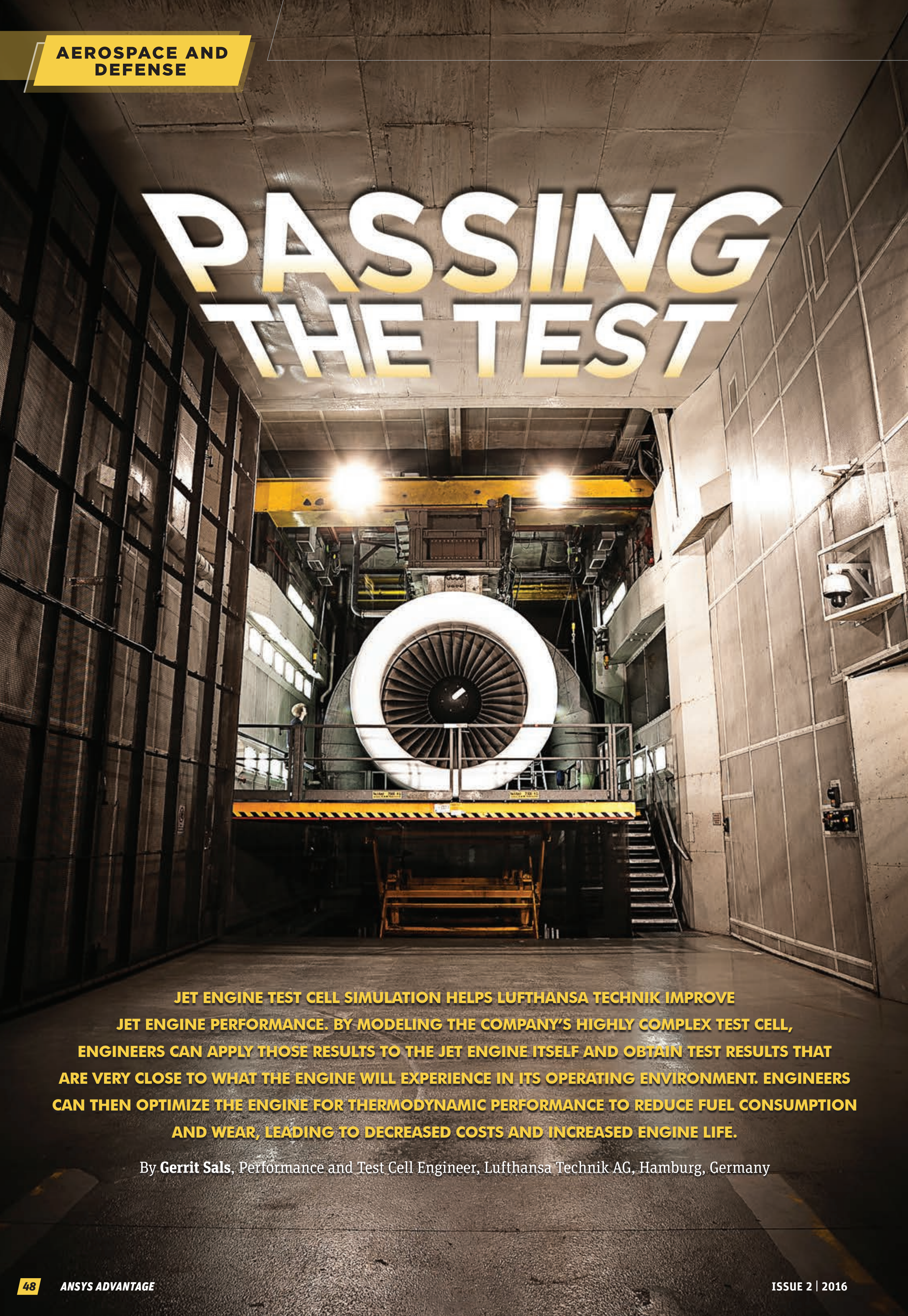
Player tests were also conducted to compare the clubhead speed of the drivers with and without the turbulators. The tests showed that average clubhead speed increased by about 1 mph — from 105 mph with the standard clubhead to 106 mph for the clubhead with turbulators. The ratio of ball speed to clubhead speed is typically between 1.3 and 1.5, which indicates that the increase in ball speed could reach 1.5 mph.

The addition of turbulators to the G30 and G drivers allow them to combine some of the highest moments of inertia of any driver on the market with a high level of aerodynamic efficiency. When Bubba Watson first tested the G30 driver with turbulators, he picked up 2 mph in clubhead speed and 4 mph in ball speed, resulting in about 10 additional yards in driving distance. “This technology has made me longer off the tee and more accurate on the fairway,” Watson said. The G30 driver has also been very successful in the marketplace, leading the industry in sales for eight straight months during 2015, its first year on the market.

PING, Inc. is supported by ANSYS channel partner Phoenix Analysis & Design Technologies (PADT).



Bubba Watson, consistently ranked one of the top professional golfers in the world, uses a driver designed with CFD to achieve up to 10 additional yards off the tee.

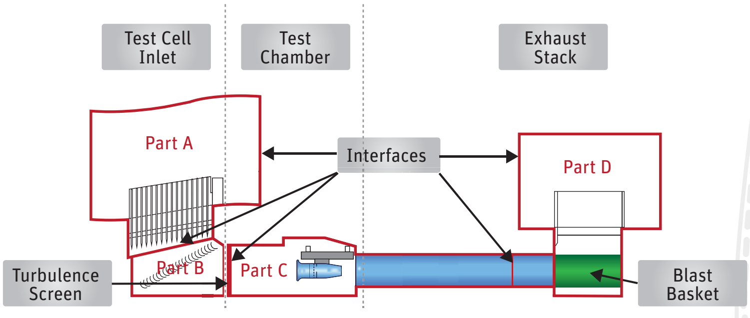


PASSING THE TEST

JET ENGINE TEST CELL SIMULATION HELPS LUFTHANSA TECHNIK IMPROVE JET ENGINE PERFORMANCE. BY MODELING THE COMPANY'S HIGHLY COMPLEX TEST CELL, ENGINEERS CAN APPLY THOSE RESULTS TO THE JET ENGINE ITSELF AND OBTAIN TEST RESULTS THAT ARE VERY CLOSE TO WHAT THE ENGINE WILL EXPERIENCE IN ITS OPERATING ENVIRONMENT. ENGINEERS CAN THEN OPTIMIZE THE ENGINE FOR THERMODYNAMIC PERFORMANCE TO REDUCE FUEL CONSUMPTION AND WEAR, LEADING TO DECREASED COSTS AND INCREASED ENGINE LIFE.

By **Gerrit Sals**, Performance and Test Cell Engineer, Lufthansa Technik AG, Hamburg, Germany

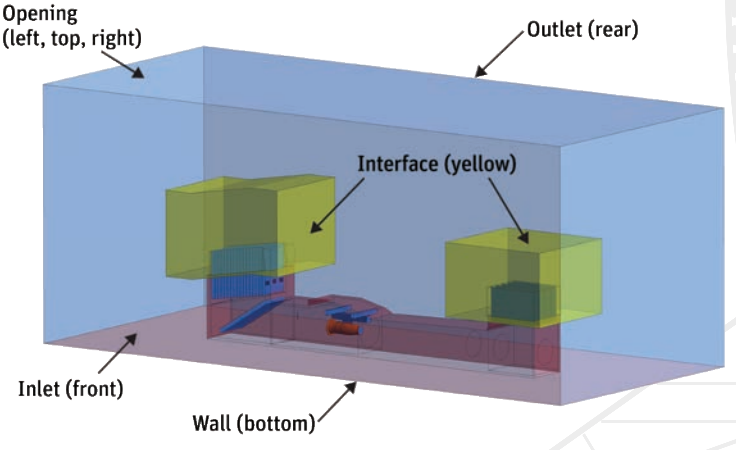
Overhauling a typical commercial jet aircraft engine might cost about $2 million as an expert team inspects and services or replaces up to 40,000 parts. Such an overhaul could be necessary each time the engine flies between 2,000 and 10,000 flights. Overhauls can vary greatly in their work scope, which describes the engine components that are to be serviced or replaced. The work scope is vital because it largely determines the overhaul cost and the performance of the overhauled engine. Lufthansa Technik is improving the engine overhaul process by simulating individual engines at a very detailed level to quantify the relationship between the condition of specific components and the operating behavior of the engine. The insight gained from these simulations allows the team to develop a customized work scope in close consultation with the customer. This work scope might allow engineers to increase the thermodynamic engine performance, which reduces fuel consumption and wear, thereby decreasing future maintenance costs. The understanding acquired from simulation also makes it possible to obtain maximum use from thermo-dynamically as well as economically critical parts, for example, by operating expensive turbine blades for longer periods.



The test cell was partitioned into five models joined with interfaces to enable simulation of the complex model.

Until recently, these simulations were based solely on the engine operating in the air or on the runway, in contrast to jet engine diagnosis and acceptance testing, which is performed in test cells where operating conditions can be significantly different. Lufthansa Technik engineers have long wanted to simulate engines as if they were operating on the company's jet engine test cell. This would require modeling the test cell so the results could be used in modeling the engine. However, test cells are challenging to simulate due to the size and complexity of the geometry, the large range of length and velocity scales present, and flow Mach numbers ranging from near zero to transonic.

Lufthansa Technik engineers have recently overcome these challenges by simulating one of the company's test cells and validating the results against physical testing measurements. Once the team is able to use the test cell simulation results as input to the engine simulation, engineers will be able to better understand the results of diagnostic testing in the test cells, and will also be better able to predict the effects of different overhaul work procedures on acceptance testing. The result should be improvements in engine performance and more accurate overhaul work scoping with resulting cost reductions.



Outer boundary conditions

OPTIMIZING THE OVERHAUL PROCESS

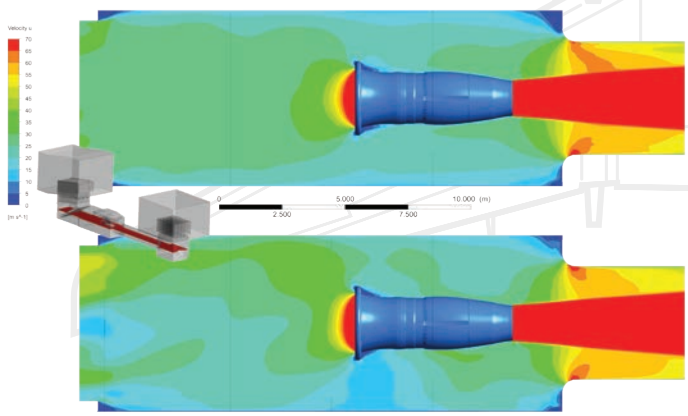
Lufthansa Technik AG is one of the world's leading providers of aircraft maintenance, repair and overhaul services. To improve engine efficiency while avoiding unnecessary work during engine overhauls, detailed knowledge of the internal interactions in the engine is essential. Lufthansa Technik constantly monitors important components so they can be replaced as a function of their condition. Further efficiency improvement can be achieved by precisely determining how the condition of individual components will affect the engine behavior as a whole. By establishing this link between component condition and the operating behavior of the engine, it is possible to target critical components to address during overhaul.

Lufthansa Technik engineers perform three levels of simulation to determine a cause-and-effect link between component condition and engine operating behavior. The highest level is the overall engine level, in which general engine parameters such as thrust, fuel consumption and exhaust gas temperature (EGT) are determined using commercially available thermodynamic cycle analysis software. The second level is a flow simulation of the entire engine based on the multiple mean-line approach. The third level consists of detailed ANSYS CFX computational fluid dynamics (CFD) simulations of sections of the engine.

> **"The understanding acquired from *simulation* makes it possible to obtain *maximum engine* life."**

Recently, Lufthansa Technik engineers set out to further improve this process by simulating the company's test rig to obtain boundary conditions for engine simulations. Internal boundary conditions are derived from the cycle analysis in 95 percent of the cases, which in turn is based on test-cell data. Employing data obtained from a 3-D flow field of the test cell helps the engineers simulate behavior under specific conditions, such as considering the inlet flow of the fan to determine the effects of humidity, rain and crosswinds. This, in turn, enables them to better predict the relationship between component condition and performance on the test cell. Because of the complexity of the test cell geometry, it was split into five models with interfaces between them so the adjoining models provide boundary conditions for each other.

By partitioning the test cell, engineers reduced the model complexity and size, and enabled a modular approach whereby different simulation configurations can easily be constructed by assembling individual components. The CFX flexible general grid interface (GGI) enables such a modular approach. Part A contains the inlet to the test cell and inlet splitters; Part B includes turning vanes; Part C comprises the test chamber, turbulence screen, thrust stand, engine and augmenter tube; and Part D contains the exhaust stack and outlet splitters. The area surrounding the test stands was modeled separately and called the Environment. In addition, the turbulence screen and blast basket were each incorporated into the simulation as subdomains.



^ Axial velocity inside test chamber for static conditions (top) and crosswind (bottom)

MODELING THE TEST CELL

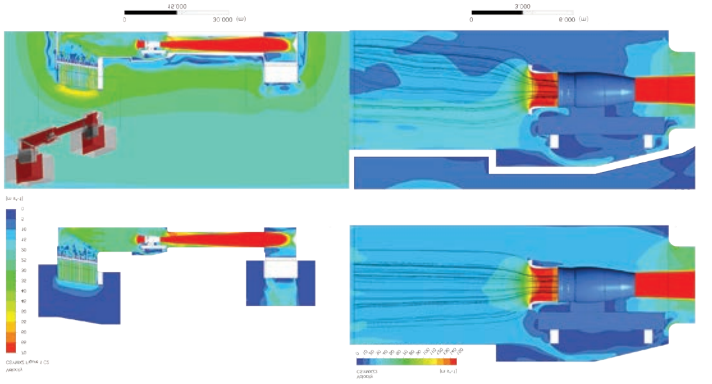
Engineers generated each mesh segment individually using ANSYS ICEM CFD Hexa capabilities, part of ANSYS meshing. Creating the mesh was the biggest challenge in this simulation process. Lufthansa Technik engineers used the mesh diagnostic and repair tools to maintain high levels of mesh quality throughout the mesh generation process. The mesh structure for Parts A, B, D and the Environment was generated as hexahedral H-grids because a hex mesh provides the best trade-off between accuracy and resource requirements. Additionally, small changes can be performed easily. On the other hand, Part C was meshed as a structured hexahedral O-grid for maximum accuracy in this critical section of the model. The interfaces reduced computational time by making it unnecessary to propagate the structured hexahedral O-grid through the turning vane geometry in Part B.

The air enters the test cell through the inlet, where it accelerates when passing through the flow splitters. The turning vanes deflect the vertical flow without significant acceleration. Downstream, the flow passes through the turbulence screen, which leads to a drop in total pressure along with more uniform air flow. The engine then adds energy to the air flow,

Testing the Next Generation of Rockets
ansys.com/testing

increasing the temperature, velocity and total pressure behind the engine. This in turn leads to an acceleration of the air bypassing the engine, which is called the ejector effect. The exhaust gas then leaves the test cell through the augmenter tube, blast basket and exhaust stack.

Engineers simulated the test under two different sets of environmental conditions, which were used as boundary conditions. The first assumed no air movement at the inlet and outlet of the test cell, and the second assumed a 20 m/s crosswind at the inlet and outlet. While different wind directions and speeds are not used in testing, adjustments were made to the CFD model to account for crosswinds, and simulation was used to evaluate those adjustments. The external boundary conditions, which are needed only during the crosswind simulation, include an inlet in front, an outlet at the rear, and openings in the left, top and right of the model. The model's internal outlet boundary (engine inlet) is dependent on the model's internal inlet boundary (engine outlet). The mass flow of these boundaries is coupled through functions based on the static pressure and total temperature at the engine's exhaust nozzle. The functions were derived using thermodynamic cycle analysis. This setup increases the accuracy of the model as the engine changes its operating point according to the test cell flow conditions.



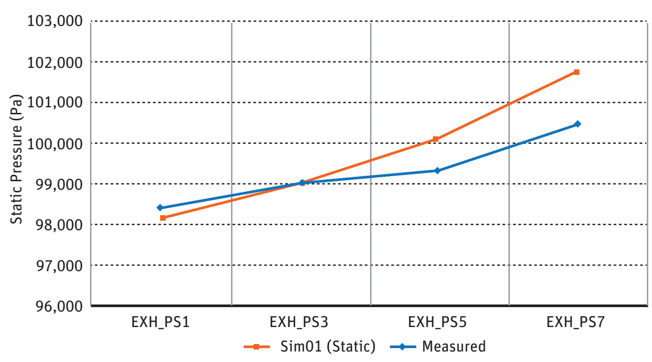
^ Fluid flow in the test cell predicted by simulation for static conditions (top) and crosswind (bottom). This enables engineers to better understand the test cell under real-life conditions to aid jet engine overhaul.

VALIDATING THE SIMULATION

To better understand the test cell results, all that is needed from the test cell simulation is to determine the boundary conditions at the engine inlet and outlet. However, Lufthansa Technik engineers wanted to validate the complete model — including its ability to predict pressures and velocities at any point in the solution domain — so that this information could also be used in evaluating proposed changes to the test cell. The test cell model was validated by comparing simulation results and test cell measurements of static pressure at various points inside the augmenter tube. The deviation between the simulation and test results was very good (from –0.05 percent to –1.33 percent at four different points). However, Lufthansa Technik engineers are working on further improvements in accuracy by refining the mesh in the area of the blast basket and further downstream.

The test cell model will soon be used to provide boundary conditions for engine simulations used as part of the work scoping process for engine overhauls. Accurate engine-in-test-cell simulation will help engineers further improve the performance of overhauled engines and refine the work scoping process with the potential for significant cost savings. For example, the customer may specify that the overhauled engine must provide a certain EGT on the test cell. Lufthansa Technik engineers will be able to better evaluate the impact of different possible work scopes on the EGT as measured on the test stand. In addition, the test cell model will be used to improve the test cell design and evaluate the impact of different sensor placements in specific tests.

Using simulation, Lufthansa Technik will not only improve jet engine performance for customers but fine-tune internal processes to reduce costs. Simulation accuracy reduces risk and makes the company more competitive. ^



^ Comparison of simulated and measured pressure inside the augmenter tube shows acceptable agreement.

AEROSPACE AND DEFENSE

CALM LANDING

Performing flight tests that include water landings of unmanned aerial vehicles is cost-prohibitive. Simulation of this challenging landing maneuver that includes multiphase flow, compression of water and small computational time steps saves physical testing time and costs.

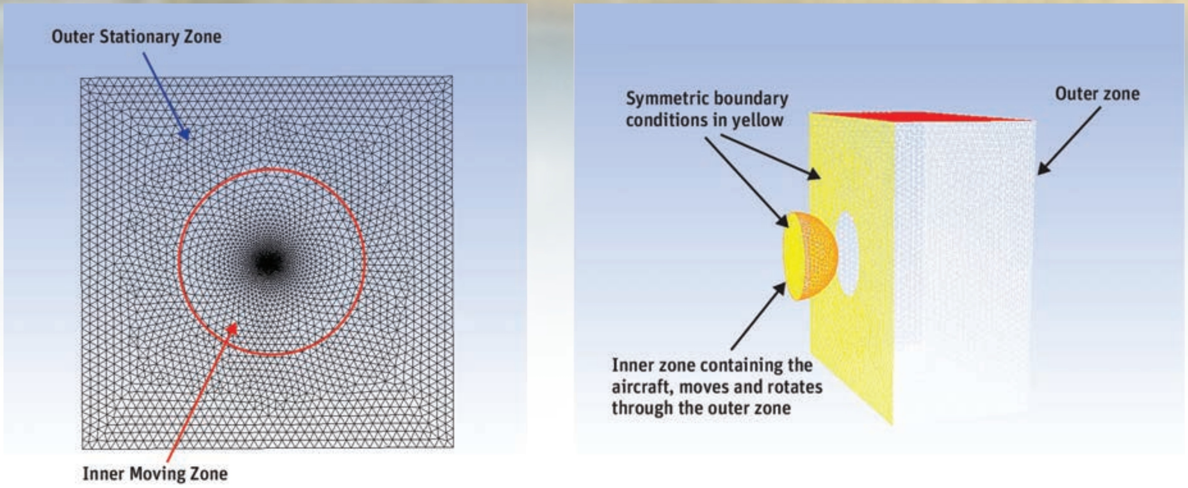
By **Keen Ian Chan**, Principal Engineer, Singapore Technologies Aerospace, Singapore

Unmanned aerial vehicles (UAVs) are being tasked to complete an increasingly diverse set of missions. These can include flying over large bodies of water to perform operations such as maritime surveillance.

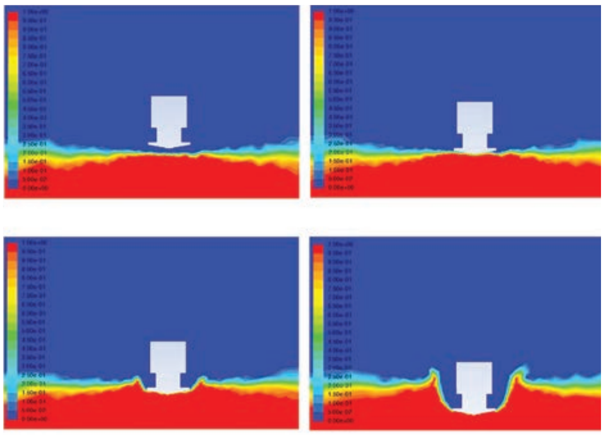
Depending on the UAV's size and its payload, an unplanned water landing, or ditching, can cause damage costing thousands or millions of dollars and even result in the loss of the entire system. For example, impact with water at speed generates large transient pressure loads on the air frame, and the natural properties of the water (dynamic buoyancy and compressibility) may cause the UAV to tumble. Either eventuality can cause airframe failure and break-up. Understanding how to mitigate such scenarios is therefore an important design consideration for UAVs.

However, performing flight tests of a water-landing maneuver for a new UAV design is not practical because of the time and cost involved to build prototypes, arrange airspace clearance, extensively instrument the test aircraft, and understand and replicate the sea state and environment in which the impact occurred.

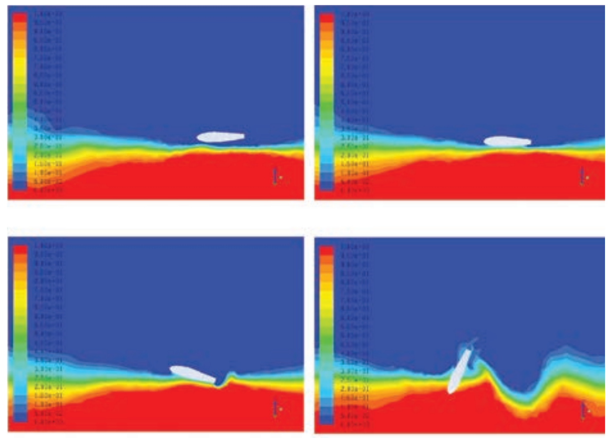
Simulation of water-landing scenarios is a practical alternative to extensive flight testing, but it can be challenging because engineers need to consider multiphase flows (air and water), the compressibility of water, and the very small computational time steps



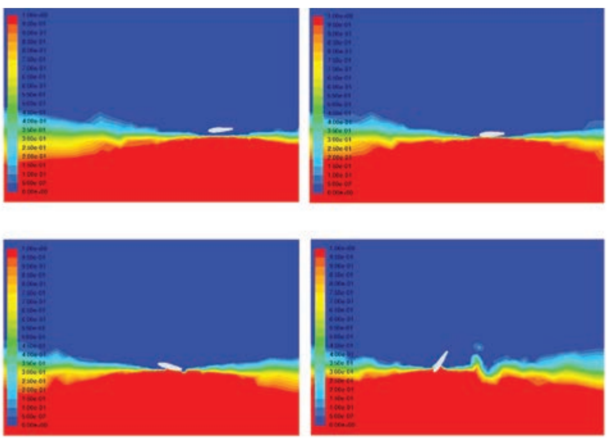
^ Engineers were able to reduce time step size by dividing fluid domain into two zones.



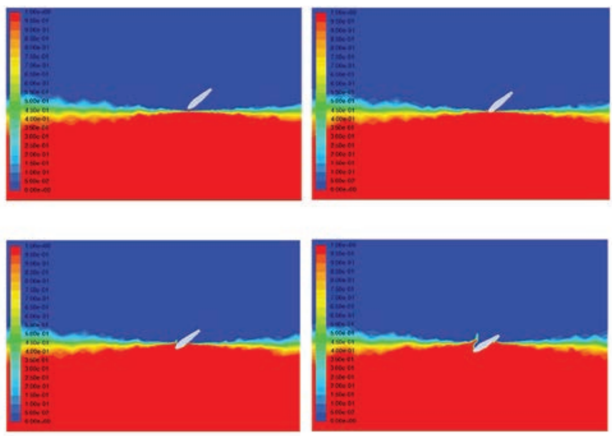
Validation of the simulation method



Steep descent landing shows undesirable tumbling behavior.



Belly landing maneuver simulation reveals undesirable tumbling behavior.



Nosedive landing maneuver simulation with desirable results

required to capture impulse loading. Singapore Technologies Aerospace (ST Aerospace) engineers used ANSYS CFD software to overcome these challenges and accurately simulate a wide range of water-landing scenarios. This saved a large amount of time and money.

MULTIPHASE FLOW

ST Aerospace is an integrated service provider that offers a wide spectrum of maintenance and engineering services to a customer base that includes the world's leading airlines, airfreight and military operators. To capture the multiphase properties of the flow fields in water impact simulations, ST Aerospace engineers used the volume of fluid (VOF) model in ANSYS Fluent. In this model, the volume fraction of each phase, which is defined as a fraction of volume occupied by that phase in a computational cell, is tracked throughout the domain, and the interface between phases is captured simultaneously. The geometric reconstruction interface-capturing scheme used in this study computes the evolution of the water surface by representing it using a piecewise–linear approach. This scheme is most accurate and compatible with unstructured, moving and deforming meshes (MDMs).

The pressures generated during water impact are large enough to compress seawater, so the compressibility of water must be included in the simulation. During the simulation, a user-defined function (UDF) calculates the compressibility of water by determining its density based on its bulk modulus, which is defined in terms of pressure and density change.

DIVIDING FLUID DOMAIN TO LENGTHEN TIME STEPS

To simulate the aircraft moving relative to adjacent cells, the time step needs to be small based on the fine adjacent grid resolution. In this case, engineers were able to increase

> **"Performing *flight tests* of a water-landing maneuver for a *new UAV design* is not practical."**

the time step size by dividing the fluid domain into two zones. An inner hemispheric zone contains the aircraft and remains fixed relative to the aircraft, so that as the aircraft moves and rotates in response to forces generated by water impact, the inner zone also moves and rotates. The outer zone is stationary and fixed in space. This is accomplished in ANSYS Fluent using the MDM modeling approach. MDM efficiently re-meshes the volume cells at the interface of the two zones as the inner zone moves through the outer zone as the computation progresses. The time step size is based on the larger volume cells at the interface of the two zones, rather than the much smaller cells directly adjacent to the aircraft, enabling larger time steps to be used and greatly reducing the number of time steps required to complete the simulation.

Engineers used symmetry boundary conditions in the CFD model so that only half of the aircraft was modeled. This halved the number of volume cells and reduced the computational time by 50 percent. A limitation of this approach is that pitching motion can be captured but rolling and yawing motions cannot.

The water impact simulation starts with the aircraft a short distance above the water and proceeds in small time steps. At each time step, CFD simulations are performed to resolve the flow field at that instant. The flow field yields the forces and moments acting on the aircraft. The forces and moments are input to Fluent's built-in six degree of freedom (6DOF) solver to compute an incremental translation and rotation for that time step. The UAV is moved to the new position and orientation, carrying the inner fluid zone with it. The movement of the aircraft and body-fixed inner zone distorts the volume cells at the boundary with the outer fluid zone. Regions of distorted cells are re-meshed by MDM to maintain good quality. The cycle is repeated for each successive time step.

VALIDATING THE METHOD

ST Aerospace engineers validated their computational approach by simulating a published experimental test case [1]. The case involves dropping a 160-degree cone into the water at different masses and impact velocities. The impulse forces upon impact were measured. Simulations were performed for the case of a 0.324 kg mass impacting the water at 5.04 m/s. The experimental measurements showed a peak force of 317.844 N while the simulation showed a peak force of 310.977 N, a difference of only 2.2 percent.

EVALUATING DIFFERENT WATER-LANDING APPROACHES

With the simulation method validated, ST Aerospace engineers ran 20 different water-landing simulation cases for the new UAV. The team simulated steep-descent landings, belly landings and nosedive landings. They also modeled a belly landing in which the UAV's belly was replaced with a NACA 84 flying boat hull.

The steep descent, belly landing and flying boat hull landings all showed tumbling behavior, which is an undesirable result because it increases the forces on the UAV. The nosedive landing, on the other hand, was free of tumbling behavior and provided the lowest forces. Images of the water landing are as seen from the symmetry plane of the UAV, extracted from animations of the simulations.

The CFD simulation of the UAV landing on water yielded valuable results and insights that were used in the airframe's structural design to enable it to withstand impact with the water. The results will also be valuable for UAV operators to determine the best procedure to execute a water-landing maneuver. These solutions were achieved without having to embark upon a costly and high-risk flight test campaign, thus substantially reducing the time and cost required to design the UAV.

Singapore Technologies Aerospace is supported by ANSYS channel partner CAD-IT Consultants (Asia) Pte Ltd.

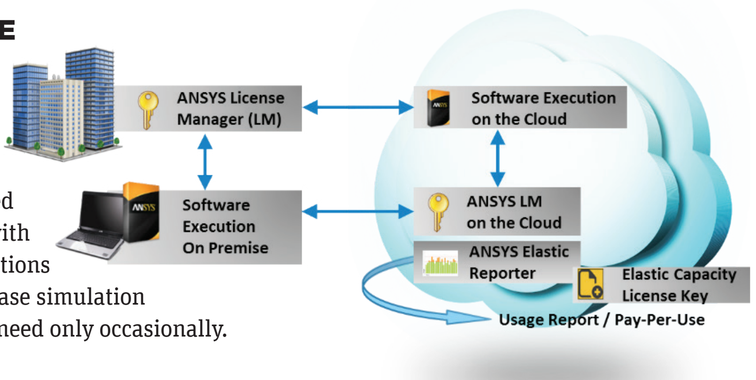
References

[1] Watanabe, S. Resistance of Impact on Water Surface, Part I – Cone, *Inst Phys Chem Res Tokyo*, **1930**, Vol. 12, No. 226, pp. 251–267.

Simulation in the News

ANSYS'S CAE SIMULATION SOFTWARE HAS A NEW PAY-PER-USE LICENSE

engineering.com, March 2016
ANSYS has released a new pay-per-use licensing plan that enables customers to access its computer-aided engineering (CAE) software on an hourly basis. Offered alongside the traditional lease and paid-up models, with cloud options, ANSYS Elastic Licensing gives organizations the flexibility of buying a pay-per-use license to increase simulation capacity at peak usage times, or to use software they need only occasionally.



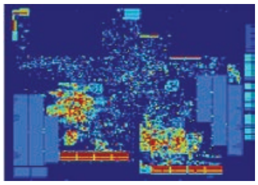
"We expect that elastic licensing will give us access to the broad suite of multiphysics solutions that we need to support our customers globally, and bring our innovative products to market faster and more reliably than ever."
— **Sukhvinder Kang**, *CTO, Aavid Thermalloy*

RED BULL RACING: DYNAMIC ENGINEERING

themanufacturer.com, April 2016
Before any new component is physically prototyped or is used in the manufacture of Red Bull Racing's Formula 1 cars, it is evaluated in virtual 3-D by supercomputers. ANSYS's computational fluid dynamics (CFD) and high-performance computing (HPC) software enables Red Bull Racing to simulate the airflow around its vehicle designs under a range of conditions. In this way the team can assess, select and optimize designs for aero-critical components and assemblies, such as braking, cooling and exhaust systems.

ANSYS QUALIFIES FOR SAMSUNG FOUNDRY CHIP DESIGN

engineering.com, April 2016



ANSYS and Samsung Foundry, a full-scale foundry solution provider, have collaborated to create process design kits and electromigration flow tools to help engineers design system-on-chip (SoC) integrated circuits. The certification ensures that the simulation software meets the accuracy and reliability requirements for 14nm and 10nm fin field-effect transistors (FinFETs). These new simulation tools will help engineers designing devices for HPC, mobile devices, automotive electronics and other IoT applications.

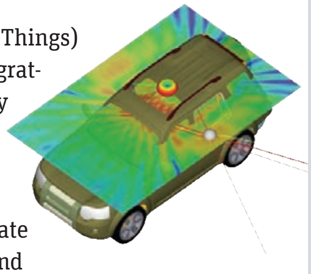
TE CONNECTIVITY ANNOUNCES DEAL WITH ANSYS

bizjournals.com, March 2013
TE Connectivity, designer and maker of connectivity and sensor solutions, signed an agreement that enables Switzerland-based TE to leverage ANSYS software in 24 countries to optimize and verify product design while shortening product cycles and reducing the need for physical prototypes and testing.

ANSYS EXPANDS ANTENNA SIMULATION PORTFOLIO

engineering.com, April 2016

ANSYS is helping engineers simulate IoT (Internet of Things) devices and large-scale antenna installations by integrating the latest releases of antenna and radio frequency (RF) simulation software Savant 5.0 and EMIT 5.0 into its simulation environment. The two software products, acquired from Delcross Technologies, work with ANSYS HFSS. Savant enables engineers to simulate antenna performance on electrically large platforms and in complex environments, such as cars, airplanes and ships. EMIT lets engineers create RF simulations that identify where RF interference can affect their designs.



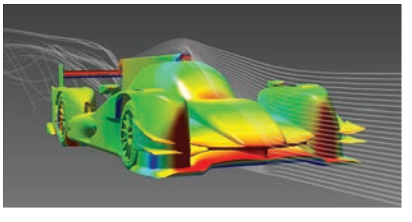
ENGINEERING SIMULATION MOOC TEACHES PRO SKILLS

news.cornell.edu, March 2016

Cornell University is offering a massive open online course (MOOC) that gives thousands of students around the world the opportunity to learn engineering simulation skills using the latest, industry-standard ANSYS simulation software. The course, titled "A Hands-on Introduction to Engineering Simulations," will teach students how to create physics-based simulations of their designs before building a physical prototype. It will also teach high school students and college undergraduates about the mathematical models on which simulations are based, along with fundamental principles of structural mechanics and fluid dynamics. Both the six-week course and the software are free to students, teachers and industry professionals through this collaboration between ANSYS and Cornell.

ORECA OPTIMIZES RACE CAR DESIGN WITH ANSYS SIMULATION TECHNOLOGY

itbusinessnet.com, February 2016



The motorsport company ORECA has signed a new agreement to use ANSYS software to enhance its race car designs, including their future car, the ORECA 7. ORECA will take advantage of ANSYS's high-performance computing (HPC) and CFD software to simulate airflow around the cars under a wide range of conditions and enable their engineers to test multiple designs before physical testing.

ANSYS AND CARNEGIE MELLON UNIVERSITY

EurekAlert!, June 2016

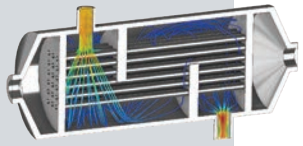
Future Carnegie Mellon University engineers will design new innovative products more efficiently and effectively, thanks in part to a collaboration with ANSYS. The partnership brings together two world leaders in engineering, computer science and simulation technologies to impact the future of engineering education and research. ANSYS and Carnegie Mellon want to boost engineers' use of simulation to enable unparalleled opportunities for exploration of many more materials and designs at the beginning of the development process.

Carnegie Mellon University

ANSYS 17.1 EXPANDS SYSTEM SIMULATION AND IMMERSIVE MULTIPHYSICS

deskeng.com, May 2016

With the release of ANSYS 17.1, engineers now have access to systems simulation capabilities within ANSYS flagship products to precisely analyze how component physics and embedded software affect overall systems design. ANSYS AIM features extended structural and fluid flow simulation capabilities, and now provides product designers with magnetostatics and coupled magnetic–thermal–structural analysis to design innovative electromechanical products. ANSYS 17.1 delivers advancements across the entire portfolio and ANSYS Workbench platform.



BIG DATA AND MACHINE LEARNING SYSTEM FOR ENGINEERING SIMULATION

eurekamagazine.co.uk, May 2016

Engineering simulation generates tremendous amounts of data. ANSYS SeaScape allows organizations to innovate faster than ever by bringing together the advanced computer science of elastic computing, big data and machine learning and the physics-based world of engineering simulation. ANSYS SeaHawk gives electronic system designers an innovative new approach to reduce the size and power consumption of next-generation semiconductors. SeaHawk is the first application built on the SeaScape architecture, It complements the other ANSYS power noise sign-off and reliability tools.

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

Send address corrections to
AdvantageAddressChange@ansys.com