COMSOL NEWS

THE MULTIPHYSICS SIMULATION MAGAZINE

Engineering the Perfect Snack
Cornell University
PAGE 7

THE RUMBLE AND ROAR OF MAHINDRA **MOTORCYCLES**
PAGE 14

Custom designed hi-fi transducers

at Xi Engineering and WAT

PAGE 4

COMSOL

Surpass Your Design Challenges with Custom Simulation Apps

Given the complexity of today's products and the many competing requirements designers are faced with, achieving an optimal design requires a company-wide collaboration to succeed.

This is what emerged as the leading theme of this year's edition of COMSOL News: design teams are reaching optimal solutions faster by involving other departments in their organization. But how? Simulation specialists are developing custom simulation apps with the COMSOL Multiphysics® software and deploying them with the COMSOL Server™ product. Engineers from manufacturing and sales can easily access high-fidelity multiphysics simulation results through app libraries developed in-house with their specific needs in mind. Without being experts in mathematical modeling, they have access to the most powerful computational tools. They can experiment with parameters affecting form and function. They can suggest a design iteration based on their specific skill set and customer feedback.

Thanks to the adoption of simulation apps across departments, the design workflow is streamlined and inclusive. Mathematical modeling and numerical simulation is accessible to the largest group of users ever. The knowledge of an entire organization is put to work to create the best products. Learn about custom designed hi-fi transducers at Xi Engineering and WAT, motorcycle development at Mahindra Two Wheelers, and food science at Cornell University. Join the users of COMSOL® software as they inspire you to achieve your best design yet.

Enjoy!

Valerio Marra
Marketing Director
COMSOL, Inc.

INTERACT WITH THE COMSOL COMMUNITY

COMSOL, Inc. | COMSOL Multiphysics | @COMSOL_Inc | plus.google.com/+comsol

BLOG comsol.com/blogs
FORUM comsol.com/community/forums

We welcome your comments on COMSOL News; contact us at info@comsol.com

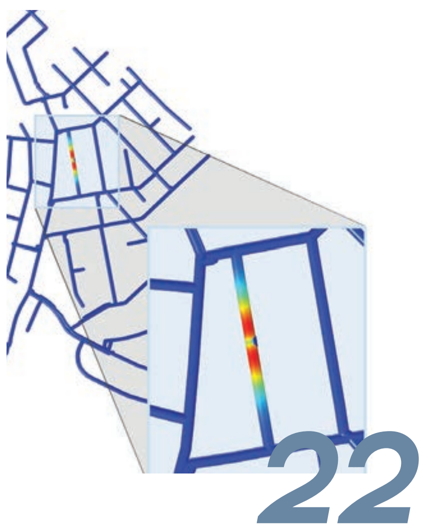
COMSOL NEWS
2017

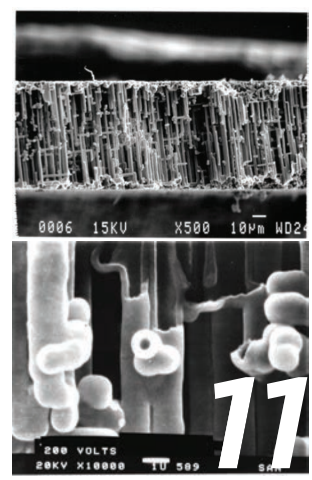
CONTENTS

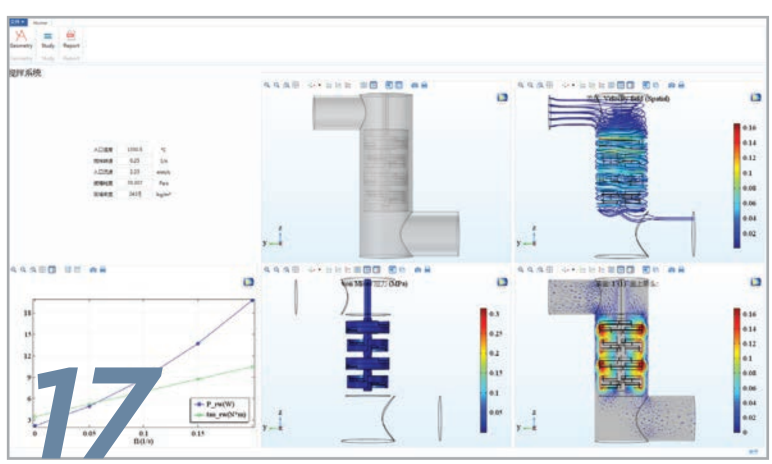
ON THE COVER

HPEL (High-Precision Electrostatic Laminate) ultra-high performance headphones and fully integrated audio drive system. Image credit: Warwick Audio Technologies Ltd.

FEATURES







Music to Your Ears: New Transducers Meet Electrostatic Headphones

An audio technology startup delivers new manufacturable transducers for high-end electrostatic headphones and reduces low-end roll-off.

by **JENNIFER HAND**

Serious hi-fi enthusiasts get excited about the musical experience delivered by electrostatic headphones. Producing a natural, airy sound, they provide greater clarity, less distortion, and extended bandwidth when compared to other types of headphones where high resolution audio sources are involved.

Most electrostatic speakers apply an electric charge on a thin elastic membrane situated between two conductive plates. The charged membrane moves in direct response to the electrical input, generating the sound waves that our ears and brain interpret as music, and moving us to joy and tears.

Despite their high quality and accurate audio reproduction, electrostatic speakers can be prohibitively expensive, sometimes fragile, and until recently, were handmade because of mechanical precision requirements. Seeing a need for affordable, high-quality headphones that could be manufactured more easily, Warwick Audio Technologies Limited (WAT) designed the High-Precision Electrostatic Laminate (HPEL) transducer, a patented technology based on an ultrathin diaphragm and a single conductive plate instead of a pair. With its origins at Warwick University in the UK, WAT has developed a lightweight laminate membrane only 0.7 mm thick that is perfectly suited for electrostatic headphones.

The new HPELs are lightweight thin-film structures manufactured through a continuous roll process. “The technology we’ve developed is unique,” explains Martin Roberts, CEO of WAT. “The HPEL transducer is made up of a metallized polypropylene film, a polymer spacer with hexagonal cells, and a conductive mesh" (Figure 1).

IIn the typical setup, direct current (DC) bias voltage is applied to the elastic membrane and alternating current (AC) drive signal to the surrounding plates. WAT's one-sided speaker involves both the DC bias and the AC drive signal applied to the elastic membrane, with a single wire mesh (plate) positioned opposite the membrane as a ground plane.

The fabrication method makes it possible to reproduce the transducers at a significantly lower cost than traditional electrostatic speakers. This means that for the first time, electrostatics may be considered a commercially viable high-res audio option across a wide range of device types and market segments.

⇨ SIMULATING ACOUSTIC PLAYBACK

To develop a transducer like this, which can be easily manufactured and inexpensive without compromising sound quality, the WAT team thoroughly investigated the influence of many design elements before settling on a final version. “We had developed numerous prototypes that clearly performed. The big issue was that we were not entirely sure how varying individual material and design parameters affected the transducer’s performance,” Roberts says.

The dynamics of the HPEL are dependent on the extremely complex interplay between membrane tension, AC signal level, speaker geometry, elastic and dielectric material properties, thermoacoustic losses, and the added mass effects of the air next to the open side of the membrane. The designers wanted to improve bass performance by reducing low-end roll-off, minimizing distortion, and maximizing the sound pressure level for a given electrical input. But they discovered that small changes to any component greatly affected the acoustic output.

> “We went from making multiple prototypes by hand each week to simply dialing up a new one in the software. In addition to settling on a final design we’re very happy with, it is now easy for us to customize our transducers for clients’ custom requirements.”
>
> — MARTIN ROBERTS, CEO, WAT

Although WAT had significant mechanical, electrical, and acoustic expertise, they had no in-house simulation capability to help them understand this interplay. In order to perform a virtual optimization of the

FIGURE 1. Top to bottom: WAT's HPEL transducers; single laminate, assembled, and exploded views of a finished HPEL transducer. All laminates are made in the UK.

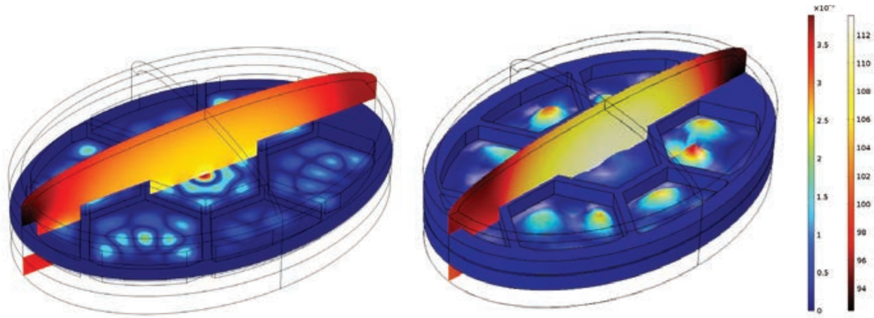


FIGURE 2. Simulation plot showing the sound pressure level (thermal color surface) in dB and the displacement of the membrane (rainbow color surface) in mm from a fully coupled acoustics-MEMS model solved in the frequency domain. Left: solution at 5,000 Hz. Right: solution at 5,250 Hz.

HPEL transducer design they enlisted the help of Xi Engineering, a COMSOL Certified Consultant that specializes in computational modeling, design recommendations, and solving noise and vibration problems in machinery and other technology.

Dr. Brett Marmo, technical director of Xi Engineering, oversaw the development of the COMSOL Multiphysics® software models they used to analyze the behavior of the HPEL. COMSOL® software, which allowed Xi Engineering to model nonlinear effects that would arise with amendments to the HPEL's asymmetrical design.

"We kept the early model simple, focusing on specifics that influence sound quality, for example keeping the first harmonic as low as possible to understand the acoustic-structure interaction and the HPEL's performance at low frequency," Marmo explains, describing their preliminary tests. "Our model showed how applied voltage affects signal levels, which helped us understand sound distortion for an initial case."

Because the transducer is one-sided, the electrostatic force varies with the position of the vibrating membrane, decreasing with the square of the distance between the membrane and the mesh. Once they understood the resulting nonlinear distortion and were able to predict its effects, the WAT engineers could then cancel any related distortions electrically.

⇨ PERFECTING THE HPEL TRANSDUCER DESIGN

In a more extensive simulation that involved a structural-MEMS-acoustic coupling, he examined the impact of adjusting parameters like the size of the hexagonal cells in the wire mesh, thickness of the wires, membrane tension, spacing between membrane and mesh, and material properties of each component. Marmo and his colleagues also studied the effects of different DC biases, which are often responsible for distortion at low frequencies, and looked at conductivity along the plate to discern whether voltages were higher in one area than another. They then used COMSOL to study the thermoacoustic losses and model the displacement of the membrane for different frequencies (Figure 2).

"We found that this type of simulation was the only accurate way to truly model planar electrostatic transducers," Marmo continues. "For this case, lumped parameter modeling can characterize limited aspects of performance, such as low-frequency amplitude response. One parameter might be excellent but there may be significant distortion created elsewhere. Multiphysics modeling encompasses all dimensions that affect our perception of sound, such as the time-domain response and nonlinear distortion."

The simulations made it possible for the engineers at WAT to tweak design parameters in order to optimize overall performance. Ultimately, they were able to predict what was causing spikes in the frequency response and smooth out the signal for better fidelity.

"This represented a huge cost and time benefit for us," says Roberts. "We went from making multiple prototypes by hand each week to simply dialing up a new one in the software. In addition to settling on a final design we're very happy with, it is now easy for us to customize our transducers for clients' individual requirements."

Marmo's team compared each model with physical measurements provided by the WAT design team. "The simulation results were astoundingly close to the physical measurements," comments Dan Anagnos, CTO at WAT. "That was probably the most exciting aspect, seeing the simulation come to life and knowing it was giving us an accurate picture of how the speaker would perform."

⇨ FREEDOM AND FLEXIBILITY WITH A SIMULATION APP

With simulation results verified and WAT satisfied with their design, the next step was for Xi Engineering to put WAT in control of further modeling. The Application Builder available in COMSOL software enabled Marmo's team to build an app from their simulation and host it online.

The app's interface allows users to change certain inputs to test changes to a number of parameters, such as the DC bias, AC signal level, frequency range and resolution, material properties, speaker size, wire mesh shape and size, and spacer placement (Figure 3). The original model setup is not accessible from the app; instead, it allows users to run further tests without needing to learn the software.

"Providing WAT with a simulation app removed the need for them to purchase the software or appoint an experienced user," Marmo says. "Simulation apps put our customers in charge, so they don't have to come back to us for small changes and they can test exactly what they want. It also frees us to explore new challenges, rather than working on variations of the same problem." Xi Engineering expects to use computational apps more and more in the course of its work for other customers.

WAT is doing the same, sharing the app with their own customers — companies wanting to find the HPEL transducer best suited to their particular headphone designs. "The team at Xi Engineering have been superb. They have deep expertise and helped to unpack the complexity of our product," adds Roberts. "The intuitive app that Xi developed for us is an additional bonus. Without revealing any intellectual property we can give our own clients access to our design through the app, so they can test and incorporate the technology into their own high-end headphones." ❖

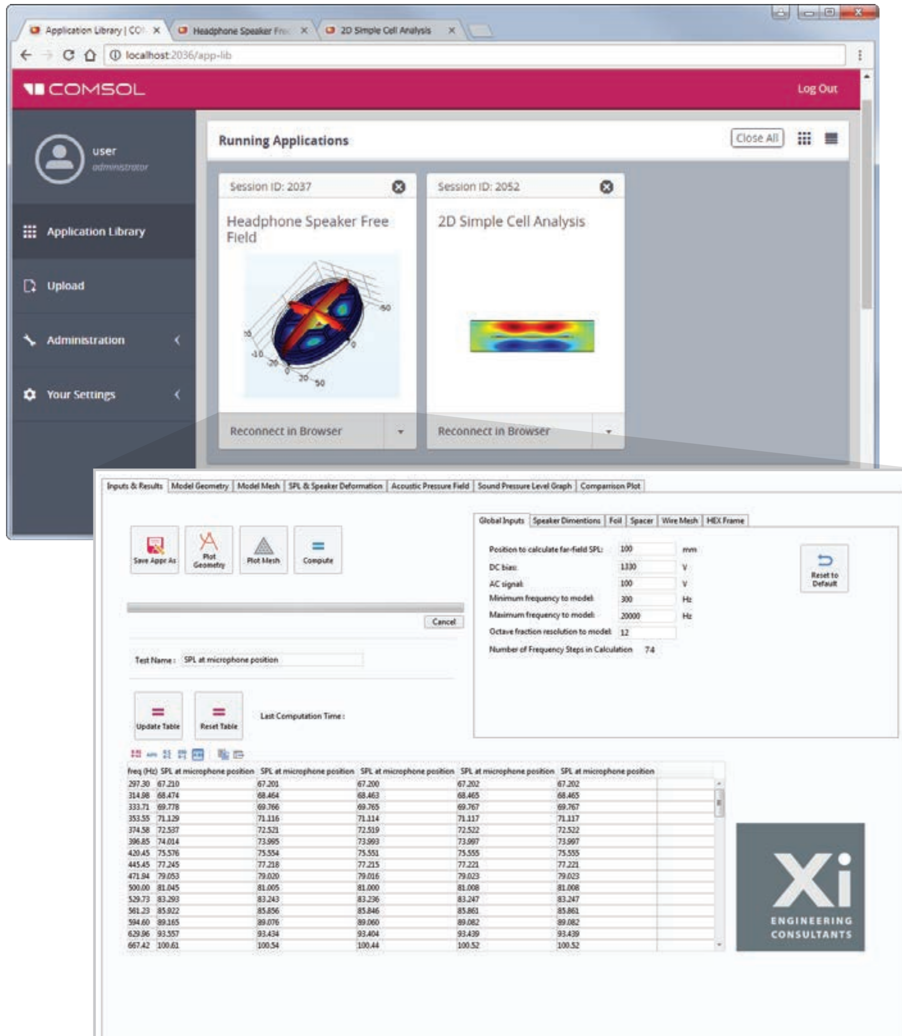


FIGURE 3. Foreground: The app developed by Xi Engineering allows engineers to vary parameters related to frequency, electrical input, speaker dimensions, and properties of the membrane, spacer, and wire mesh. Results give the sound pressure levels for different cases, membrane displacement, frequency response to different DC biases, and a comparison of the simulated design against experimental results. Background: The app is shared through the COMSOL Server™ product and accessible from a web browser.

Left: Brett Marmo, technical director at Xi Engineering.
Center: Martin Roberts, CEO, Warwick Audio Technology.
Right: Dan Anagnos, CTO, Warwick Audio Technology.

ENGINEERING PERFECT PUFFED SNACKS

A Cornell research team supports the food industry with mathematical models of rice puffing.

by **LEXI CARVER**

FIGURE 1. Parboiled rice, unpuffed (top) and puffed (bottom).

A common snack in parts of Asia for centuries, puffed grains have become a staple in mass-produced cereals and snacks on grocery store shelves around the world. The delightful crunch of rice cakes, varieties of puffed corn, and crispy bites in chocolate desserts is familiar (and delicious) for many.

Also familiar is the less desirable sensation of biting into a puffed snack and discovering that it is too soft, too chewy, too dry, or slightly soggy straight out of the bag. What causes these abrupt mishaps?

What happens inside a rice kernel during puffing, for instance? To a casual observer watching the process, a single piece would heat up and then suddenly and explosively changes shape, like popcorn (Figure 1).

But the physics of rice puffing involves an incredibly complex interplay of mass, momentum, and energy transport; rapid water evaporation; material phase transition; pressure buildup; and plastic deformation.

Food companies have put in many hours working to achieve the right moisture and texture in puffed food that will keep customers happy. They've worked to create reliable processing conditions so that the occasional rubbery piece is an anomaly and not the norm. For scaling up puffing methods for production, food companies need to optimize processing for consistent texture, flavor, moisture content, and in some cases, food safety.

⇨ RESEARCHING OPTIMAL PROCESSING CONDITIONS

Using a research grant from the United States Department of Agriculture (USDA) Agriculture and Food Research Initiative (AFRI) program, Cornell University has performed a study of the transport processes in deformable porous media with phase-dependent properties, with a focus on food. Prof. Ashim Datta, from the Department of Biological and Environmental Engineering, led a team to model the dynamics and material behavior during the puffing of parboiled rice[1].

In addition to studying the intricacies of phase change, energy transfer, and mechanical behavior during puffing, their extensive investigations looked into the effects of salt preconditioning, temperature, and initial moisture levels to facilitate the desired final texture.

At the core of their research was the need for a modeling methodology that would be transferrable to many scenarios. "We built a framework for studying the physics of food processes and made it applicable to different problems; for example, frying incorporates a certain set of physical phenomena, while baking involves a somewhat different set but within the same framework," explains Prof. Datta. He elaborates on the particular concerns of the food industry: "Consumers want the texture of fried food, but without the health cost; the same quality, but not the same method.

"So food companies looked into baking and 'popping' as an alternative to frying. They are constantly updating products and processes. When they change something, they have to know the new optimal conditions. The framework we developed allows us to swap conditions more easily to test the effects of different processes on the final food product.

"Once we know how various combinations of temperature and moisture for one way of processing lead

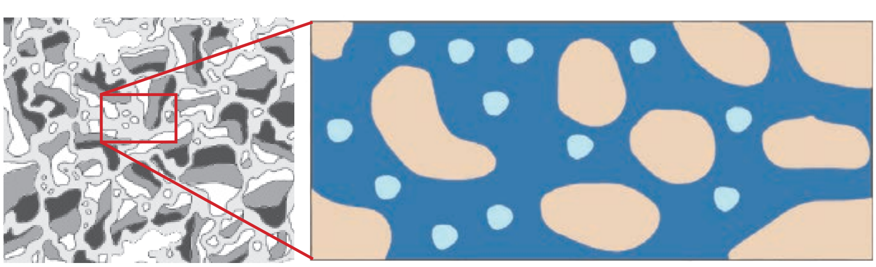


FIGURE 2. Depiction of rice as a porous, elastoplastic solid. The kernel contains liquid water subject to capillary diffusion, convection, and phase change (dark blue); gas composed of water vapor and air, subject to bulk flow, binary diffusion, and phase change (light blue); and a solid starch skeleton that undergoes large deformations (beige).

to certain mechanical properties, we can see whether other processing routes will produce the same food quality. We wanted to determine how different processes affect texture, water or oil content, and even the corresponding health implications."

One of the biggest challenges facing the research team was the fact that so many different factors influence the final state of the food. Heating a parboiled rice kernel to temperatures of 200°C leads to the rapid evaporation of liquid water, resulting in large gas pressure buildup and a phase transformation in the grain. It transitions quickly from a rigid, glassy state to a soft, compliant (rubbery) one that allows the kernel to balloon into its final shape. Heating time and initial water and salt content also play a deciding role.

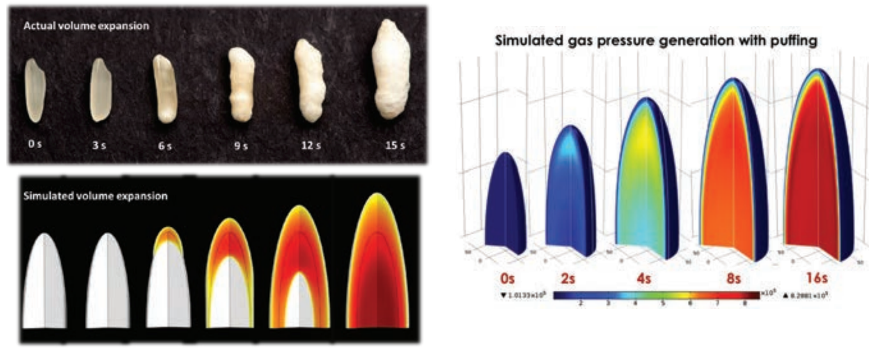


FIGURE 3. Left: Measured volume expansion and simulated volume expansion during a 15-second puffing sequence. Right: Simulation showing gas pressure generation.

⇨ MODELING INTERCONNECTED PHYSICS

In order to understand how these factors work together and hone in on the ideal processing conditions, Tushar Gulati, (a student of Prof. Datta at the time), headed up the work to break down the mysteries of rice puffing.

He used the COMSOL Multiphysics® software to analyze the interconnected mechanical, thermal, material, and fluid behavior within a puffing parboiled rice grain.

"Numerically, this is a very challenging problem," Prof. Datta commented. "The team studied flow through the porous medium, multiphase transport, solid mechanics, heat transfer, and in other situations incorporated the electromagnetic behavior involved in microwave heating."

> **"Simulation apps bring new opportunities to education. In a food safety class, apps enable multidisciplinary learning where students can simulate many what-if scenarios realistically."**
>
> — PROF. ASHIM K. DATTA, CORNELL UNIVERSITY

Gulati built a multiphase porous media model to study the mass and momentum changes, energy transport, and large volumetric expansion. The model analyzed the different phases of solid rice, liquid and gas water, and moisture transport modes such as capillary flow, binary diffusion, and pressure-driven flow. He assumed the rice to be an elastoplastic material and obtained mechanical displacement and expansion.

The corresponding simulation revealed the spatial and temporal distributions of temperature, moisture, pressure, evaporation rates, volumetric strain, porosity, and stress levels at different times during puffing (Figures 3 and 4).

The team validated the computational model using a reconstruction of micro-CT images used to determine the expansion ratio and visualize the microstructure development. Gulati also found that the expansion ratio was sensitive to evaporation rates and the intrinsic permeability of the modeled solid matrix.

By the end of his work, they had a fully coupled model that linked the different behaviors occurring during puffing, including the phase change. Gulati coupled the transport model to the large deformation, and also tested how different levels of salt affected volumetric expansion, evaporation, and material properties. Salt lowers the glass transition temperature, meaning that the rice puffs more quickly and at lower temperatures.

"The simulation illustrated how properties vary within the rice grain initially, as well as how they change over time during heating," Prof. Datta adds. "This would be impossible to measure experimentally. The model tells how the

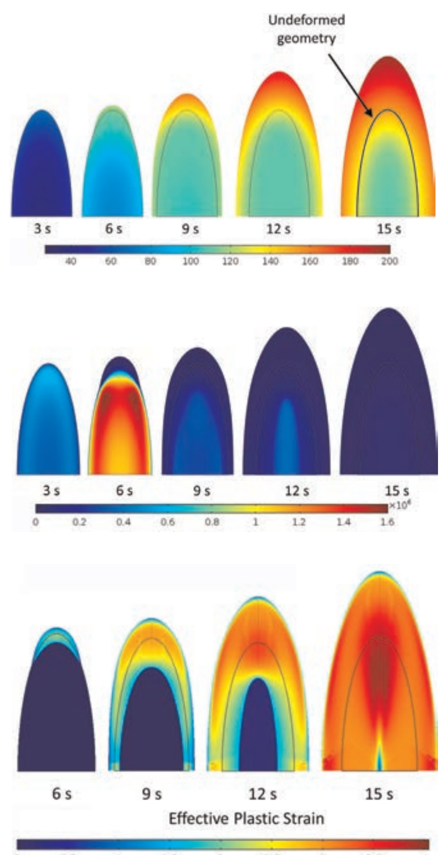


FIGURE 4. Temperature (top, shown in degrees Celsius); first principal tensile stress (center, shown in Pa); and effective plastic strain (bottom) during puffing.

rice grain expands, dries, and shrinks."

The model also provided an understanding of how the porosity developed, illustrating pore formation beginning at the kernel tip and progressing inward (Figure 5).

Based on the results, they determined the optimal amount of salt, moisture content, temperature, and heating time to produce the ideal puffed rice grain. The simulation also showed the conditions needed to maximize the expansion ratio.

⇨ LOOKING FORWARD IN FOOD ENGINEERING

In addition to this model framework, Prof. Datta's research team has extended their simulation practices to studies of food safety. This has big implications for food companies that need to predict the health benefits of certain foods, know when they will expire, and ensure that their processes are safe.

Prof. Datta is currently the PI on a USDA NIFA-funded project where his students are using COMSOL to not only build simulations, but also construct computational apps that extend the analyses to nonengineers. At Cornell University apps are deployed on a large scale via the COMSOL Server™. Apps are beneficial for students and teachers because they don't need to invest in the software or hardware directly.

"Simulation apps bring new opportunities to education," he remarks. "In a food safety class, apps enable multidisciplinary learning where students can simulate many what-if scenarios realistically." The app developed at Cornell is used by several universities around the US.

They have provided food scientists with an app that covers canned food, for analyzing how the necessary heating time for sterilization varies with different container sizes (Figure 6). The app user can adjust the temperature and calculate how long it will take to heat food to a safe temperature for a given can. It also provides an adjusted rate of bacteria dying, to confirm whether or not the final product will be safe for consumption.

Prof. Datta says that, while puffed rice was the starting point, their work is easily transferrable to other biomaterials such as corn — or even to completely different applications. "The physics and modeling knowledge is useful in other industries," he says. "For example, one of my students later studied microwave drying of molds for cars' catalytic convertors, using simulation techniques similar to those we developed here." While he teaches the next generation of engineers about the fundamentals of physics modeling, he looks forward to seeing what comes next for the food industry. ❖

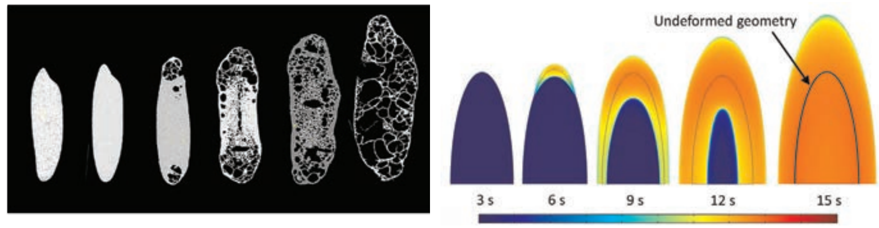


FIGURE 5. Left: CT scan of rice at different times during puffing. Right: Simulation showing predicted porosity profiles.

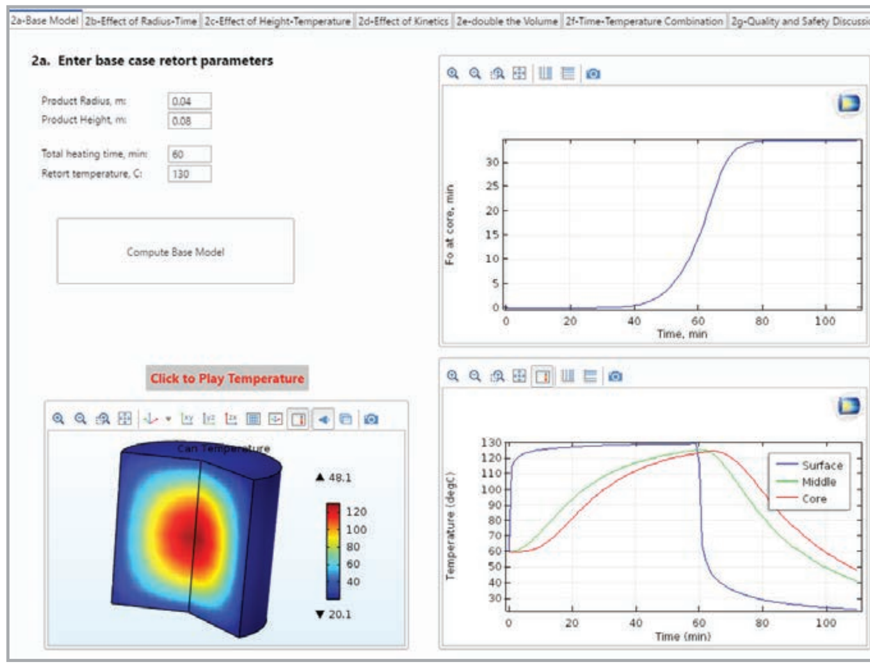


FIGURE 6. The computational app created by Prof. Datta's students for studying canned food. Users can change parameters such as the can dimensions and heating time.

One of Prof. Datta's favorite parts of his work is teaching a course at Cornell that introduces students to using COMSOL for biomedical process modeling. Because of the software's ability to fully couple integrated multiphysics phenomena, such as those present in rice puffing, he finds COMSOL to be a powerful tool for students to learn about simulation and the underlying physics in different applications of biomedical science.

This year Datta will also present at a workshop at The International School on Modeling and Simulation, a short-term international school that is the outcome of a special interest group for virtualization in food engineering.

REFERENCES

1. Gulati, Tushar and Datta, Ashim K. "Coupled multiphase transport, large deformation and phase transition during rice puffing," Chemical Engineering Science 139 (2016) 75–98.


Connect on social media:
#CTF2017
CAN YOU DESIGN
A BETTER
TOMORROW?
THEY DID.
Life-Saving Bacterial Detection
Grab-and-Go Booster Seat
High Performance Armor
Oil Change Transformation
NOW IT'S YOUR TIME TO SHINE!
ENTER TO WIN $20,000 GRAND PRIZE
createthefuturecontest.com
Create THE Future
DESIGN CONTEST 2017
PRINCIPAL SPONSORS
COMSOL
MOUSER ELECTRONICS
ANALOG DEVICES
intel

ACCELERATING CUSTOM CAPACITOR DESIGN WITH SIMULATION APPS

Engineers at Cornell Dubilier Electronics use simulation apps to evaluate and optimize custom capacitor designs. These apps allow design and manufacturing engineers to quickly explore configurations on-site, bypassing the complexity of the underlying computational model.

by SARAH FIELDS

Capacitors are ubiquitous across common electrical devices used today, as well as in applications where extreme conditions must be considered. In each of these applications for which capacitors are necessary, the requirements can vary greatly. A capacitor may require an exact power specification, may need to function within a certain temperature range, or be made of specific materials.

One of the biggest manufacturers of custom capacitors used around the world, Cornell Dubilier Electronics, develops capacitors for some of the most demanding military and aerospace applications, including fighter jets and radar systems, as well as civilian applications such as wind turbines and solar energy. Engineers at Cornell Dubilier use mathematical modeling and custom simulation apps to fine-tune the design of custom capacitors.

"By using COMSOL Multiphysics and its Application Builder I can create high-fidelity multiphysics models and build apps based on them, which allows my colleagues in other departments to test different configurations and pick the best design," comments Sam Parler, research director at Cornell Dubilier.

FIGURE 1. Aluminum electrolytic capacitors. The windings are composed of aluminum foils and cellulosic separators, and exhibit thermal anisotropy.

⇨ WHEN THINGS HEAT UP

Cornell Dubilier's capacitors are specific to the application for which they are designed and can comprise one or more elements, such as electrolytic windings composed of aluminum foils and cellulosic separators; electrostatic windings of offset, metallized dielectric films; or interleaved, stacked plates of metal foils and dielectrics such as mica (Figure 1).

One matter at the forefront of the issues considered by capacitor designers is heat. Passing current through the aluminum foils of the windings results in Joule heating, which must be taken into consideration during the design to gain a full understanding of the thermal profile within the capacitor. Too much heat dramatically shortens the capacitor lifetime, which is cut in half each time the capacitor's temperature is 6-10 degrees higher than the maximum. Engineers at Cornell Dubilier use simulation to minimize heat generation and to optimize dissipation of heat.

In optimizing heat generation and heat dissipation, the complex materials of the capacitor must be accurately represented. One capacitor can easily include as many as six materials, some of which have anisotropic properties. In one design, the winding is composed of cellulosic separators and aluminum foils, and exhibits anisotropy of thermal conductivity over two orders of magnitude higher in the axial than in the radial direction.

Parler is able to accurately capture the thermal profile of capacitors with COMSOL Multiphysics® thanks to the flexibility that allows him to directly input the thermal conductivity tensor. For example, a typical simple capacitor tensor of a z-oriented cylindrical electrolytic winding can be approximated as orthotropic with a diagonal thermal conductivity tensor of {1,1,100} [W/m/K].

In one case, Parler considered two power capacitors of similar size and ripple current rating, but with entirely different construction: that of a metallized polypropylene (plastic) film capacitor and an aluminum electrolytic capacitor

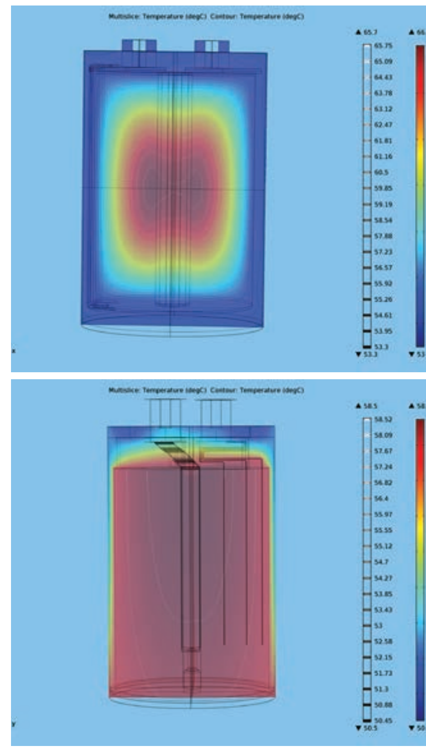


FIGURE 2. A thermal comparison of a metallized polypropylene film capacitor (top) and an aluminum electrolytic capacitor (bottom), both approximately 76x120 mm and dissipating 5 watts in a 45°C environment.

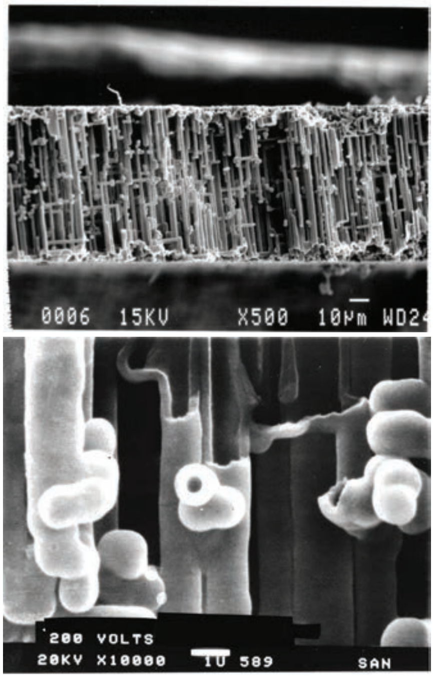


FIGURE 3. Coaxial microstructure of large aluminum electrolytic capacitors. The dielectric is aluminum oxide (Al_2O_3), grown in an anodizing bath on the tortuous surface of highly etched aluminum foil. In the images here, the aluminum surrounding the alumina dielectric tubes has dissolved away.

(Figures 1 and 2).

The plastic film capacitor (top) has a much lower axial thermal conductivity than the aluminum electrolytic capacitor (bottom). Using multiphysics simulation, Parler was able to quantify how much hotter the plastic film capacitor becomes compared to the aluminum electrolytic capacitor for a given dissipated wattage.

⇨ DEMYSTIFYING THE MICROSTRUCTURE WITH SHAPE OPTIMIZATION

As the capacitors developed at Cornell Dubilier are often new technological developments, in some cases, it is necessary to characterize the impedance of cutting-edge materials in house. In designing one large aluminum electrolytic capacitor, Parler needed to represent the impedance of an aluminum oxide (Al_2O_3) dielectric with a complex microstructure. This dielectric was produced in an anodizing bath on the tortuous surface of highly etched aluminum foil (Figure 3).

While a zero-dimensional electrical circuit simulation carried out in a different software could reproduce the frequency response, it was not able to perform the transient simulation due to 'noncausality' errors arising from the limitation of its internal inverse-Laplace-transform algorithms.

Using a shape optimization technique with the COMSOL software, Parler was able to calculate the correct transient solution for a customer. He began with a single cylindrical, electrolyte-filled capacitive pore, applied a known excitation at the opening, and used the sparse nonlinear optimizer solver (SNOPT) available in the software to find the solution to his nonlinear optimization problem where the shape of the axisymmetric pore wall needed to be varied until the experimental impedance data was fitted.

The resulting geometry (Figure 4) demonstrated that the software could accurately reproduce the time-dependent pulse-current response that was measured experimentally, allowing further design work based on a validated mathematical model.

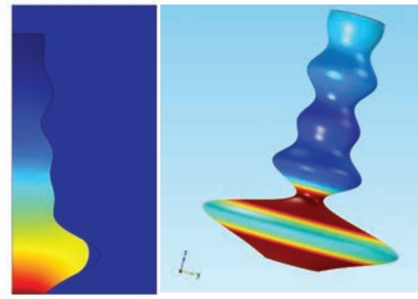


FIGURE 4. An alternative approach to capturing the electrical behavior of the coaxial microstructure of the dielectric material is to use shape-optimization techniques. Optimized microstructure is shown.

> "Using COMSOL Multiphysics® and its Application Builder I can create high fidelity multiphysics models and build apps based on them, which allows my colleagues in other departments to test different configurations and pick the best design."
>
> — SAM PARLER, RESEARCH DIRECTOR, CORNELL DUBILIER

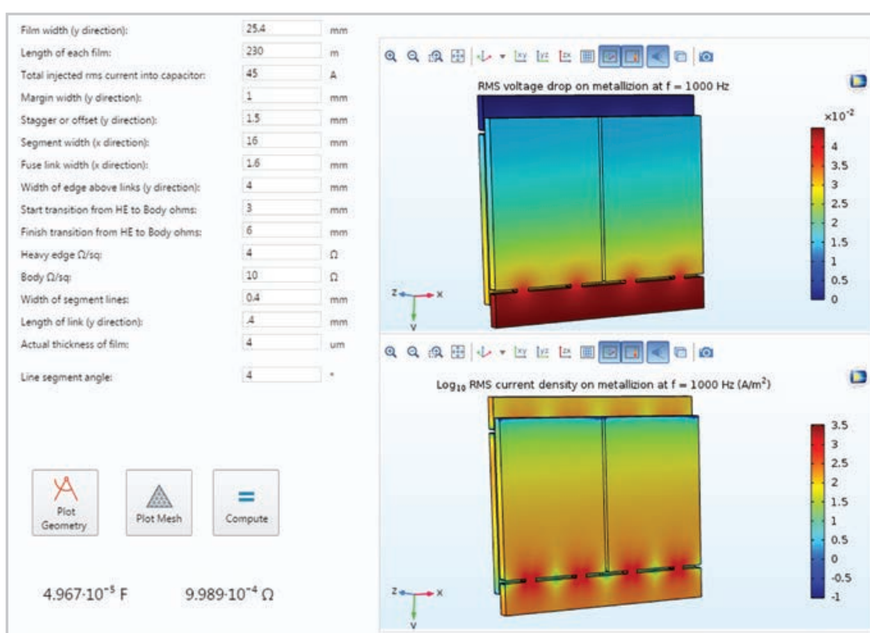


FIGURE 5. A design app for a power film capacitor used to determine capacitance and resistance.

➾ A FAMILY OF APPS FOR ELECTRICAL OPTIMIZATION

After using COMSOL to create models to analyze their designs, Parler and his team convert the models into simulation apps that are ready to be deployed to design engineers and manufacturing sites to assist in the design process.

Using one simulation app for a power film capacitor, a design engineer can enter the film width (typically a few centimeters), film length, surface resistances, and transition region location into the interface to determine the capacitance and resistance of a segment of the metal film (Figure 5). The result is scaled to yield a reported capacitance and resistance for the entire winding, providing engineers with an initial validation of their design.

Another app calculates power density for the metal film in a cylindrical capacitor. It also predicts the core temperature distribution, including throughout the tabs and terminals, taking into account the customer's operating conditions, such as ripple current, ambient temperature, and air velocity (Figure 6).

A third app is used to calculate the effective series inductance (ESL) of a single-tab film capacitor (Figure 7). Geometric parameters such as the terminal diameter, terminal height, terminal spacing, tab width, winding diameter, and the outside diameter of the core can all be modified by the app user. The underlying model utilizes a frequency-domain study and the electromagnetic modeling capabilities of COMSOL. The ESL is a key aspect of the design of any capacitor, and is directly linked to capacitor performance.

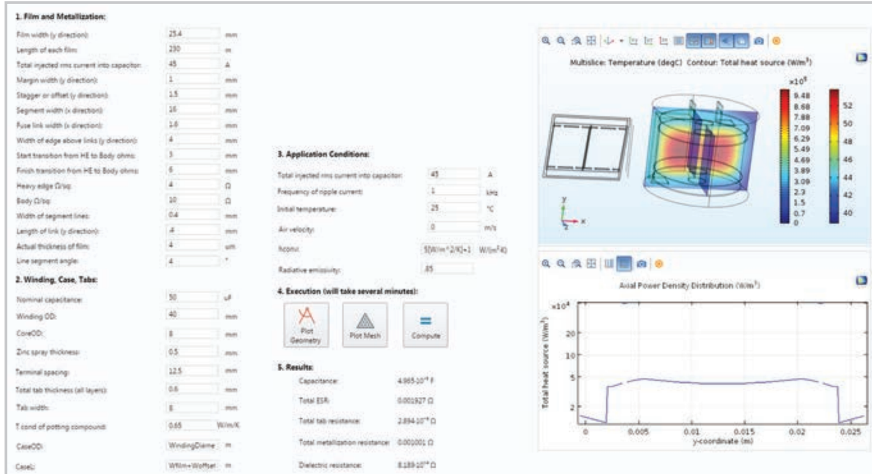


FIGURE 6. A simulation app that predicts the core temperature distribution and the power density for the film of a cylindrical capacitor with tabs and terminals.

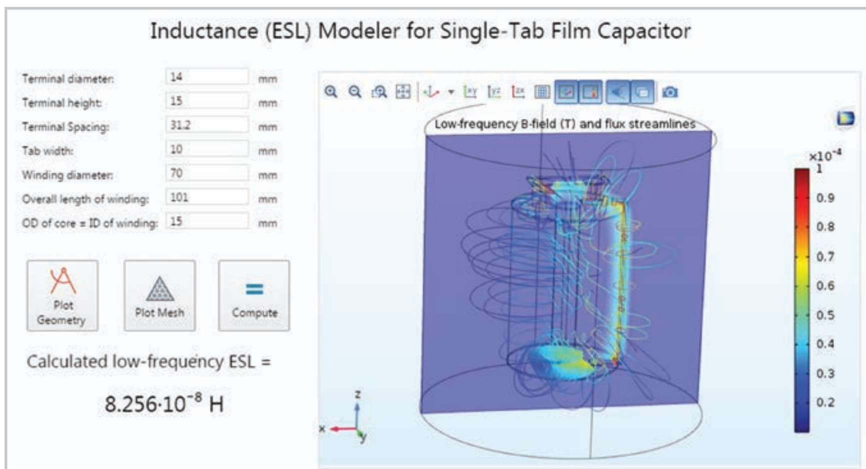


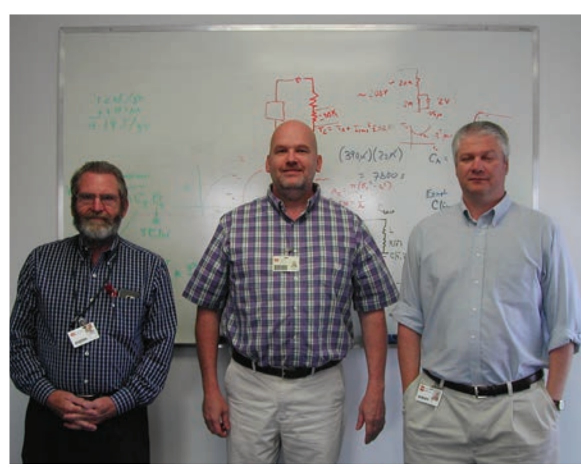
FIGURE 7. An app used to calculate effective series inductance (ESL) of a single-tab film capacitor.

➾ APPS GUIDE THE FUTURE OF MANUFACTURING

With multiphysics simulation, Parler's team is able to accurately predict the performance of their capacitor designs, speeding development and ensuring reliability of their products.

Simulation apps based on the underlying COMSOL models allowed other members of the design team and engineers at a manufacturing site to adjust key parameters of the simulation through a simplified user interface to test how their capacitors will perform and show the effect of design adjustments. This spreads the power of simulation throughout the design and manufacturing process.

Parler concludes, "The ability to build multiphysics models and simulation apps has streamlined our capacitor design process and sped the development of customized capacitors for customers all over the world." ❖



Left to Right: David Leigh, staff scientist; Sam Parler, research director; Trent Bates, capacitor engineer; Cornell Dubilier.

BEHIND THE RUMBLE AND ROAR OF MAHINDRA MOTORCYCLES

Mahindra Two Wheelers used multiphysics simulation to meet engine noise regulatory requirements in its high-end luxury motorcycles while maintaining customers' satisfaction.

by **VALERIO MARRA**

Mahindra Two Wheelers builds a wide range of scooters and motorcycles for the Indian market. Thanks to the adoption of numerical simulation tools early in the development cycle, drivers and passengers can enjoy great performance and mileage, along with a superior ride experience on tough Indian roads. Mahindra used multiphysics simulation to study the NVH (noise, vibration, and harshness) performance of the engine, intake, and exhaust systems of their motorcycles.

The knowledge gained from numerical simulation studies enabled their engineers to improve the structural design of their motorcycle engine and achieve desired noise levels. "COMSOL software helped us to significantly reduce the number of design iterations that we had to go through, thereby saving time," said Niket Bhatia, deputy manager R&D, Mahindra.

⇨ ACHIEVING OPTIMAL NOISE LEVELS

In an engine, there are many sources of noise, including the intake and combustion processes, pistons, gears, valve train, and exhaust systems. Combustion noise is due to structural vibrations caused by a rapid pressure rise within the cylinders. These vibrations continue from the powertrain to the engine casings through bearings, radiating noise.

Acoustics analysis solely through physical testing can be an expensive and time-consuming process. The team at Mahindra decided to complement physical testing with acoustics modeling to analyze how the engine's structure might encourage noise radiation. The research goal was to find the parts of the engine that generate the most noise and come up with changes to the structure that could reduce it.

Using the COMSOL Multiphysics® software, the researchers performed an acoustic-radiation analysis of a single-cylinder internal combustion (IC) engine under combustion load. The engineers enclosed the engine skin in a computational domain surrounded by a perfectly matched layer (PML).

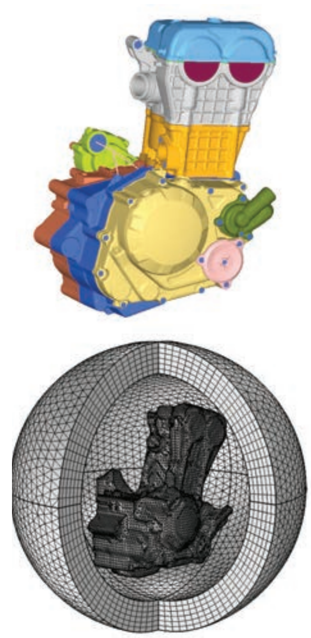


FIGURE 1. Top: engine CAD geometry. Bottom: meshed 3D model enclosed in a perfectly matched layer (PML).

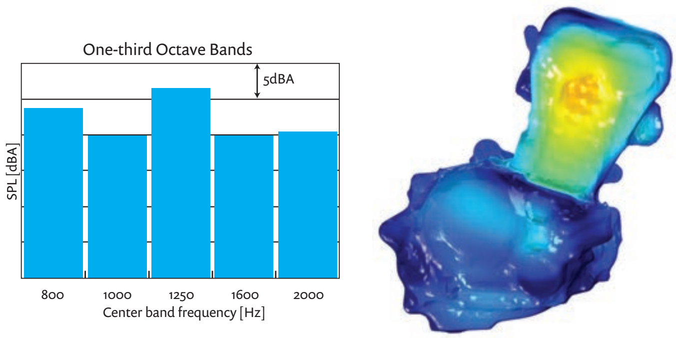


FIGURE 2. Left: One-third octave band plot. Right: 3D surface plot of the sound pressure level (SPL) simulation results.

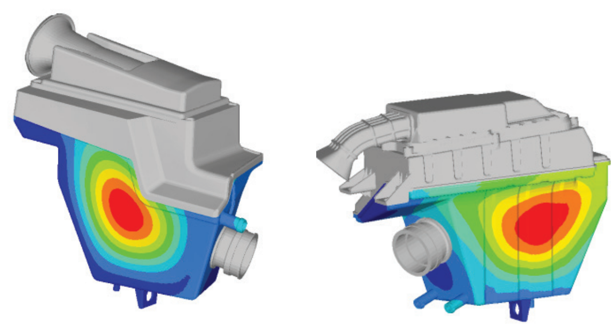


FIGURE 3. Air filter structure. Left: Original design. Right: Modified design, featuring ribs to improve the ATF.

PML's dampens the outgoing waves with little or no reflections (Figure 1). This allows for accurate results while reducing the size of the computational domain.

The team decided to focus their analysis in the 800 Hz -2000 Hz frequency range, as physical experiments indicated that the motorcycle's engine noise radiation under combustion load was dominant in that region of the acoustic spectrum. This choice allowed the team to save computational resources and better understand what areas radiate the most noise.

Based on this analysis, the sound pressure level (SPL) was studied and modifications, such as increasing rib height and wall thickness and strengthening the mounting location, were made to the cylinder head and block (Figure 2). By adjusting these parameters, reduction in SPL was achieved at the targeted frequency range.

⇨ REDUCING INTAKE STRUCTURAL NOISE

Both intake and exhaust noise are major contributors to pass-by-noise. Noise radiating from the air filter structure, usually made of plastic, is one of the major contributors to intake noise. An acoustic transfer function (ATF) analysis was carried out for the plastic air filter walls. The air filter structure was modified by providing ribs to improve the ATF (Figure 3). This helped in reducing the structural noise of the air filter (Figure 4).

⇨ ANALYZING TRANSMISSION LOSS TO IMPROVE MUFFLER SOUND

Regulatory requirements are always competing with customer demands for louder ‘rumbling’ from the muffler, as it is perceived as an important indicator of the motorcycle’s power. Within the constraint of pass-by-noise, the challenge for Mahindra engineers was to increase the ‘rumble’ sound from their muffler at low frequencies while reducing the sound level for higher frequencies.

While attenuation of engine exhaust noise is the primary function of the muffler, factors such as the ability to provide low back pressure and meet pass-by-noise regulations also need to be considered. The performance of a muffler in an automotive exhaust system is characterized by three parameters: transmission loss, insertion loss, and radiated noise levels. Transmission loss is considered the most important parameter, and it is determined solely by the muffler design and is independent of the pressure source. The challenge for the team at Mahindra was to predict the transmission loss for a motorcycle muffler and then optimize the loss to desired levels for a certain frequency range.

A muffler of a single cylinder motorcycle engine was considered for the analysis. Transmission loss analysis of the muffler was carried out using COMSOL Multiphysics. With the Acoustics Module, boundary conditions such as continuity and sound hard wall were applied at appropriate locations.

Perforations in pipes were defined by giving porosity details for the perforated area using a built-in transfer impedance model. The inputs required for analysis were the area porosity, baffle and pipe thickness, and diameter of holes. For porous materials such as glass wool, flow resistivity was defined with a poroacoustic model available in the software. Unit pressure was given as input at the inlet and a plane wave radiation condition was applied to both inlet and outlet boundaries.

Based on the results, the muffler design was modified by increasing the pipe length inside the muffler. With the modified muffler, the team achieved reduced transmission loss at low frequencies (Figure 5). As a result, the desired outcome of increased noise levels at

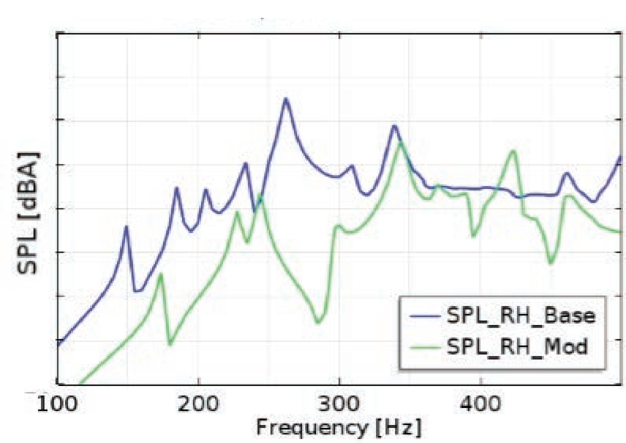


FIGURE 4. Simulation results show a reduction in the structural noise for the modified air filter design.

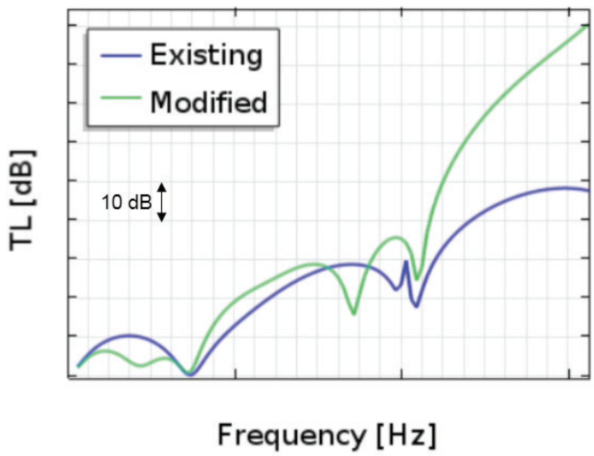


FIGURE 5. Transmission loss (TL) comparison between different designs. The modified design is characterized by reduced transmission loss at low frequencies and increased transmission loss at high frequencies. The modified design achieved the sought after 'rumbling' noise while meeting regulations.

low frequencies, or the 'rumbling' noise, was achieved.

⇨ OPTIMIZATION EARLY IN THE DESIGN CYCLE LEADS TO COST AND TIME SAVINGS

"I personally really liked the software's flexibility and available tools like the COMSOL API," said Ulhas Mohite, manager of R&D, Mahindra. "It allowed us to carry out process automation using Java code which, while dealing with acoustic analysis for example, enabled us to use different meshes for different frequency steps to find the right compromise between simulation accuracy and computational time. It also enabled us to automatically export desired outputs such as surface SPL plots and far-field SPL data in the middle of the simulation run. This helped save a lot of time with respect to manual postprocessing and exporting the data."

Mohite also found the Application Builder tool available in COMSOL extremely useful. "We created a simulation app (Figure 6) using the Application Builder to compare analysis output files and plot the SPL data, which was a great time saver."

> "We created a simulation app using the Application Builder to compare analysis output files and plot the SPL data, which was a great time saver.
>
> — ULHAS MOHITE, R&D MANAGER, MAHINDRA

Analysis results proved to be very closely correlated with physical experiment data. With simulation, the engineers at Mahindra were able to take corrective actions by carrying out structural modifications based on analysis results early in the design stage. This helped reduce both time and cost involved in product development. "When supported with experiments, these simulations lead us in the right direction to find an efficient solution to motorcycle noise issues," concluded Bhatia. ❖

FIGURE 6. With the Application Builder, Mahindra engineers created an easy-to-use simulation app that is used to compare analysis files and plot sound pressure level (SPL) data.

REFERENCES

1. Mohite, U., Bhatia, N., and Bhavsar, P., "An Approach for Prediction of Motorcycle Engine Noise under Combustion Load," SAE Technical Paper 2015-01-2244, 2015, doi:10.4271/2015-01-2244. (http://papers.sae.org/2015-01-2244/)

2. Reducing Motorcycle Engine Noise with Acoustics Modeling, COMSOL Blog, https://www.comsol.com/blogs/reducing-motorcycle-engine-noise-with-acoustics-modeling/

Optimizing Screen Protection During the Manufacturing Process

Kornerstone Materials Technology (KMTC) optimizes glass manufacturing processes by using multiphysics simulation technology, fostering collaboration across company departments with simulation apps.

by VALERIO MARRA & LEXI CARVER

Nowadays, smartphones and tablets have become inseparable parts of our daily work, learning, leisure, and entertainment. The screen of a mobile device has a protective top layer that is known as the cover glass. The cover glass is mainly used for screen protection, offering highly visible light transmittance and an elegant appearance.

Cover glass is a high-end product, with prices tens or even hundreds of times higher than that of traditional soda-lime glass. The high price is due to the extremely challenging manufacturing process. Things get more complicated due to the different behaviors cover glass exhibits under different conditions. These particularities challenge a company's product and development department to perform at its highest caliber.

Kornerstone Materials Technology Co., Ltd. (KMTC) specializes in displays, touch component products, and high-tech material development and production (Figure 1). "Our products focus on cover glass used in electronic devices such as smartphones and tablets," explains Frank Hong, R&D manager at KMTC. "To meet the growing diversified needs of our customers and to cope with the opportunity and challenge of the touch display industry, KMTC is taking advantage of multiphysics simulation to evaluate and optimize the glass manufacturing process."

⇨ MULTIPHYSICS IN GLASS MANUFACTURING PROCESS

The display glass production industry is dominated by three major manufacturing processes: float glass, slot down-draw, and overflow down-draw. Within these three, the overflow down-draw process is the most widely used technique. In this process, an ultrathin glass sheet forms naturally in air as two overflowing molten glass regions join together (Figure 2, left). The glass produced has flat surfaces and dust-free cutting areas. It does not require grinding or polishing to fix characteristic differences on the glass surface.

As a pioneering company producing high-alumina cover glass, KMTC has enhanced the application of overflow technology to the domestic leading level. "Our unique process can produce the glass without any mark or damage on the surface during the forming process," says Hong. "The surface of the formed glass substrate is smooth, pure, and flawless, meeting the demands of the consumer electronics market."

During the overflow down-draw process, highly viscous and homogenized molten glass flows from a melting furnace through a platinum channel and then to an overflow forming block made from refractory materials (Figure 2, left). The molten glass flows down either side of the block under the influence of gravity (Figure 2, right). The glass regions rejoin at the bottom of the block and continue downward, guided by a plate, and cools in the air to form an ultrathin glass sheet.

The sheet thickness is controlled by the inlet flow and the guiding plates, which also determine glass output. Temperature is monitored closely since it affects glass viscosity and flow velocity, and must be carefully controlled to avoid warping. The complete glass manufacturing process is a multiphysics problem involving fluid-solid-thermal-electric coupling.

Engineers at KMTC use multiphysics models to evaluate the electric heating efficiency of the glass melting systems. They also use apps built from their models to simulate the real-time manufacturing processes. The resulting data is used as guidelines for manufacturing processes adopted by the production department.

Their models, built in the COMSOL Multiphysics® software, include coupled fluid, structural, thermal,

FIGURE 1. 3D cover glass allows designers to deliver better displays for smartphones and tablets.

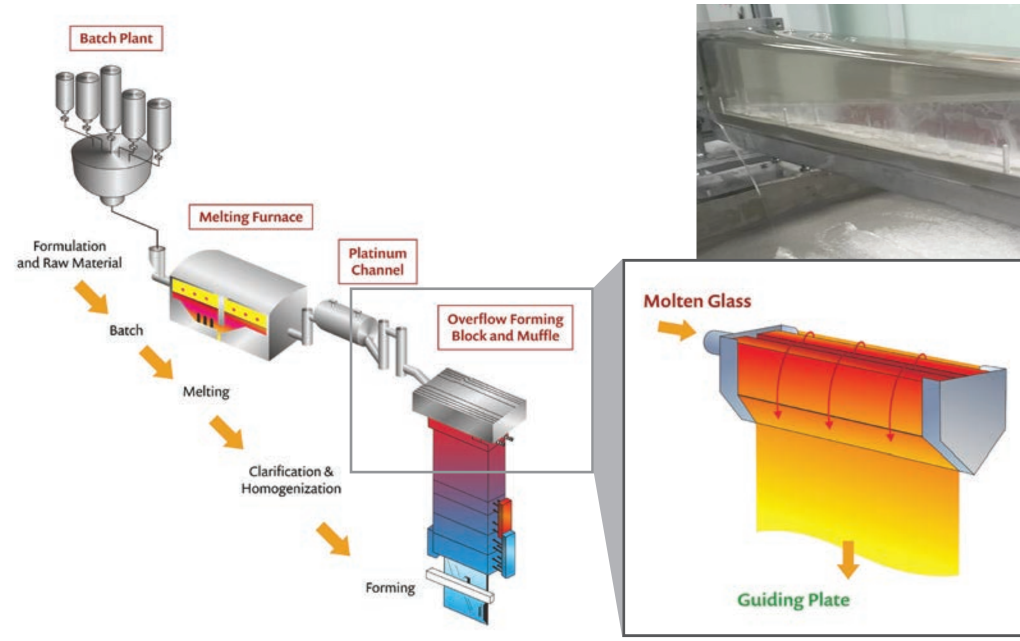


FIGURE 2. Overflow down-draw process: schematic (left), detailed view of the forming block (right), and photo of the molten glass overflow.

and electrical phenomena. The team performs fluid and heat transfer simulations of molten glass to calculate the thickness distribution on the outflow glass surface before cooling, and the strains generated during the forming process (Figure 3). "The software makes it possible for us to input our own constitutive equations or vary parameters such as inlet velocity or tilt angle of the forming block, and to optimize process conditions before mass production," explains Hong.

Multiphysics simulation results are used to predict the glass sheet quality, based on factors such as the thickness, uniformity, smoothness, and defects of the final sheet, as well as to optimize the device and process conditions. "COMSOL allows us to solve multiphysics problems with the needed level of customization," says Hong.

⇨ SIMULATING THE ELECTRIC HEATING SYSTEM

When the molten glass contacts the refractory materials of the forming block, its composition changes to include air, which can introduce bubbles and change the weight of the final glass sheet. The presence of the platinum channel compensates for this by allowing preconditioning such as refining, homogenization, stirring, and temperature adjustment before the glass meets the forming block. Each section of the channel can deliver molten glass with different viscosities thanks to real-time temperature control by an electric heating tube using controlled AC (alternating current).

There are two types of AC heating tube structures in the channel — a straight tube with nonuniform wall thickness and a Z-tube with uniform wall thickness (Figure 4). "Using COMSOL Multiphysics we are able to evaluate the current density distribution on tube walls and the heating effects of both tube structures," says Hong.

The multiphysics model includes Joule heating by coupling electric currents and heat transfer. The governing equations are discretized with the finite element method and solved using a frequency-stationary study. The computed results include the AC heating effects and surface current density distributions, as shown in Figure 4. The results showed Hong's team how the heating effects differ between the two tubes. Experimental validation demonstrated that the simulation results were in good agreement with real-world measurements.

⇨ SIMULATION APPS ENHANCE COLLABORATION

"To build such specialized models, simulation expertise and knowledge of the system being simulated is required," explains Hong. The most efficient way to share this knowledge with others in the company is to take advantage of the Application Builder tool available in the software. Simulation specialists can customize their models so that only

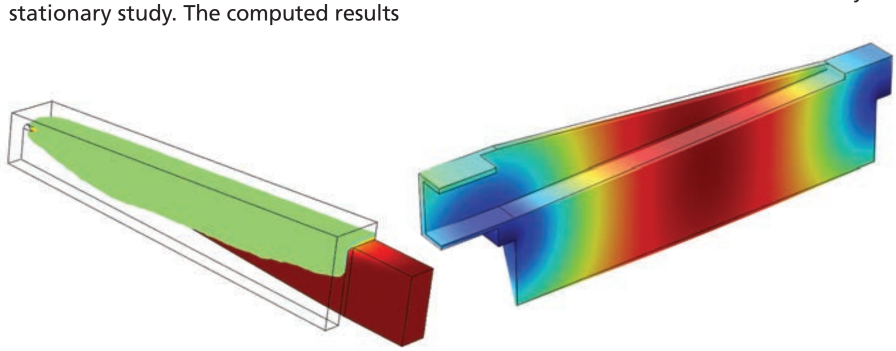


FIGURE 3. Simulation of the molten glass overflow process in COMSOL software. These results show the flow region of molten glass (left, in green) and strain in the forming block (right).

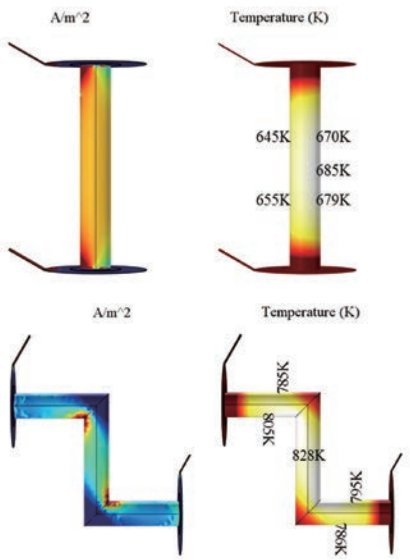


FIGURE 4. A simulation for predicting the state of the glass within a heating tube. Top: Straight tube with nonuniform wall thickness. Bottom: Z-tube with uniform wall thickness. Left to right: simulation results depicting current density distribution and temperature distribution.

chosen parameters will be accessible to the user. The individualized interface is based on the model, which can be distributed internally and allow colleagues to run complex analyses independently.

The simulation team reports to the R&D department of KMTC, but simulation apps extend their work to the engineering department as well. The next challenge is to make simulation accessible to customers directly. Then they can make detailed adjustments according to their design needs, without needing to be experts in the underlying multiphysics model.

The KMTC simulation team built an app to study the platinum channel. The app can be used for calculating fluid temperature and velocity, as well as stress distribution in the heating tube during manufacturing to predict the stress levels and final state of the glass (Figure 5). Input parameters available to the app user include glass temperature, velocity at the inlet, and heating power. A stationary analysis evaluates and optimizes the operating conditions, while a time-dependent analysis simulates the real-time manufacturing processes. The app provides the necessary data to define site instructions for production.

"It is very convenient for engineers to modify input parameters through the interface of the application to, for example, predict the state of the glass within the heating tube," says Hong. "This has simplified the designer's work, while improving the team's efficiency."

The overall simulation workflow has been streamlined. Simulation specialists work on a parameterized mathematical model with various design parameters added for later use, then turn the model into a simulation app, to deploy to other engineers. Designers who are not familiar with multiphysics simulation are empowered to solve practical problems with flexibility and efficiency.

The team deploys simulation apps to colleagues with the COMSOL Server™ product installed on a computing cluster, to support requests from the R&D and engineering departments. Users are able to run the apps based on customer demands and immediately provide insights into the manufacturing process. "Simulation apps are a great tool for the long-term development of the CAE simulation team as they take both cost and information security into consideration," says Hong.

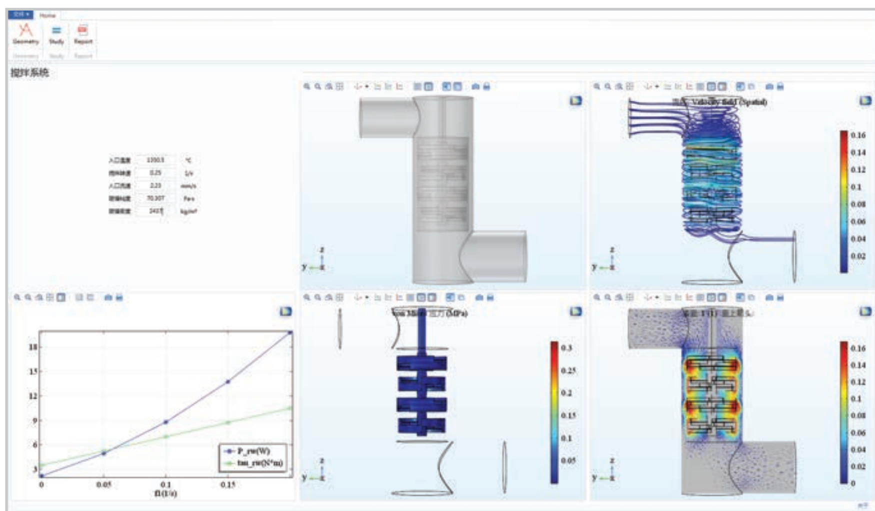


FIGURE 5. App used to predict the state of the glass within the heating tube. Users can change parameters such as glass temperature, velocity at the inlet, and heating power at different sections to investigate fluid temperature, velocity, and stress in the heating tube.

From left to right: Zhenlong Yang; Jiming Yang; Liyao Tao; Lixin (Frank) Hong: research and development manager at KMTC.

BOOST SALES AND BUILD STRONGER RELATIONSHIPS WITH THE DEPLOYMENT OF SIMULATION APPS

Multiphysics simulation experts at GrafTech are enhancing the sales process by deploying multiphysics apps throughout the organization and beyond.

by NATALIA SWITALA

While salespeople do not often think of simulation specialists as a resource to their sales cycle, a lot can be accomplished when we bring people from different departments together. Rick Beyerle is a senior scientist in the Innovation and Technology group at the Advanced Energy Technologies (AET) subsidiary of GrafTech. He has been collaborating with the sales teams to introduce simulation apps in the sales process of their carbon and graphite products.

⇨ BUILDING TRUST TO WIN CUSTOMERS

Rick and his associates use multiphysics simulation to examine the electrical, structural, and thermal performance of carbon and graphite, as well as for design and process optimization for several industrial applications. In collaborating with the sales team, it was evident that building trust with prospective customers was one of the most important factors in the sales cycle. “Oftentimes, providing sales leads with a ‘proof of concept’ serves as the building blocks for establishing trust between the two parties,” explains Rick.

Before the availability of customized simulation apps, this ‘proof of concept’ required Rick and his team to divert their R&D resources to modify and rerun validated models for each customer’s specific configuration. The sales team was not trained in numerical modeling and the application engineers had been instructed to prioritize live tests over time-consuming simulations. “Some models feature hundreds of parameters and boundary conditions that do not appear significant to an untrained eye, but significantly impact the simulation results,” says Rick.

⇨ SIMULATION APPS PROVIDE A ROADMAP FOR COLLABORATION

Rick is a visionary who is committed to using simulation in a new way that benefits the entire organization. With the Application Builder available in the COMSOL Multiphysics® software, Rick and a team of application engineers built a user-friendly interface based on their standard multiphysics model. The result was 'SpreaderCalc', an app that allows the sales engineers and field specialists to predict the performance of a range of virtual prototypes before testing a costly physical prototype (see Figure 1).

Achieving great technological and sales results is a company-wide effort. This is why Rick wanted to provide his colleagues in the sales departments with the software tools to collaborate and offer prospective customers real-time answers. “Frequently, a sales lead asks us to recommend thermal management options, not only to meet the safety or reliability requirement but more to maximize their user’s experience by removing limitations due to temperatures, especially when under specific space and geometry constraints” commented Pierre Hatte, sales director. “The new tool that our colleagues in the Innovation and Technology group developed is helping my team to get a second meeting with a large sales opportunity. And if you work in sales, you know that securing that second meeting is where the real opportunities are uncovered.”

⇨ PUTTING APPS IN THE HANDS OF CUSTOMERS

Once simulation apps are ready for salespeople, they can be deployed with COMSOL Server™ product, which hosts them centrally and makes them accessible through a secure web connection. “With simulation apps, you do not have to be an expert in order to access high-fidelity multiphysics simulation results,” Rick comments. “With the adoption of simulation apps the workflow is more streamlined and inclusive.”

> **“Showing a prospective customer how the app compares heat transfer among different configurations is like letting them try the suit on before they buy it. They are confident that the results are tailored to fit their needs."**
>
> — PIERRE HATTE, SALES DIRECTOR, GRAFTECH

Once the sales force is fully equipped with apps and has tested the various scenarios that come up during the sales cycle, the company puts apps into customers’ hands with the use of the COMSOL Server. “Until now, one customer-oriented model per year was provided; the policy was that simulations were too expensive to provide for most customers,” he continues. “Now, they cost one hour of an application engineer’s time to run a DoE (design of experiments) of models, instead of running a battery of tests in a week.” “The apps provide our customers with

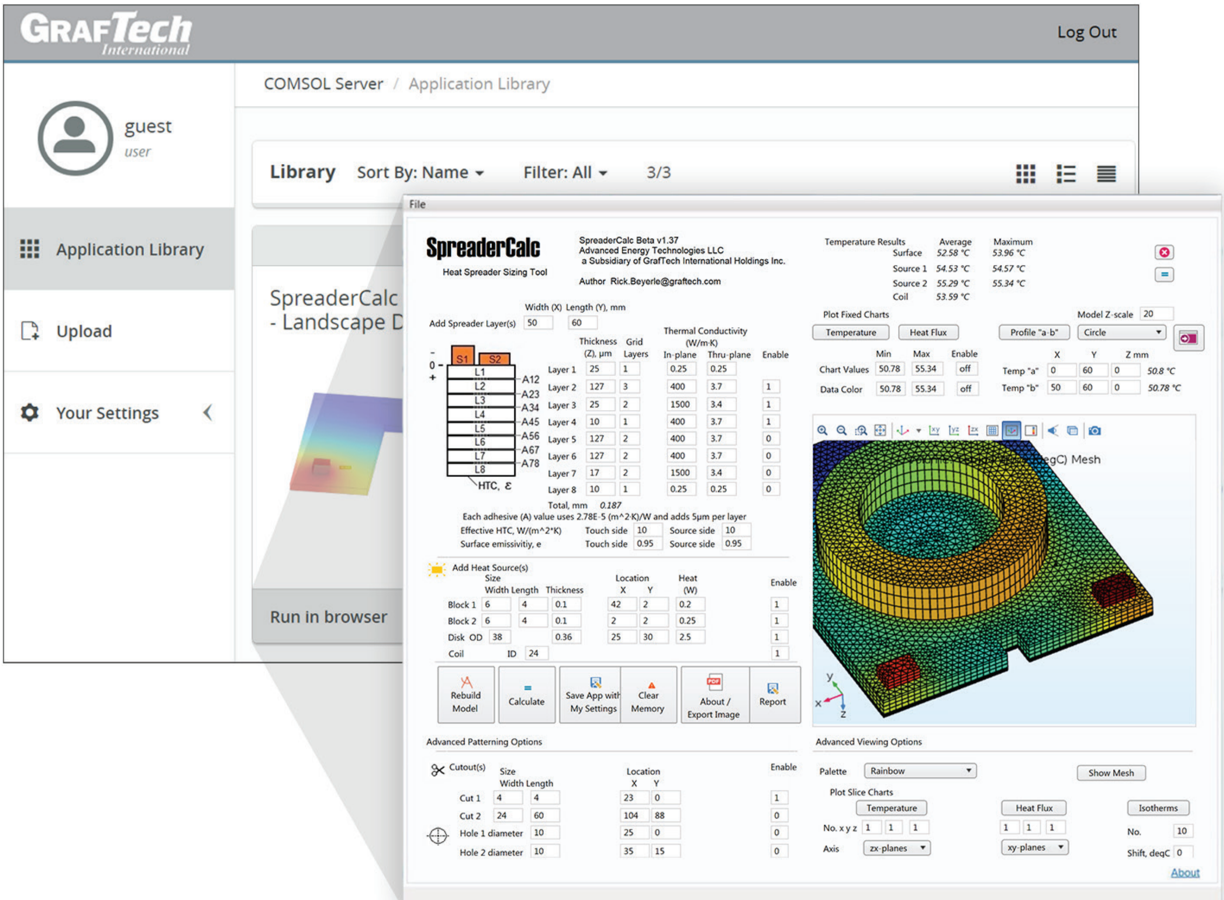


FIGURE 1. Using a local installation of the COMSOL Server™ product, GrafTech AET gives colleagues worldwide access to apps over the company's intranet. The SpreaderCalc app compares heat transfer among the graphite foils that dissipate heat in consumer electronics.

a sense of trust behind our products," says Pierre. "They are intuitive and easy to use, yet powerful because they are based on a multiphysics model built by our specialists. Showing a prospective customer how the app compares heat transfer among different configurations is like letting them try the suit on before they buy it. They are confident that the results are tailored to fit their needs."

"We envision that apps will empower customers to make more informed decisions prior to placing orders. Some purchasing departments are compelled to shave costs and occasionally disregard the engineering specifications. Without technical assistance, they are stressed to justify a high performance material when the only metric is price," said Rick.

"The simulation of graphite is challenging because of its highly orthotropic ratios. Even a 'bad' analysis was impossible. Now, the engineers can get a 'good' estimate without leaving their desks. To me, this is the key – the COMSOL software enables something specific that could not otherwise be accomplished."

Simulation apps also foster collaboration and transparency. For example, apps will allow customers to more effortlessly present their buying choices to their organizations. With COMSOL Server, all a customer needs is login information to run the app and download its results. This way everybody will be more comfortable in approving a purchase order. Based on the success of SpreaderCalc, and expanding upon its infrastructure of software, GrafTech AET is already creating variants to assist with niche markets such as thermal interface materials, EMI/RFI shielding, and rapid heating processes. ❖

LEARN MORE

If you are interested in knowing more about industrial applications of carbon and graphite for thermal management, please see the article published on page 3 of Multiphysics Simulation 2016 [www.comsol.com/offers/mphsim16].

Left: Pierre Hatte, sales director, GrafTech. Right: Rick Beyerle, senior scientist, GrafTech.

ECHOLOGICS, CANADA

MULTIPHYSICS ANALYSIS ADVANCES WATER MAIN LEAK DETECTION

Predicting the speed of sound is important for accurately locating leaks in buried pipes such as water mains. Echologics Engineering has implemented a finite element simulation framework to model acoustic behavior in pipes and estimate variations in the speed of sound.

by VALERIO MARRA

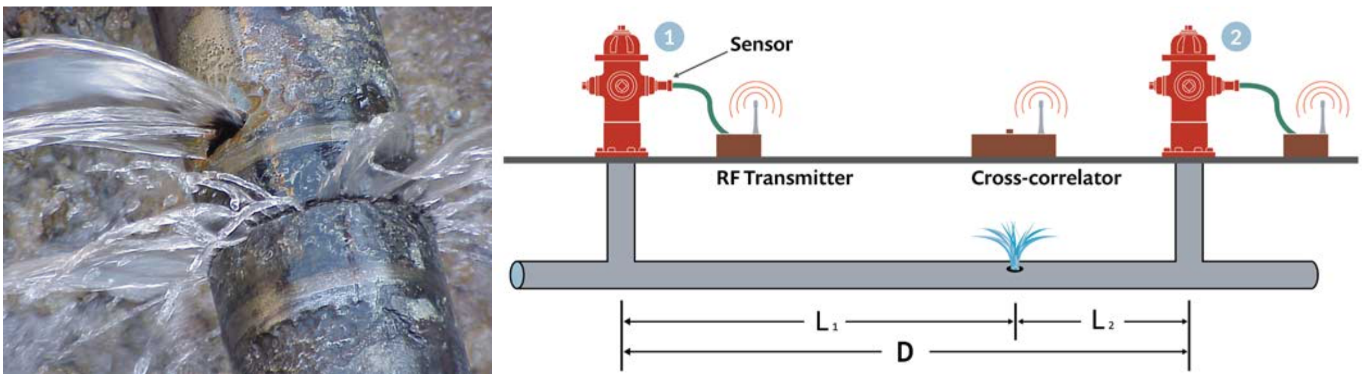


FIGURE 1. Left: Leaky pipe under investigation. Right: Schematic of leak detection setup. A leak is bracketed by two sensors whose distance is D. The leak sound propagates in both directions and a correlator measures the time it takes to reach each sensor. Based on the speed of sound in the pipes, the exact leak location can be found.

Fresh, clean water is a precious commodity that municipalities cannot afford to waste in underground pipe leaks. As pipe infrastructure ages, finding leaks becomes more difficult. As water grows in value, finding leaks becomes more critical.

That is where the Toronto-based company Echologics, a division of Mueller Canada, Ltd., with its unique acoustic technology for noninvasive leak detection, enters the picture. "Leaks make noise," explained Sebastien Perrier, R&D acoustical scientist at Echologics. Perrier is a mechanical engineer who specializes in acoustics and vibrations, the coupling of structures, as well as signal processing. "The pipes talk and, if you listen, they'll tell you where leaks are located," he said.

Echologics measures the time-of-flight of the sound using a correlation function and acoustic sensors set on the pipes or fire hydrants. If a leak occurs somewhere between two sensors, the leak is detected and the correlation result is used to determine the time difference the leak noise takes to reach each sensor. This provides the distance from the leak to each sensor once the speed of sound is known in the pipes under investigation (Figure 1).

A leading innovator of acoustic systems for water infrastructure, Echologics designs technologies that exploit this correlation to find leaks and to continuously monitor pipes for leaks. Examples of Echologics products include the LeakFinderST™ leak noise correlator (Figure 2) and the EchoShore®-DX pipeline monitoring system (Figure 3). Echologics correlators allow field specialists to investigate leaks in a variety of pipes using transmitters, sensors, and a user interface that can be set up on a standard laptop. This acoustic technology can detect even very small leaks in the early stages of formation, saving municipalities' money and pipe damage since they monitor leaks as they grow and are able to take action quickly.

The technology powering Echologics' products requires a precise understanding of the speed of sound in different types of pipes. It is material dependent, proportional to the stiffness of the pipe, and influenced by the pipe geometry. "The key was developing technology sensitive enough to make leak detection possible in PVC pipes," explained Perrier. Plastic has high attenuation and dampening compared with metal. Even trickier is the fact that older water systems originally made with cast iron pipes are being repaired — in individual segments — with plastic.

Keeping the sophisticated acoustic correlation algorithms up to date and accurate is one of Perrier's

FIGURE 2. The LeakFinderST™ correlator is a compact, intuitive leak noise correlator.

FIGURE 3. The EchoShore®-DX System turns existing fire hydrants into smart leak detection technology.

responsibilities. He must understand the physics involved at a fundamental level in order to optimize and develop next-generation solutions for buried pipe infrastructure. To help him speed up the design process and share his findings with other departments, Perrier creates computational acoustic models and builds simulation apps based on them.

⇨ CATCHING LEAKS BEFORE THEY CAUSE FAILURES

How does numerical simulation help predict acoustic wave propagation in pipes? The pipe network analysis can be complex and time consuming. One may want to understand the sound propagation and vibration response from a single pipe perspective or from an entire network. Therefore, the complexity of the model and the time it takes to run the analysis can change considerably depending on the level of details needed for the physics involved in the model to be accurate.

Making sure that the sound propagation speed is accurate for each pipe segment is at the heart of the problem that Perrier solved at the early stage of the design process. He then adopted multiphysics simulation to give him faster access to the values relevant to his work. In a pipe networks analysis, multiphysics couplings between acoustics, flow, and structural mechanics are needed.

In Perrier's work, there are multiple uses for simulation. Such as being able to understand slight margins of error and fine-tune the technology. Exploring material and geometry parameters for a pipe network through acoustics simulation reveals predictions for different scenarios. The acoustics simulation exhibits the presence of signal noise when the sensors' distance varies, or indicates that a plastic repair must have occurred that wasn't included in the test. Perrier's simulation also predicts the pressure in a pipe network as the acoustic wave travels to the sensor, as well as mechanical damping accounting for sections of different materials, offering a way to visualize the problem (Figure 4).

⇨ ROUTINE USE AND SIMULATION APPS

With routine use of the computational model, Perrier saw the advantage in building a custom simulation app. Based on his COMSOL Multiphysics® analysis and using built-in tools in the software, he created his own app that combines acoustic-structure interaction, pipe acoustics, and time-dependent and frequency studies (Figure 5). The app allows the user to vary geometry and material properties in multiple runs, and analyze a pipe segment or an entire network.

Using the app a user can define a water main network by specifying segment lengths, number of segments, and pipe characteristics. Speed of sound is computed by selecting material properties from a predetermined list, such as cast iron or plastic. The simulation then incorporates the results from field measurements, which a user would manually enter based on correlations to predict leak locations.

Turning the multiphysics model into a simulation app is convenient for

> "By building simulation apps I can share a complex model with colleagues and make it accessible anywhere."
>
> — SEBASTIEN PERRIER, R&D ACOUSTICAL SCIENTIST, ECHOLOGICS

interacting with others in the company. "By building simulation apps I can share a complex model with colleagues and make it accessible anywhere," Perrier said. Simulation apps can be password protected and deployed with a local installation of the COMSOL Server™ product, making it possible to quickly push app updates and maintain confidentiality.

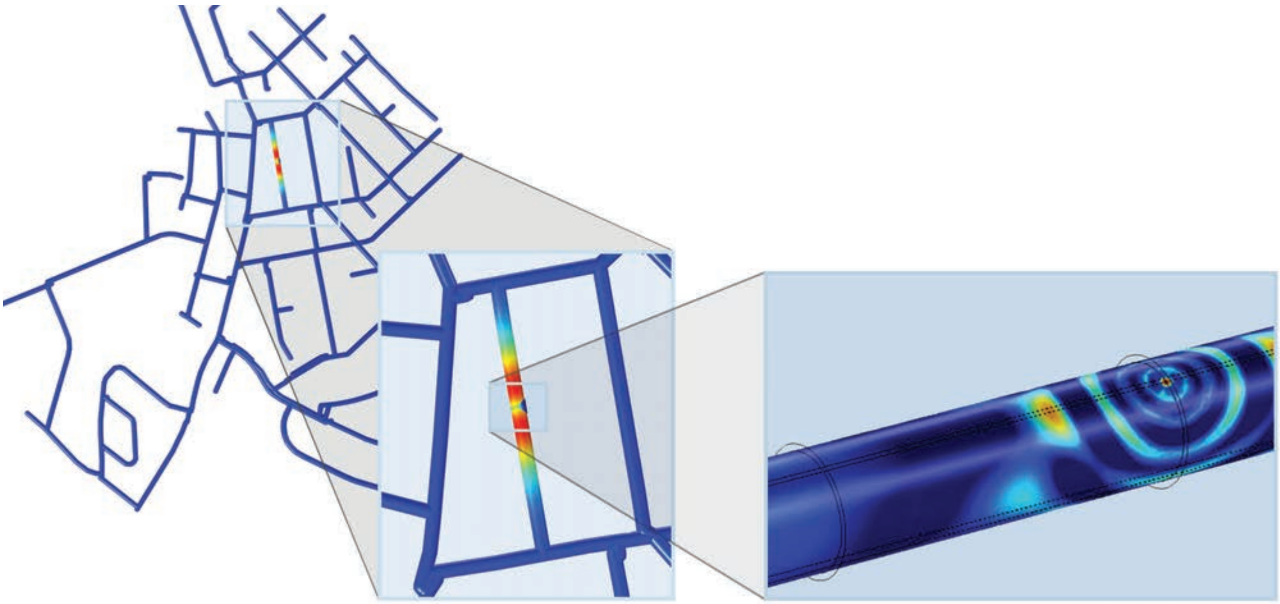


FIGURE 4. Sound propagation analysis of a leak noise in a pipe network. The plot shows the acoustic pressure in the area surrounding the leak.

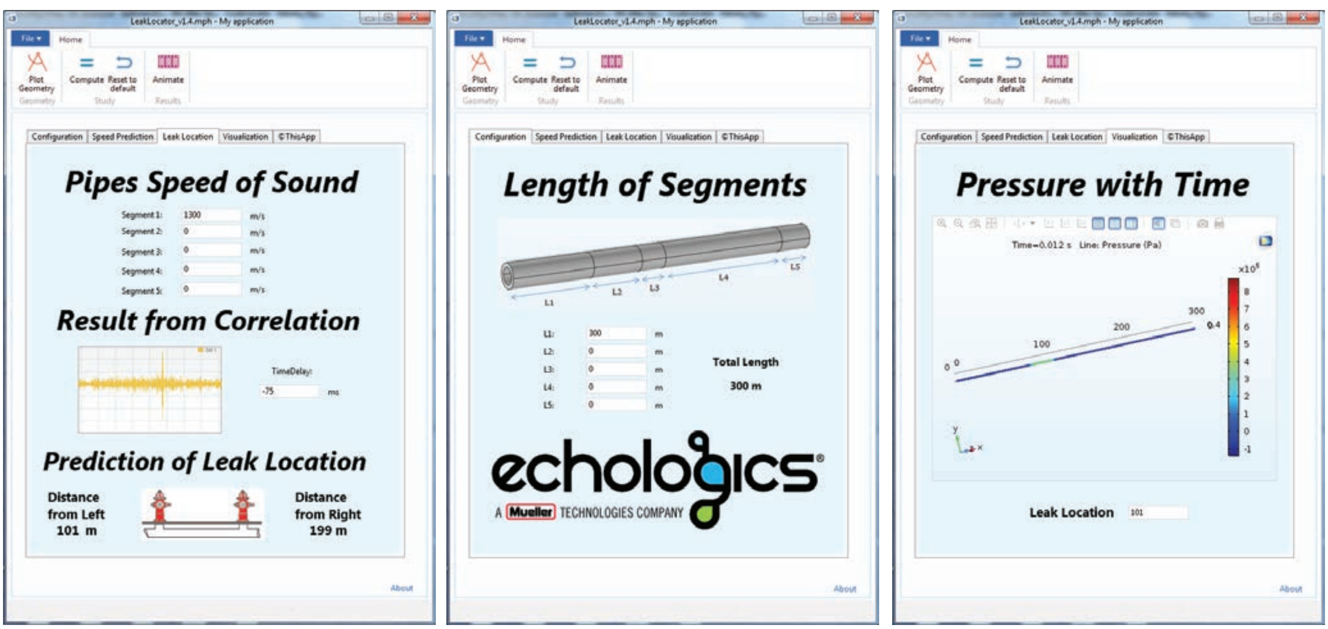


FIGURE 5. An easy-to-use interface guides a user to predict an accurate leak location by defining geometry and pipe characteristics. The app calculates the speed of sound in the pipe and allows the user to visualize, with an animation, the sound propagation from the leak location, while hiding the complex calculations for acoustic-structure interaction and location prediction.

This was a key attribute for him, noting that much of what he does is confidential. He created the app so it could be run by field engineers' on-site.

He expects that the app will be broadly used within Echologics. The key is for Echologics field engineers to be able to quickly and accurately find leaks without having a detailed understanding of the mechanics or mathematics behind the simulation. A powerful tool, in Perrier's vision, is a simulation that visualizes the propagation of sound and lets users see whether the speed of sound is decreasing or increasing when the geometry or material properties change. ❖

Sebastien Perrier, R&D acoustical scientist at Echologics.

Revving up Electrohydraulic Power Steering with Virtual Prototyping

FZB Technology uses multiphysics simulation to guide design improvements for electrohydraulic power steering systems.

by **LEXI CARVER**

If you've ever driven a car with no power steering, you'll recall having to pull very hard on the steering wheel to turn the tires to match your movements. Thankfully, those days are past us. Power steering systems — which make driving much more comfortable by providing assistance through steering gears — have gone through many iterations over the years and continue to evolve with improved designs.

Until the 1990s, hydraulic and electric versions were most common. But these were the forerunners to the birth of a more fuel-efficient method called electrohydraulic power steering (EHPS). EHPS builds on the conventional hydraulic setup, but relies on an electric motor to power a hydraulic pump, rather than power from the car's engine (Figure 1). Since the motor output is adjusted according to steering wheel angle and vehicle speed, much less power is wasted.

The pump sends fluid from a reservoir to steering gears, which apply extra torque to turn the tires when the driver turns the steering wheel. The system also includes an electronic control unit (ECU), torque sensor, valve to control fluid pressure, and a pipe system.

⇨ THE INTRICACIES OF EHPS

Designing a system with so many interrelated components is no small task, as the response of one part frequently depends on another. Seemingly minor adjustments can greatly affect successful function, efficiency, and reliability.

"One tool that expedites the design refinement process is multiphysics simulation," explains Feng Qi, senior mechanical engineer at FZB Technology in Plymouth, MI. FZB offers R&D for the automotive market, including the development of motors, sensors, keyless RFID ignition systems, as well as EHPS.

Engineers at FZB frequently use CAD and the COMSOL Multiphysics® software to model components of their EHPS designs (Figure 2). This helps them understand the behavior of the inner workings of the system, and move as close as possible to a final design before shifting from virtual prototyping to physical testing.

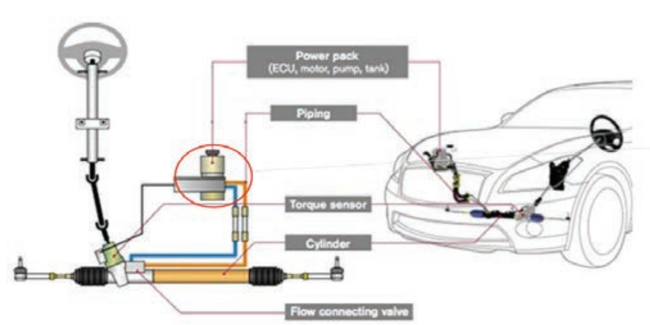


FIGURE 1. Schematic of electrohydraulic power steering (EHPS) system.

"Simulation helps us clearly understand a problem so that we can meet automotive requirements before building a physical model," Qi remarks. "We need to know how the system will perform at many levels: mechanical, thermal, fluid, acoustic, and electromagnetic."

He explains that validation and physical testing are expensive and time consuming, taking up to six months — but even after testing, a successful prototype will still need further optimization. "That would be too slow for the design cycle, so we use simulation to speed up the process. We regularly talk with the engineers, for example, at Chrysler to improve our design in COMSOL software before any physical validation. Otherwise, we wouldn't be able to meet the requirements."

Seeking insight into the behavior under the hood, Qi's team at FZB modeled the major components of their latest EHPS design: the ECU, isolated mounting bracket, permanent magnet based motor, fluid reservoir, and helical gear pump. They performed simulations on each part separately, but also ran multiphysics analyses on the entire complex assembly, which differ in design for each vehicle model.

These analyses of thermal, mechanical, fluid, and electromagnetic phenomena helped the team to more quickly solve problems surrounding thermal performance, dynamic motion control, fluid delivery in the pump, and noise, vibration, and harshness (NVH) (Figure 3).

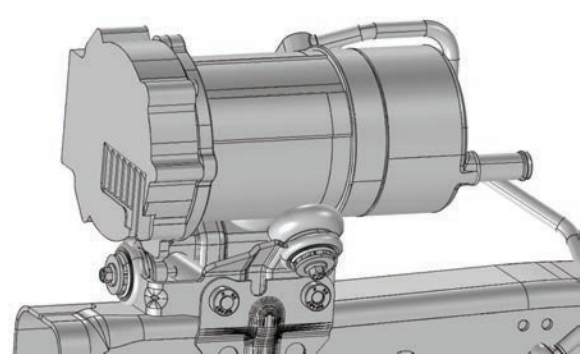


FIGURE 2. Geometry used for multiphysics analysis of the EHPS design. This design includes fins to support heat transfer to the surrounding air, mounting components, and fluid ports.

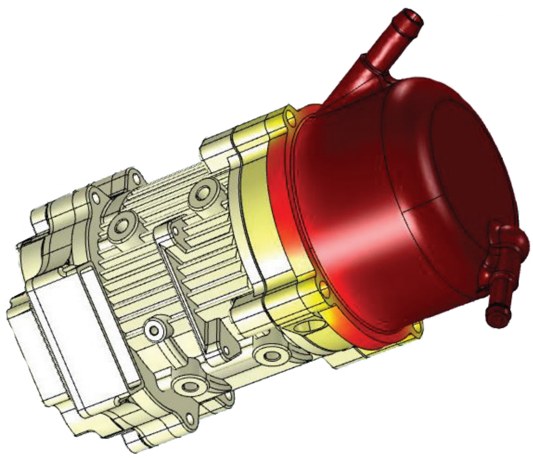


FIGURE 3. Steady-state and transient simulations were performed to obtain the temperature distribution in different parts when the pump is tested under various vehicle loads. The simulation results suggested ideal ranges for the local geometry, motor design parameters, size and number of vias on the ECU, and other thermally conducting features.

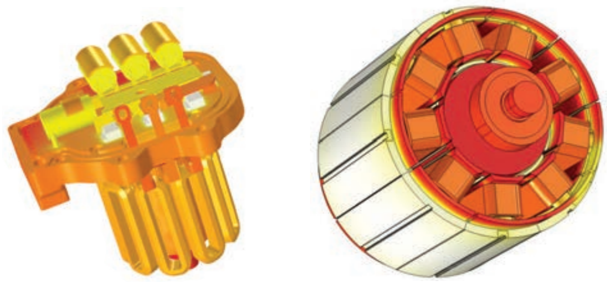


FIGURE 4. Temperature throughout the ECU and rotor assembly (left), and stator and rotor assembly (right). The model predicts system thermal performance when the product is installed on the vehicle.

⇨ HEAT UNDER THE HOOD

One major factor in system performance is the ability to operate in a safe temperature range. Qi created a model that accounted for heat transfer in the pump and heat generation in the fluid that lubricates the steering gears. Using the fluid temperature as a variable boundary condition, his team was able to predict temperature distribution throughout the system for different operating scenarios.

Qi explains that the pump is under the most strain when a car's wheels are locked against a curb while a driver tries to turn them. When this happens in real life, the car's battery will still send power to the pump even though the wheels are stuck, generating heat in the ECU and motor magnet.

Based on operating conditions supplied by car manufacturers, his team was able to model how the power steering fluid would behave in this scenario. They also studied how the ECU components, such as MOSFETs and the wiring harness, would respond to temperature levels from the heat generation

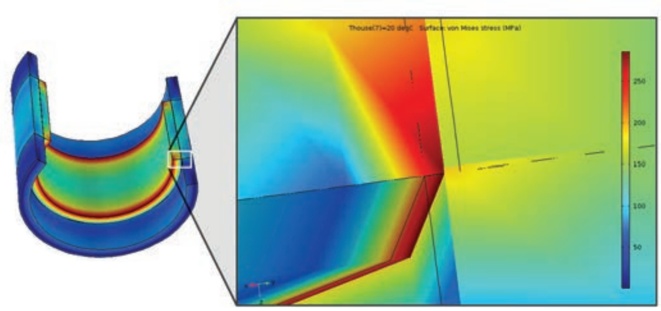


FIGURE 5. Stress levels in a simulation analyzing the interference fit of the housing and stator.

occurring when the car's wheels are locked. They used a multiscale modeling method, beginning with individual component simulations that they integrated to the system level, and correlated the results to physical test data. With this analysis, they were able to tweak boundary conditions and material properties to understand various configurations.

Temperature distribution also affects structural components such as the housing, stator, rotor, and rods in the motor (Figure 4). Thermal expansion of the metal affects the motor's efficiency, requiring more torque and RPM to deliver a desired power output to the pump. Fluid properties such as dynamic viscosity and density also change with temperature, requiring the gears to be adjusted in order to maintain smooth, consistent handling.

"This is the most challenging situation, when the vehicle is not moving and the pump has to do a lot of extra work," Qi says. "We want to stay in the temperature range where parts won't fail, so we modeled the setup for extreme situations to make sure it would hold up and perform just as well."

> *"Carrying a vehicle from concept to market depends on many factors... We needed a truly multiphysics tool for this cross-disciplinary teamwork."*
>
> — STEVEN QI, MANAGER, FZB

Qi modeled changes in the wall thicknesses of the motor, stator, and pump due to the thermal expansion and to see whether stress levels would surpass the yield stress of any part (Figure 5). The motor's stator presented a unique challenge, where thermal expansion would cause it to fail. Because they deduced this early on from the COMSOL simulation results, the team added a groove that would allow it to change shape without causing trouble.

This geometry factor became critical for considering the interference fit and amount of the housing and stator. Because the housing and stator have different coefficients of thermal expansion (CTE), the interference amount and geometric thickness needed to be carefully selected to ensure that neither part would fail within the temperature range.

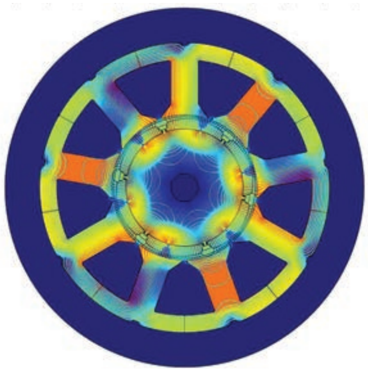


FIGURE 6. Simulation showing magnetic flux density and magnetic vector potential in a time-dependent study of the EHPS motor behavior.

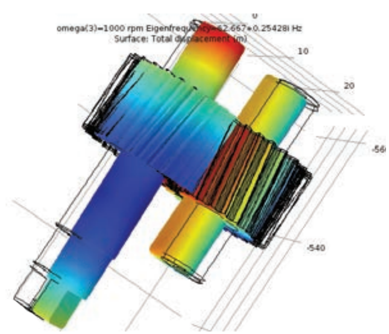


FIGURE 7. COMSOL results showing displacement of the helical gears for an RPM of 1000 and eigenfrequency of 2718.2 Hz, following the import of data from a fluid analysis in PumpLinx® software.

⇨ FLUID, NOISE, AND ELECTRONICS ALL PLAY A ROLE

The team also built an electromagnetic model to analyze the performance of the helical magnet and helical gear pump for different time steps during the pumping process (Figure 6). This gave them an understanding of how well the motor would perform over time, including an accurate estimate of heat loss on the coils and irons. This led to changes in the geometry that would accommodate a more uniform temperature distribution among components and parts.

They coupled the electromagnetic simulation to a CFD analysis to understand how it influenced fluid delivery and pump efficiency. They used PumpLinx® software, a program specifically designed for modeling pumps, to obtain fluid efficiency, flow rate, and pressure ripples.

Qi transferred the fluid data into his COMSOL Multiphysics® model, made geometry updates through SOLIDWORKS® software, and then created an acoustics simulation to study the vibrations (Figures 7 and 8). An additional rotordynamics simulation helped him identify critical speeds where the vibrations would increase dramatically, causing the gears to fail. This would generate abnormal noise, and decrease efficiency.

"Not only did we need to understand how loud the system would be, we also needed to know how it affected the electromagnetic and fluid behavior," Qi says. "They're all interconnected. We modeled a pressure ripple in the fluid, and in COMSOL analyzed how the ripple altered the airborne noise. From the results, we were able to optimize bearings, shafts, and the shape of the helical gears and fluid pressure release grooves in the helical gear pump bushing."

⇨ PAVING THE WAY FOR IMPROVED EHPS

Ultimately, the FZB team made significant design improvements to their pump geometry based on their COMSOL results. From the simulations they also generated a report on the power consumption limits in order to guide the design engineers in meeting automotive requirements. They studied how different boundary conditions affected energy consumption and power output from the pump, and checked the simulation results for different scenarios against data from real driving tests.

"We chose COMSOL because we needed to analyze all the coupled physics behavior," Qi concludes. "Successfully carrying a vehicle from concept to market depends on many factors, and the timing of the design cycle can be very tight. We needed a truly multiphysics tool for this cross-disciplinary teamwork. COMSOL is very powerful for connecting many areas of physics with different boundary conditions that provide us with an accurate picture of how our EHPS design will perform." ❖

REFERENCES

1. *Qi, F., Dhar, S., Nichani, V., Srinivasan, C. et al., "A CFD study of an Electronic Hydraulic Power Steering Helical External Gear Pump: Model Development, Validation and Application," SAE Int. J. Passeng. Cars - Mech. Syst.*

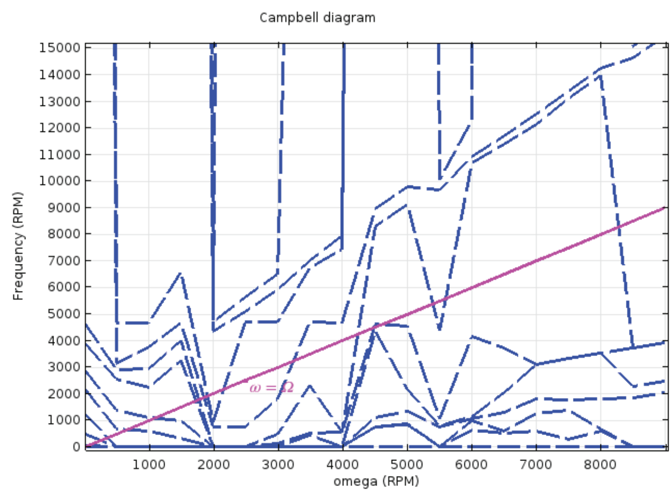


FIGURE 8. Campbell diagram generated in COMSOL showing the variation of eigenfrequencies with angular velocity for the helical gears.

Top row: Jinming (Jim) Yang; CEO, Zhonghui (Max) Bing; director, Steven Qi, manager, FZB.
Bottom row: Liang (Leon) Yang, manager, FZB; Dahong Yu, executive director; Ying Xie, manager; Fuxin Dare Automotive Parts Co., Ltd., China.

HOW FAST DO ELEVATED TEMPERATURES REACH THE CELL INTERIOR?

Numerical simulation is used to investigate stacking of lithium-ion cells during the fabrication of high-end battery systems.

by JENNIFER HAND

The performance and durability of lithium-ion (Li-ion) batteries are heavily influenced by their operating temperature. Their performance decreases at low temperatures while the battery degrades quickly at high temperatures. This means that overall reliability is compromised, creating a potential safety issue.

Industry research has led to standards regulating the ability of a battery to withstand fluctuations in temperature when it is in operation. In contrast there has been much less focus on the temperatures that batteries are exposed to during the manufacturing process, which includes plasma pretreatment, UV curing, laser welding, ultrasonic joining, hot stacking, and hot gluing. A Li-ion battery may contain thousands of individual cells, which have to be stacked together. This is typically done through an assembling procedure that may involve various heat treatments, some of which can be extremely intense and expose the casing or other parts to high temperatures for short times.

Gerd Liebig of NEXT ENERGY EWE Research Centre for Energy Technology at the University of Oldenburg, Germany, explained, "It is already well known that certain processes such as welding greatly increase the temperature within a battery. What is not known is the extent to which such elevated temperatures could propagate within and compromise a cell."

Pamina Bohn from the University of Oldenburg and scientists from NEXT ENERGY set out in close collaboration to investigate whether it was possible for the manufacturing process to cause irreversible damage that would affect stability and capacity due to the onset of an electrochemical degradation process. As experiments can be lengthy and expensive with many safety precautions, the strategy was to validate their mathematical model. The research team used numerical simulation to investigate different operating scenarios and place probes to inspect results at any point in the model, which is impractical if not impossible during experiments.

⇨ DESIGNING A THERMAL STRESS EXPERIMENT

The first step was to set up a physical experiment to measure temperatures reached inside a prismatic lithium dummy cell when it was subjected to short term thermal stress. The goal was to collect data that could be used to validate the mathematical model and investigate the effect of various processes during cell manufacturing. The cell consisted of a double-coated anode and a double-coated cathode surrounding a polyolefin separator, rolled up together. The rollup structure, known as the jelly roll, was soaked with organic solvents to mimic the battery electrolyte. Eight temperature sensors were placed on the cell: three in the middle of the jelly roll, three at the surface of the cell winding and two on the copper and aluminum current collectors of the anode and cathode, also known as arresters (Figure 1).

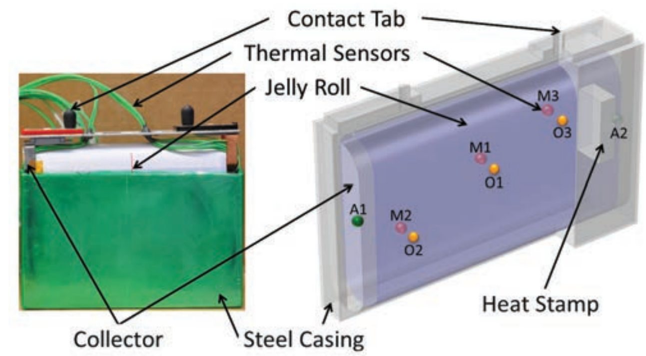


FIGURE 1. Left: Prismatic Li-ion dummy cell. Right: CAD geometry used to model the cell. The cell is equipped with eight temperature sensors: O1, O2, and O3 on the surface of the jelly roll; M1, M2, and M3 in the middle of the jelly roll; and A1 and A2 attached to the current conductors. Probes have been placed in the COMSOL® model at the same locations.

⇨ MODELING THERMAL BEHAVIOR

The team also created a 3D replica of a commercial prismatic lithium cell in Autodesk® Inventor® software and imported this into the COMSOL Multiphysics® software. They modeled heat transfer by conduction due to an external heat source at different positions on the cell corresponding to different manufacturing processes, and the natural convective cooling on other areas of the cell surface.

The physical and thermal properties of the individual materials were experimentally defined and mathematically homogenized into one jelly roll domain within a prismatic steel housing. "Due to the anisotropic nature of the cell components, directional dependence of the thermal parameters needed to be considered in the model," comments Liebig.

In the simulation, a rectangular heat stamp was positioned as it had been for the physical experiment. Figure 2 shows the temperature distribution 60 s after the heat stamp is set on the cell surface. Adaptive mesh refinement was used to adopt a finer discretization in regions where temperature gradients were higher, ensuring high-accuracy results.

⇨ LOOKING FOR SIGNS OF DAMAGE

The multiphysics model came close to replicating the behavior of the dummy cell. After validating the model, team members were in a position to simulate temperature propagation within the cell during various manufacturing processes.

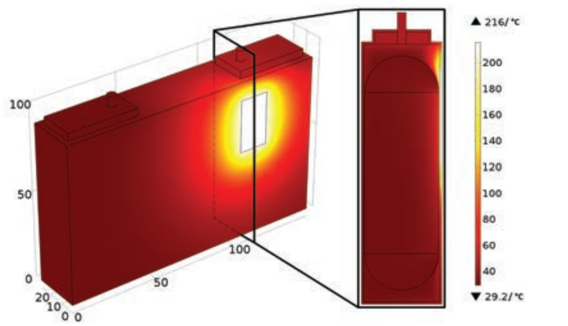


FIGURE 2. Numerical results showing the temperature distribution 60 s after a 50 W heat stamp is applied to the housing surface of the cell.

Figure 3 illustrates the distribution on the tab of the cell when a temperature load of 1100°C, typical for welding, is applied for four seconds. Heat diffuses into the cell, causing temperature to rise above 100°C. Even when the external heat source is removed the temperature within the cell continues to rise, reaching 138°C in the jelly roll four seconds after the external heating has ended.

Bohn notes, "This temperature level will induce irreversible damages such as decomposition of the electrolyte, which is highly temperature sensitive, and changes to the characteristics of the solid-electrolyte interphase. These effects will cause not only local material damages, but also overall capacity losses and an increase in the cell resistance."

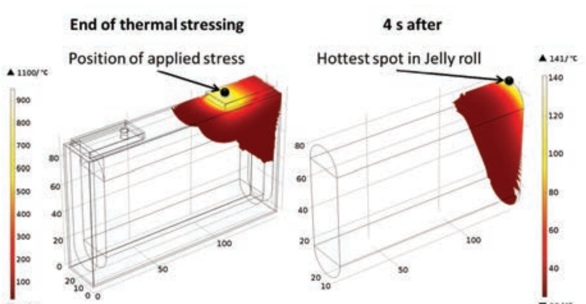
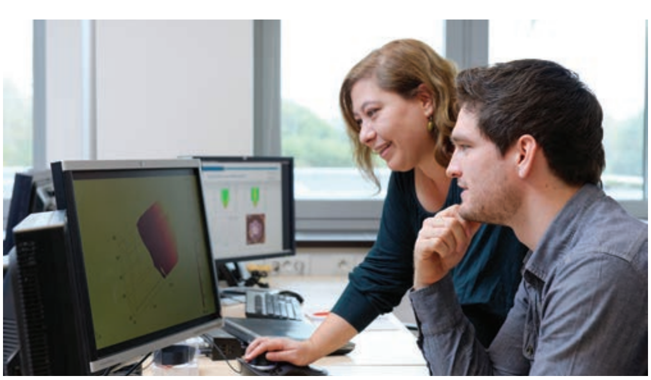


FIGURE 3. Simulation of temperature propagation after tab welding at 1100°C. Temperature distribution after 4 seconds of thermal stressing (left) and 4 seconds after the heat is removed (right) are shown.



Gerd Liebig and Lidiya Komisyska, NEXT ENERGY.

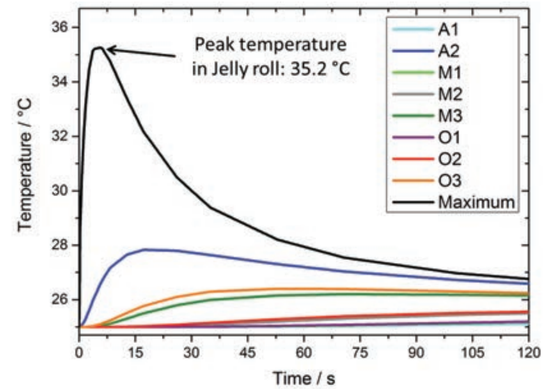


FIGURE 4. Probe temperature profiles after 0.2 seconds of thermal stressing at 1100°C.

⇨ DELIVERING POWER WITHOUT DAMAGE

The researchers wanted to confirm that the high power density of a laser beam enables high welding rates yet limits heat input into the battery cell. This technique is very fast and due to the high power density of the beam, a large variety of metals can be used.

In order to simulate laser welding, a temperature of 1100°C was applied on the cell tab for 0.2 seconds (Figure 4). The team determined that heat propagated along the cell housing, causing moderate temperatures which did not exceed 36°C and were therefore not a danger to battery components. The simulated temperature distribution within the jelly roll is in good agreement with thermographic images taken several seconds after laser welding on a 26650 Li-ion cell.

⇨ A FOUNDATION FOR THE FUTURE

Liebig says, "We now have a trustworthy simulation tool. COMSOL Multiphysics is intuitive and provides helpful tools. It is so easy to adapt to our needs, from materials to boundary conditions. The different physics interfaces, geometry tools, and flexibility can save a tremendous amount of time."

The team concludes: "Because there are so many types of batteries for different applications, it is not possible to have one perfect material, size, or shape and our model provides huge scope for future research. We can vary the geometry, the application, and materials. Thanks to numerical simulation we can confidently continue investigating lithium batteries." ❖

REFERENCES

1. G. Liebig, P. Bohn, Temperature Propagation and Effects by Short-Term Thermal Stressing of Lithium-Ion Cells, Battery Safety Conf. 2015.

2. P. Bohn, S. A. Garnica Barragan, L. Komsiyska, G. Wittstock, Performance and the characteristics of thermally stressed anodes in lithium ion cells, Journal of The Electrochemical Society, 162 (2) (2015).

3. P. Bohn, G. Liebig, L. Komsiyska, G. Wittstock, Temperature propagation in prismatic lithium-ion-cells after short term thermal stress, Journal of Power Sources, Volume 313, 1 May 2016.

4. M.J. Brand, P.A. Schmidt, M.F. Zaeh, A. Jossen, Welding techniques for battery cells and resulting electrical contact resistances, J. Energy Storage 1 (2015).

On the Cutting Edge of Hearing Aid Research

Engineers at Knowles bring the hearing aid industry together to fight feedback with multiphysics simulation.

by GARY DAGASTINE

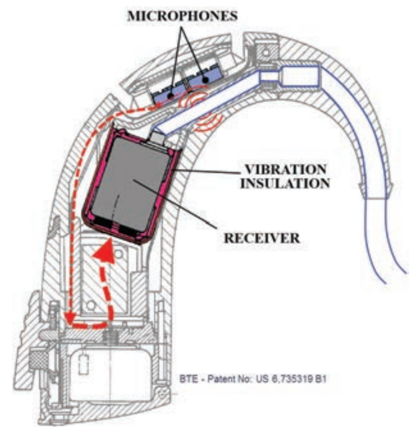


FIGURE 1. A typical BTE hearing aid includes microphones, vibration insulation, and a receiver, among other components. The tight spacing of these components invites troublesome acoustic and mechanical feedback. (Image credit: Knowles Corp.)

In the United States, nearly 20% of the population is reportedly hearing impaired — although that figure could be higher because many people are reluctant to admit they have a hearing problem. Those who are treated rely on miniature and discreet hearing aid devices to improve their hearing, hence their quality of life. Significant R&D effort is required to bring a hearing instrument from a prototype stage to a marketable hearing aid device.

Engineers face daily technical challenges in hearing aid design. Feedback is a major issue that leads to high-pitched squealing or whistling, and limits the amount of gain the aid can provide. "Feedback usually occurs when a hearing aid's microphone picks up sound or vibration inadvertently diverted from what's being channeled into the ear canal and sends it back through the amplifier, creating undesirable oscillations," explains Brenno Varanda, a senior electroacoustic engineer at Knowles Corp. in Itasca, IL.

"For many of Knowles' customers, designing a new hearing aid is a costly, time-intensive process that could take anywhere from 2 to 6 years to complete," Varanda explains. Accurate modeling helps designers select speakers, refine vibration isolation mounts, and package components to reduce the amount of speaker energy that is fed back to the microphone. The industry is in dire need of simple transducer models that will expedite that process, and provide more effective options to consumers. Complete models of speakers and microphones are quite complex, and incorporate many factors that are not necessary for feedback control. "While understanding the electromagnetic, mechanical, and acoustic physics of our transducers is important to transducer designers at Knowles, all of that complexity is not necessarily useful for our customers." Varanda says.

As a global leader and market supplier of hearing aid transducers, intelligent audio, and specialty acoustic components Knowles took a multilateral initiative to develop transducer vibroacoustic models that are easy to implement and compatible with its hearing health customers. The models are intended to help hearing aid designs graduate from a prototype stage to a final product in a more efficient manner without having to sacrifice accuracy.

⇨ HEARING AID DESIGN AND FEEDBACK

When designing hearing aids two major conflicting requirements must be accounted for by engineers. They must be compact and unobtrusive, yet still capable of providing a powerful sound output to overcome the user's hearing loss. The user is far more likely to wear a hearing aid if they are discreet and lightweight. This makes solving the feedback issue more challenging. "A common design challenge is to cram all the hardware components into the smallest space possible without causing feedback instability," Varanda continues.

A typical small behind-the-ear (BTE) hearing aid comprises microphones to convert ambient sounds into electrical signals, a digital signal processor and amplifier to process and boost the electrical signals, and a tiny loudspeaker, also known as a receiver (Figure 1). The receiver, or speaker, "receives" amplified electrical signals and converts them into acoustic energy, or sound, which is then channeled into the ear canal through a tube or an ear mold.

The receiver contains an electromagnetically controlled lever, known as the reed, connected to a diaphragm which generates sound through its oscillating motion. The internal electromechanical forces also generate reaction forces which transmit vibrations through the hearing aid package, creating sound that is picked up by the microphone. This signal in turn is magnified by the amplifier and returned again to the receiver, causing feedback. This path is shown in Figure 1.

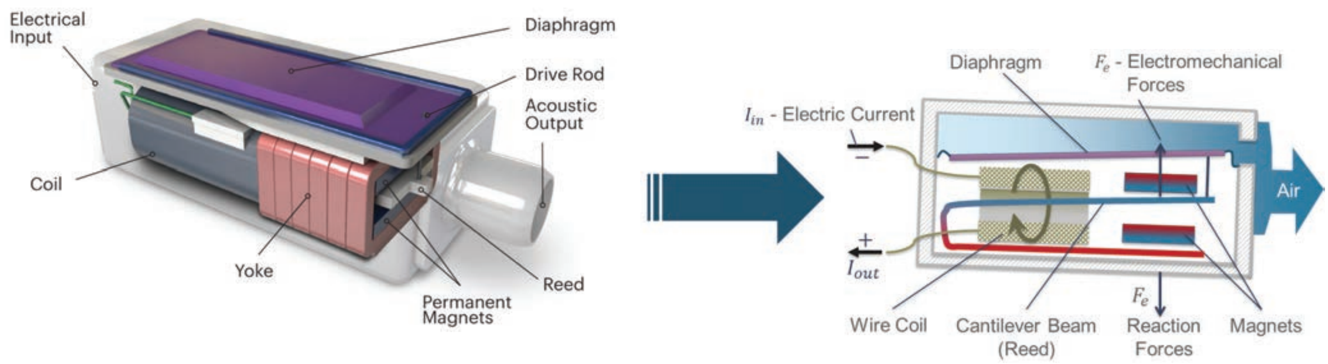


FIGURE 2. A receiver, a key hearing aid component, contains a tiny loudspeaker with an electromagnetically controlled diaphragm that generates sound. Internal electromagnetic forces cause structural vibration that results in mechanical feedback.

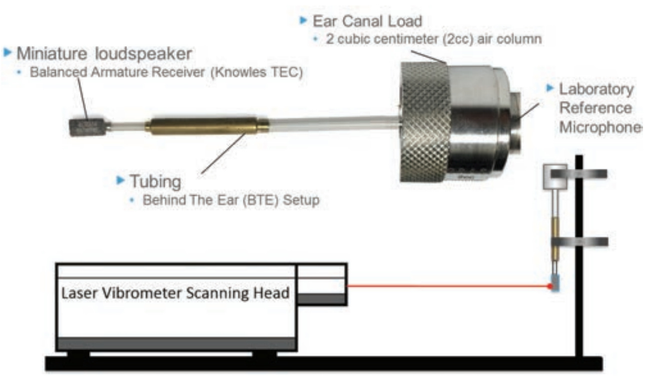


FIGURE 3. Hardware and schematic of the experimental setup.

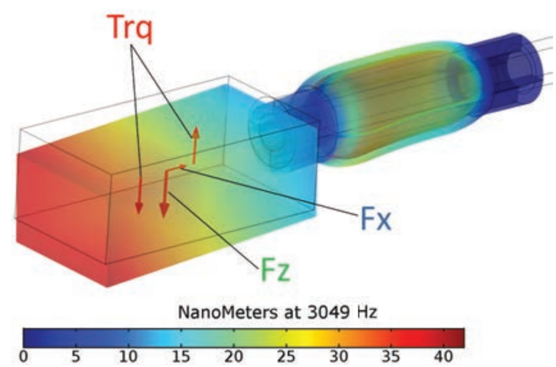


FIGURE 4. Simulation force and displacement results at 3 kHz of the receiver and silicone tube attachment.

⇨ THE "BLACK BOX" MODEL

The receiver's only function is to convert the amplified voltage signal from the microphone into sound. While the construction appears simple, the process is rather complex (Figure 2). The electrical signal is first converted to a magnetic signal, then to a mechanical signal, and finally into an acoustic signal. Each of these steps has its own frequency-dependent characteristics. Understanding the combined effects of all the internal components is vital to the ability of effectively designing receivers for all different hearing aid platforms. Engineers at Knowles have been using complex circuit-equivalents to model all of their internal electrical-magnetic-mechanical-acoustic effects since the 1960s.

Accurately modeling the full complexity of a receiver requires a dauntingly large and complex multi-physics finite element model, making it impractical for fast and efficient hearing aid design. This issue was overcome when Dr. Daniel Warren, a hearing health industry expert in receiver and microphone research, introduced a 'black box' model in 2013. The design uses a minimum amount of simple circuit elements to capture the essential electroacoustic transfer function between voltage and output sound pressure level for balanced armature receivers, while leaving out factors that are unimportant to feedback control.

A key step to simplifying the model was when Warren and Varanda demonstrated that the simplified electroacoustic circuit could be converted into a powerful vibroacoustic model while adding very little complexity to the model. "The conversion is achieved by probing a section of the 'black box' circuit where the voltage across inductors is directly proportional to the internal mechanical forces responsible for structural vibration," Warren explains.

The "black box" and vibroacoustic models needed to be tested and validated against realistic acoustic and mechanical attachments to the receiver before designers could start using them for product designs. A worldwide collaboration between Knowles and its hearing health customers got started in 2014 to validate the models using the COMSOL Multiphysics® software and industry standard tests.

⇨ WORKING TOGETHER ON VALIDATION

To validate the models, engineers needed to measure the acoustic output and vibration forces at the same time, using a structure that could be easily modeled in FEA. Like common hearing aid tests, this test involved connecting a receiver to a short section of tubing leading to an enclosed cavity with a two cubic centimeter (2 cc) volume, which is a standardized ear canal acoustic load as shown in Figure 3. The acoustic pressure inside the cavity is measured with a laboratory-grade microphone. To help verify the robustness of the model, the receiver was also measured using a complex tubing assembly similar to a BTE hearing instrument. The long tubing in this design varies in diameter, and is long enough to support multiple acoustic resonances. At the same time the acoustic output was being measured, the receiver's structural motion was captured by a laser vibrometer. Both translational and rotational motion were measured by observing the motion at multiple points on the surface of the receiver housing.

Warren and Varanda collaborated with several Knowles customers to perform the measurements described above. With the help of COMSOL Multiphysics, they were able to implement the simplified vibroacoustic circuit model into a simulated replica of the test setup described above. The simulation couples the mechanical interaction between the motion of the receiver and the silicone tubing attachment, thermoviscous losses within the various tubing cross sections, and acoustic pressure loads inside the cavity and tubing with the internal electro-magnetic-acoustic effects in the "black box" receiver model.

The COMSOL model revealed the dependence of the output pressure and mechanical forces on the applied voltage, frequency, and material properties. Figure 4 shows the displacement results from the simulation at 3 kHz and the reaction forces coupled to the receiver.

When Varanda compared the results of simulations with the physical measurements, they showed excellent agreement (Figure 5). The forces acting on the diaphragm and the reed are acoustically dependent on the output sound pressure. However, the coupling between the forces acting on the diaphragm with the structural reaction forces proves to be, as expected, directly proportional.

> "With multiphysics simulation hearing-aid designers now have the capability to reduce feedback and improve overall performance better, faster, and more economically than before, which will lead to better options for people who are hearing impaired."

⇨ SPREADING THE KNOWLEDGE

Knowles shares their model to empower engineers at other hearing aid companies to solve their own system feedback troubles. With a complete representation of the acoustic, mechanical, and electromagnetic behavior inside the hardware, designers are well set up to virtually optimize their products.

"COMSOL is one of the few modeling and simulation tools that can easily couple the lumped 'black box' receiver circuit with acoustics and solid mechanics," says Varanda. "Until now, verifying and optimizing hearing aid designs has been as much art as science. We will be very happy to see new hearing instruments designs that have benefitted from these models."

By joining forces, the intercompany effort has made it easier for everyone in the hearing health industry. "Ultimately, hearing aid designers don't want to get bogged down with complex transducer models and time-consuming simulations. They simply want focus on their own design and to swap transducers in and out to see how everything will work together," he adds. "This COMSOL model enables them to do this. The behavior of hundreds of transducers can be easily compared for one hearing aid package."

Hearing aid designers now have the capability to reduce feedback and improve overall performance better, faster and more economically than before, which will lead to options for people who are hearing impaired. ❖

Brenno Varanda, senior electroacoustic engineer, Knowles Corp.

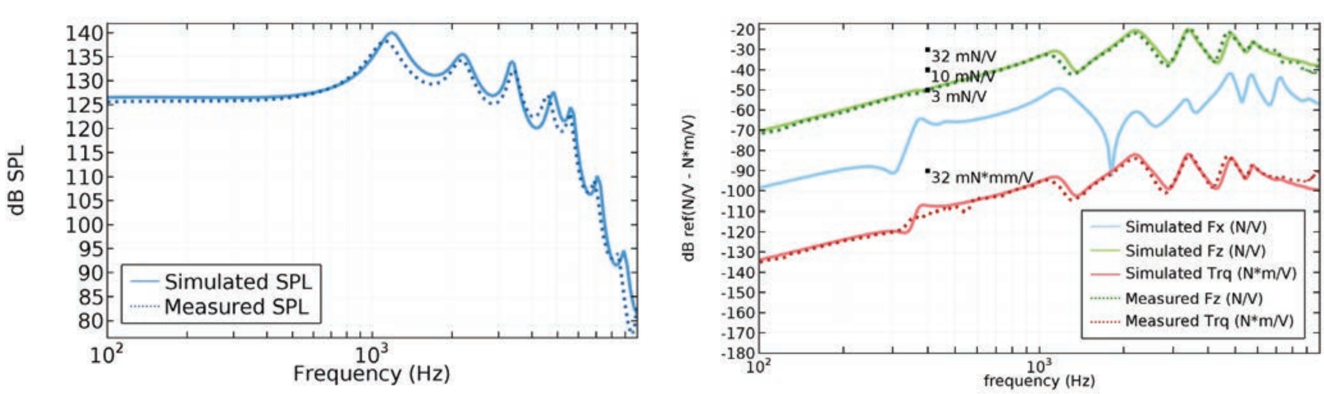


FIGURE 5. Left: Measured (dotted line) vs. simulated (solid line) sound pressure level inside a 2-cc coupler. Right: Measured (dotted line) vs. simulated (solid line) forces and torque acting on the receiver.

SEEKING OUT ELECTRICAL ARCING REGIONS IN SATELLITE SYSTEMS

Electrical arcing discharge in orbiting satellites can cause system failure, but is hard to predict. Engineers at the Russian Institute of High Current Electronics have adopted multiphysics software to find the critical regions where failures originate and to protect onboard equipment.

by **GEMMA CHURCH**

In 1995, Boeing Satellite Systems introduced a new family of communication satellite buses, the bodies that contain power, control, and propulsion systems. They used a high-voltage bus connected to a 100 V stabilized power source, instead of the standard 27 V voltage. This introduced an increase in operating voltage that decreased operating currents and lowered the corresponding ohmic losses in the conductors. However, it also introduced a potentially catastrophic failure to the satellites' electronic systems: electrical arcing (Figure 1).

Vasily Kozhevnikov, researcher at the Institute of High Current Electronics in Tomsk, Russia, explains: "The transition to the new standard of operating voltages has led to the problem of an electric arc ignition between the elements of the electronic circuit boards. In order to keep the mass of the satellite as small as possible, the space inside the circuit housing is not filled with an insulator or built to hold a vacuum. But that allows electric arc discharge or discharge cascade that can potentially spread over a large volume of onboard equipment."

"The ignition of an electric arc inside the onboard satellite system always leads to partial or complete failure. In most cases, it causes the termination of satellite use," he added.

This research closely relates to the physics of a gas discharge under extreme conditions, where electrical equipment does not always perform as conventional physics would dictate. For example, electrical discharges sometimes occur below a threshold known as Paschen's minimal values, where the voltage should not normally be sufficient to start a discharge, or electric arc, between two electrodes.

"We think this research will also have potential use for the diagnostic of electronics operated under a wide range of external parameters such as pressure, ionization levels, and so on. It's widely applicable beyond the space industry and space science," said Kozhevnikov.

As electronic systems are used in increasingly extreme environments, electrical arcing is not just an issue faced by the civil space industry. It affects any electronic application designed for long autonomous work with improved fault tolerance requirements. A solution to

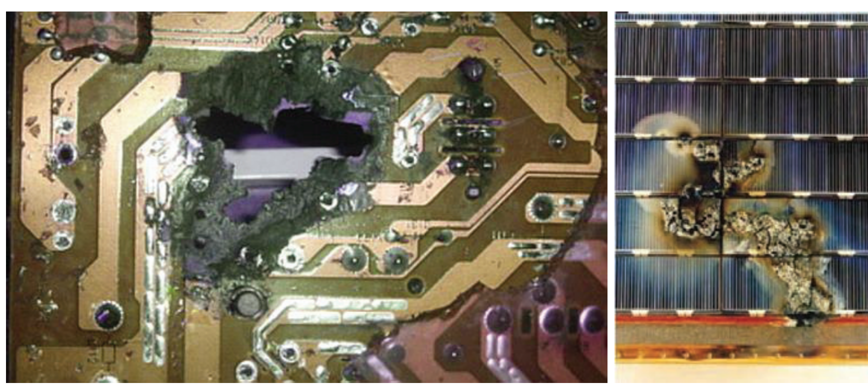


FIGURE 1. Typical damage from a primary arc in a power supply operating at 100 V.

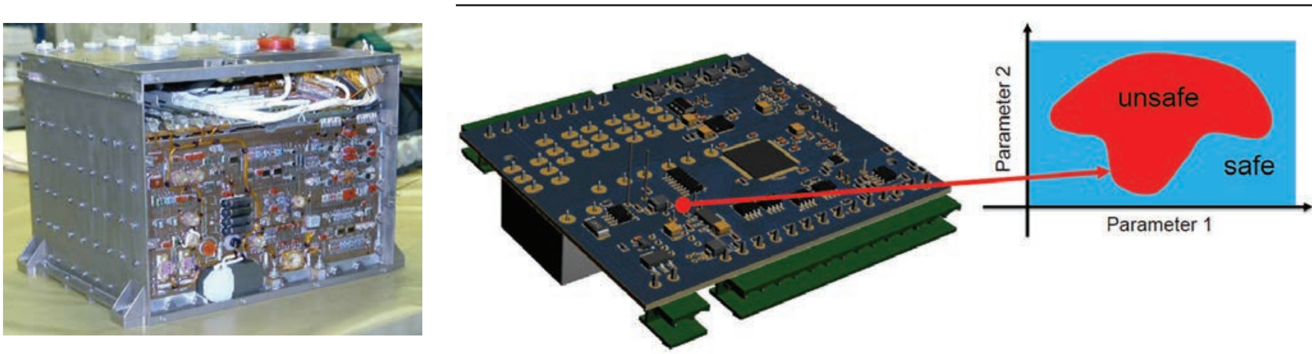


FIGURE 2. Example of a circuit board for satellite equipment. Critical regions are smaller than 5-mm wide. Engineers at IHCE must determine the range of unsafe operating conditions and properties in order to design a system that can travel aboard satellites without being destroyed.

this problem, therefore, extends beyond satellites and to terrestrial systems and underwater equipment as well.

⇨ FINDING THE CRITICAL REGION

To prevent the destruction of an onboard electronic device by a spontaneous electric arc, a so-called "critical region" must be identified, which is the area where self-sustained discharge ignition occurs. Once this potentially problematic area has been found, engineers need to conduct further investigations into what may trigger an electrical arc discharge.

Experimental studies fail to stand up to the challenge of identifying these electronic hotspots because they cannot reproduce the full range of operating parameters that exist in space orbit.

> "COMSOL made it possible to perform our research without the creation of our own computational code. We expect [it] to be most promising for our future investigations."
>
> — VASILY YU. KOZHEVNIKOV, RESEARCH ASSOCIATE, IHCE

The only remaining investigative option, simulation, also faces monumental challenges. For one, a typical onboard electronic device consists of multiple printed circuit boards distributed over a large area, placed inside a metal casing (Figure 2). Kozhevnikov explained:

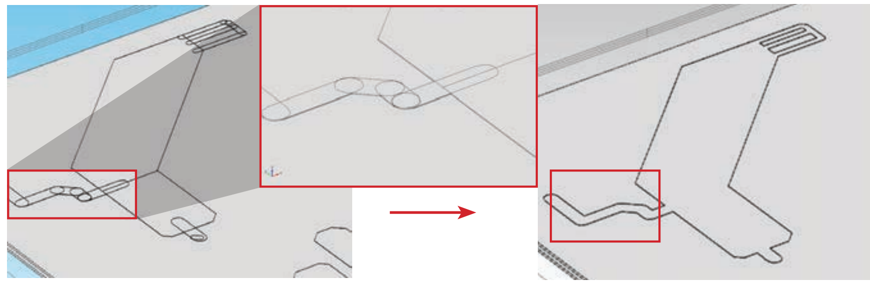


FIGURE 3. Geometry correction in COMSOL.

"The only way to identify possible self-sustained discharge regions is the numerical simulation of the discharge, but this is practically impossible for such large-scale problems due to the associated computational costs. The discharge problem is both multiphysics and multiscale."

⇨ CATCHING GEOMETRIC INACCURACIES

The Tomsk-based research team worked hard on finding a computational approach that would prove both accurate and practical. The researchers proposed a "decomposition" methodology implemented with computational tools to tackle this problem. Instead of performing a complete direct current discharge simulation for the entire electronic device, they created a custom simulation app that would autonomously partition and analyze the device to find the most probable critical regions. To this aim they used the COMSOL Multiphysics® software and its Application Builder tool to create a multiphysics model supporting the entire simulation process.

An important modeling step was preprocessing, which was carried out to apply the proper boundary conditions and import the detailed geometry of the real on-board electronic system.

With the Application Builder, the team performed preprocessing using a custom 3D macromodel method. They also implemented their own import engine with automatic correction of object boundaries. The method consisted of both import and automatic correction of object boundaries functionalities, Kozhevnikov explained (Figure 3). Without correction, these errors could have become serious obstacles in the simulation.

⇨ BREAKING DOWN THE PLASMA PHYSICS PROBLEM

After preprocessing, the modeling methodology consisted of three stages: preliminary electrostatic analysis of potential critical regions in a 3D model; extraction of field-enhancement areas and the definition of critical regions, with associated 2D models; and DC-discharge simulation of critical regions to further investigate parameters of interest.

The team initially used COMSOL Multiphysics because of its unique ability to implement all the features

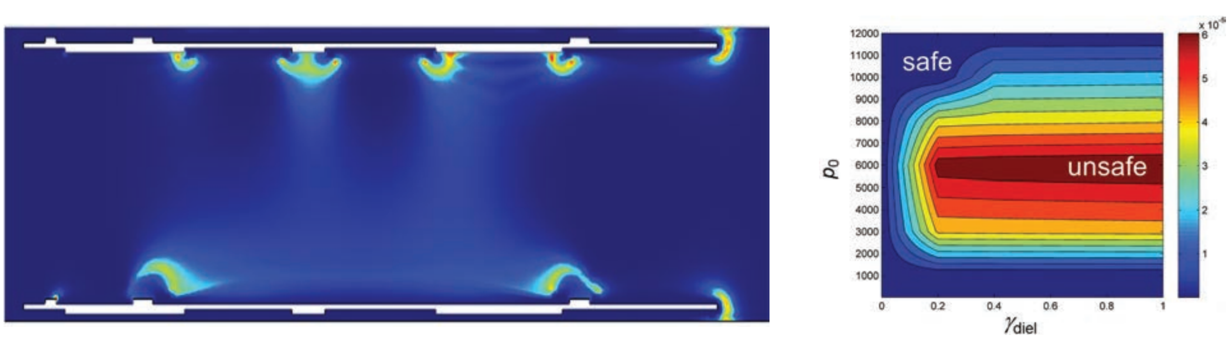


FIGURE 4. Left: Electron density distribution for the self-sustained discharge phase. This 2D model is derived from critical regions defined from the 3D model of the satellite's power supply. Right: Example of a critical parameter diagram where pressure vs. emission is shown for a critical region. The color map represents the level of discharge current density.

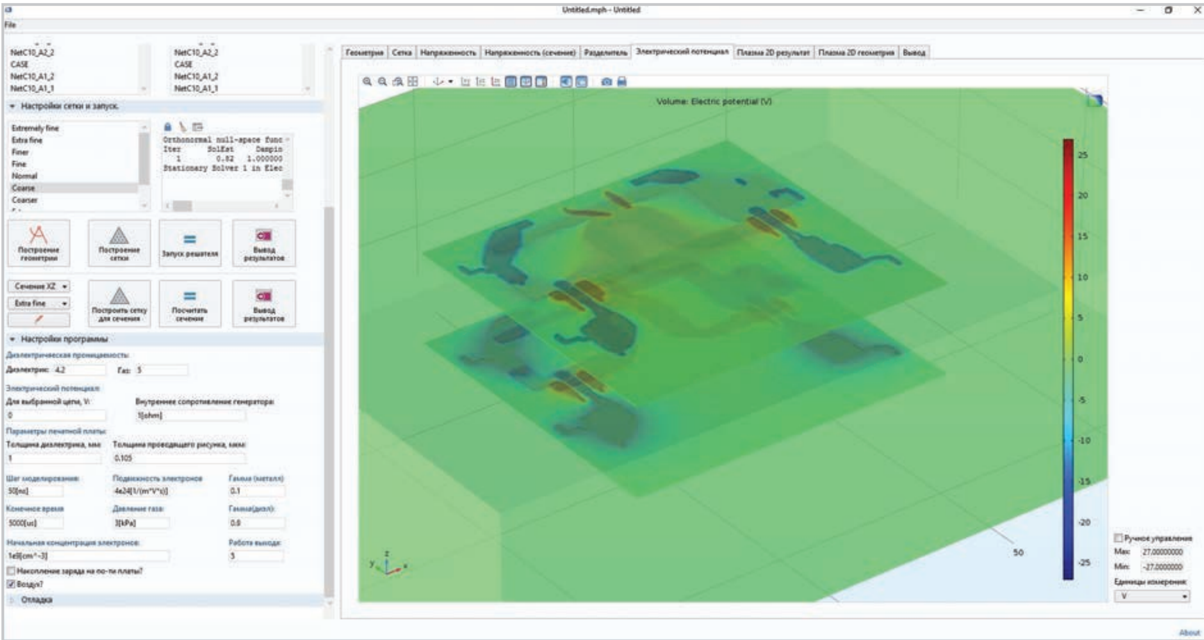


FIGURE 5. The multiphysics app Kozhevnikov developed makes it possible to vary parameters such as pressure and electron emission to search for regions where a self-sustained discharge is most probable. The app combines arc positioning with the investigation of certain regimes of discharge ignition without full-scale DC-discharge simulation and gives results such as the electric potential throughout the circuit system.

of the two-moment direct current discharge theoretical model and alter the necessary parameters. The simulation analyzed the electron density distribution and identified the critical region (Figure 4). Kozhevnikov explained: "COMSOL Multiphysics finely meets the requirements of our project, namely, an analysis of the operating pressure range. This is much faster and more convenient than a particle-in-cell (PIC) simulation for medium and high pressures."

"PIC simulations are simply unfeasible for such problems due to extensive computational costs. The simulation of simplified configurations (e.g., gas diodes) is possible, but depending on the problem, can take 5–20 times longer for medium pressures than a COMSOL simulation. The average computation time in COMSOL for this configuration is less than 2 hours."

The custom app that the team built, shown in Figure 5, hides the complexity of the physics involved in the model setup. This exposes the app user only to parameters relevant to the analysis at hand and allows for the inclusion of custom commands and algorithms.

Kozhevnikov said: "Strictly speaking, COMSOL made it possible to perform our research without the creation of our own computational code, which would be extremely complicated in light of this problem. We expect the software to be most promising for our future investigations concerned with gas discharges." Other arguments in favor of choosing COMSOL were its wide choice of pre- and post-processing tools, including CAD import features and the Application Builder.

⇨ ORBITAL AND INTERDISCIPLINARY IMPLICATIONS

There is scope to integrate such simulations with real-world investigations, Kozhevnikov explained. "If it is possible to perform fully nondestructive testing in the future, a COMSOL simulation will narrow the region of interest for experimental testing by excluding nonessential parts. Some work toward nondestructive testing development was performed by our colleagues from the Laboratory of Vacuum Electronics at the Institute of High Current Electronics, in the framework of the project we collaborate on."

"Within the spacecraft industry, the automated software system's adaptability should guarantee its continued use," he continued. "Standards in spacecraft industry change from time to time, so it is difficult to account for all the consequences of such changes. We have solved the problem of arcing diagnostics; nevertheless we expect that the voltage increase will also require serious redesign of certain on-board electronics to fit new operating conditions. Simply speaking, if the operation conditions of some device significantly differs from 'normal conditions', then you need to rebuild its architecture in the certain way. Our app provides recommendations for the redesign of printed circuit boards in order to make them more arc-resistant, but it could also be useful in designing fault-tolerant electronic systems." ❖

Vasily Yu. Kozhevnikov received his PhD in theoretical physics from Tomsk State University, Tomsk, Russia, in 2008. Since 2008, he has been a research associate with the Laboratory of Theoretical Physics of the Institute of High Current Electronics SB RAS (Tomsk). He has been using COMSOL intensively since 2012.

Simulation Transforms the Medical Device Industry

by **FREDDY HANSEN, ABBOTT LABORATORIES**

Those of us in the medical device industry strive to push the limits of technology to help alleviate pain, revive health, and extend life. Five years ago, I made the switch from a national lab to the medical device sector. At this time, the only type of simulation being conducted was computational fluid dynamics (CFD). Simulating a medical implant is particularly challenging because the human body is an incredibly complex and poorly understood control system.

It became clear that defining boundary conditions for a numerical model can be fiendishly difficult. For example, even building a thermal model, which many of us normally think of as straightforward, can get quite tricky for an implant, because of how heat is transported in perfused tissue. Another hurdle is getting the material properties right. Muscle, fat, and other body tissue can have some spectacularly odd properties, and the material of the device itself will change over time as it is subjected to a hostile environment. Metals will corrode and nonmetals will absorb fluids and diffuse those liquids and their ions.

> "Recently I've come across examples where established bench tests are considerably less accurate than simulations at predicting an implant's performance. In those cases, why not simulate?"

If it's so difficult, why simulate? Fortunately, the advantages of computer simulation are significant: time and money saved on building and testing multiple prototypes, as well as being able to "measure" anything you want, anywhere.

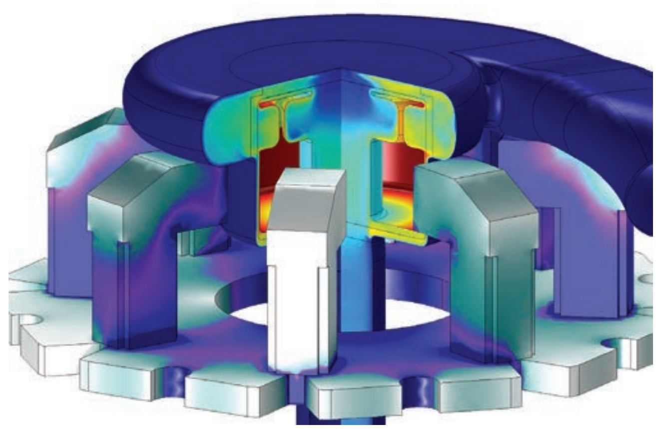
To give one example specific to medical implants, a very costly part of the design cycle includes medical device testing. While it's certainly possible in these studies to take measurements, it's often not possible to get your flow probes, thermocouples, and other sensors exactly where you want them. Simulation helps us plan and prepare for these studies. Using this technology allowed us to interpolate between experimental data acquired at different locations, which has resulted in more sophisticated analysis. Ultimately, we are able to reduce the number of tests needed. In certain situations one can even forego a study entirely, in favor of simulation.

It has been encouraging to see the current trend among regulatory bodies around the world taking steps to allow more simulation in place of physical studies in the regulatory approval process.

Often less expensive bench tests can be replaced by simulation, too. Recently I've come across examples where established bench tests are considerably less accurate than simulations at predicting an implant's performance. In those cases, why not simulate?

Simulation software has improved over the years. I especially like COMSOL because of its unparalleled multiphysics capabilities. Even when I'm not doing coupled physics, using the same interface to model heat transfer, electromagnetics, structural mechanics, or CFD, speeds up the workflow and cuts down on errors. I've also come to expect the ability to enter arbitrarily complex material properties or my own ODEs and PDEs when necessary, and couple these seamlessly into the built-in physics.

Computer simulation has a lot to offer the medical devices sector, and I've been pleased to observe the demand for and appreciation of computer simulation climb steadily over the past few years, both at my company and in the industry at large.



Left Ventricular Assist Device (LVAD) simulated in the COMSOL Multiphysics® software.

ABOUT THE AUTHOR

Freddy Hansen has a PhD in applied physics from Caltech. He lives in the San Francisco Bay area and works for Abbott Laboratories using his expertise in electromagnetics and fluid dynamics to design artificial hearts. He has written over 40 research papers; has half a dozen patents, pending or approved; and cocreated a popular college physics textbook.