COMSOL NEWS

THE MULTIPHYSICS SIMULATION MAGAZINE

FROM STUDENTS TO ENTREPRENEURS
University at Buffalo
PAGE 32

ABB POWERS UP THE TRANSFORMER INDUSTRY WITH SIMULATION APPS
PAGE 4

Rocket System

Combustion Instability Modeling at NASA
PAGE 8

COMSOL

Create, Communicate, and Collaborate with Custom Simulation Apps: Bringing Multiphysics to Everyone

Real innovation is about making technologies available for everyone to use, and this year's edition of COMSOL News reports on what is keeping simulation specialists busy on that front.

The insights offered by numerical simulation often need to be clearly communicated across several teams, as fostering collaboration within organizations is critical to their success. Users of the COMSOL® software share how, by adopting the powerful computational tools offered in COMSOL Multiphysics® software, they create detailed physics-based models and present them in intuitive user interfaces, or custom simulation apps. They also discuss how the COMSOL Server™ product enables the distribution of these applications for their colleagues and customers to use. Although numerical simulation was once the realm of only modeling experts, simulation apps are bringing the power of multiphysics to everyone.

From rocket propulsion at NASA to power transformers at ABB and technology entrepreneurship at the University at Buffalo, you will find a variety of product development and research projects featuring cutting edge modeling, simulation, and application design.

I'd like to thank all of the COMSOL users who contributed their stories for sharing their expertise and striving for excellence in their work.

Enjoy your reading.

Valerio Marra
Technical Marketing Manager
COMSOL, Inc.

INTERACT WITH THE COMSOL COMMUNITY

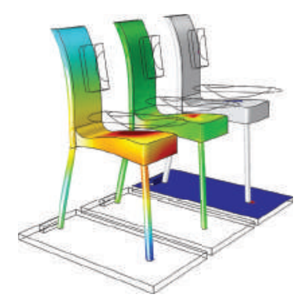
COMSOL, Inc. | COMSOL Multiphysics | @COMSOL_Inc | google.com/+comsol

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

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

COMSOL NEWS
2016

CONTENTS

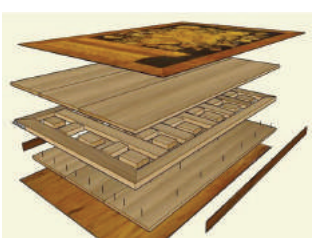


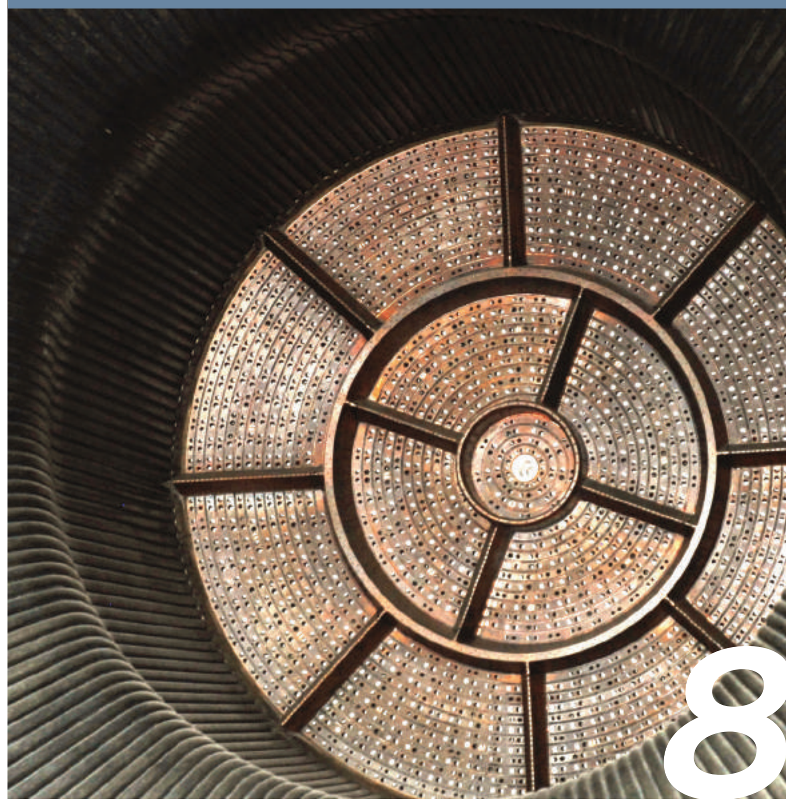
ON THE COVER

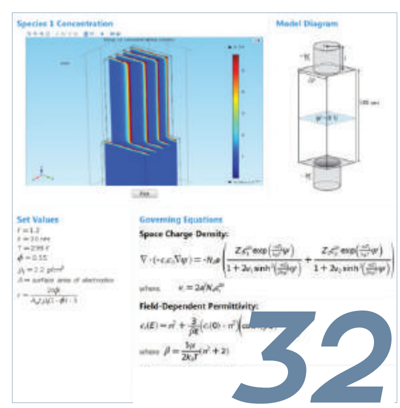
The Atlas V rocket launching from Space Launch Complex 41 at Cape Canaveral Air Force Station in Florida, USA. Image Credit: NASA/Bill Ingalls.

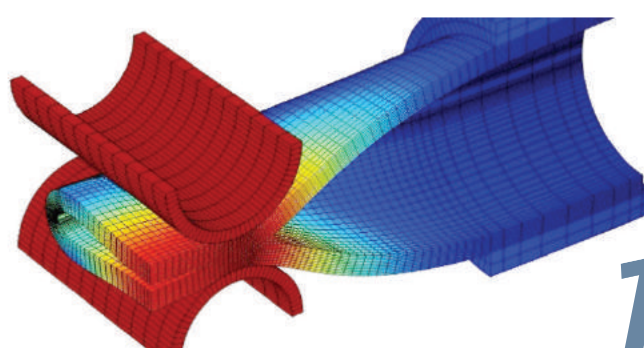
The National Aeronautics and Space Administration (NASA) does not endorse the COMSOL Multiphysics® software.

FEATURES









FROM SPREADSHEETS TO MULTIPHYSICS APPLICATIONS, ABB CONTINUES TO POWER UP THE TRANSFORMER INDUSTRY

Companies developing new and improved power transformer equipment incur costs for prototyping and testing as they work to reduce transformer hum. At ABB, a team of engineers develops multiphysics simulations and custom-built applications to offer insight into their designs.

by LEXI CARVER

For everything from cooking to charging our phones, we rely every day on the electrical grid that powers buildings like homes, businesses, and schools. This complex network includes stations generating electric power, high-voltage transmission lines that carry electricity across large distances, distribution lines that deliver power to individual homes and neighborhoods, and the related hardware used for power flow control and protection.

Among this equipment are power transformers for increasing and decreasing voltage levels in power lines that carry alternating current (see Figure 1). Power transfer with higher voltages results in lower losses and so is more desirable for transporting power long distances. However, such high voltage levels would pose a safety hazard at either end of the lines, so transformers are used to increase voltage levels at the power feed-in point and decrease them close to neighborhoods and buildings.

But transformers come with noise, often manifested as a faint humming or buzzing that can be heard when walking nearby. Although it is impossible to completely silence them, regulations require adherence to safe sound levels, and good product design can minimize these acoustic effects.

One of the biggest manufacturers of transformers used around the world, ABB (headquartered in Zürich, Switzerland), has used numerical analyses and computational applications

FIGURE 1. Photo of transformer equipment for high-voltage power lines.

in order to predict and minimize the noise levels in their transformers. Through the COMSOL Multiphysics® simulation software and its Application Builder, they have run virtual design checks, tested different configurations, and deployed their simulation results through customized user interfaces built around their models.

⇨ SILENCING SOUND FROM SEVERAL SOURCES

Transformer noise often comes from several sources, such as vibrations in the transformer core or auxiliary fans and pumps used in the cooling system. Each of these sources needs to be addressed differently to reduce noise.

ABB's transformers comprise a metal core with coils of wire wound around different sections, an enclosure or tank to protect these components, and an insulating oil inside the tank (see Figure 2, top). Passing alternating current through the windings of one coil creates a magnetic flux that induces current in an adjacent coil. The voltage adjustment is achieved through different numbers of coil turns.

Because the core is made of steel, a magnetostrictive material, these magnetic fluxes — which alternate direction — cause mechanical strains. This generates vibrations from the quick growing and shrinking of the metal. These vibrations travel to the tank walls through the oil and the clamping points that hold the inner core in place, creating an audible hum known as core noise (see Figure 2, bottom).

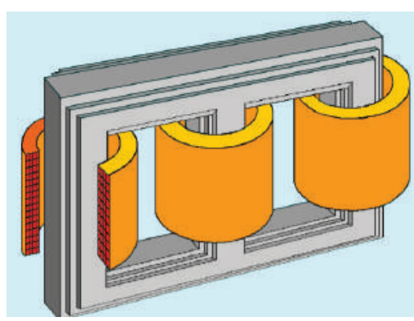
In addition to the core noise, the alternating current in the coil produces Lorentz forces in the individual windings, causing vibrations known as load noise that add to the mechanical energy transferred to the tank.

With these multiple sources of noise and the interconnected electromagnetic, acoustic, and mechanical factors at play, engineers at the ABB Corporate Research Center (ABB CRC) in Västerås, Sweden needed to understand the inner workings of their transformers in order to optimize their designs for minimal transformer hum.

⇨ COUPLING ACOUSTIC, MECHANICAL, AND ELECTROMAGNETIC EFFECTS ALL IN ONE

"We chose to work with COMSOL Multiphysics because it allows us to easily couple a number of different physics," said Mustafa Kavasoglu, scientist at ABB CRC. "Since this project required us to model electromagnetics, acoustics, and mechanics, COMSOL® software was the best option out there to solve for these three physics in one single environment."

Kavasoglu; Dr. Anders Daneryd, principal scientist; and Dr. Romain Haettel, principal engineer, form the ABB CRC team working with transformer acoustics. Their objective was to create a series of simulations and computational apps to calculate



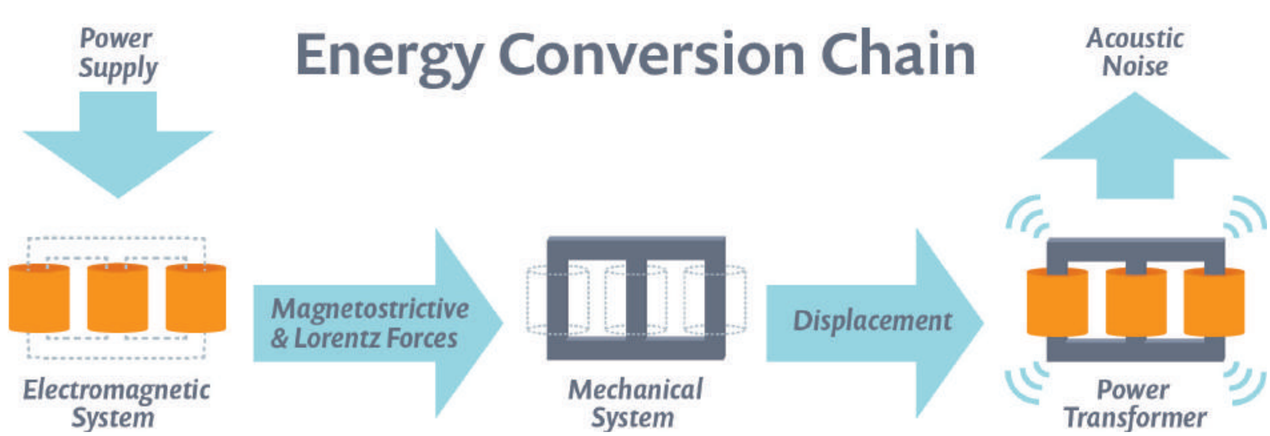


FIGURE 2. Top left: CAD model of the active part of a three-phase transformer with windings mounted around the core. Top right: The active part of a power transformer that is placed in a tank filled with oil. Bottom: The energy conversion chain for core noise and load noise generation (magnetostriction in the core and Lorentz forces in windings).

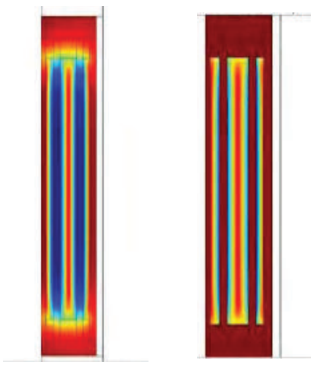


FIGURE 3. Simulation results showing the magnetic flux density (left) and Lorentz forces (right) in the transformer coil windings.

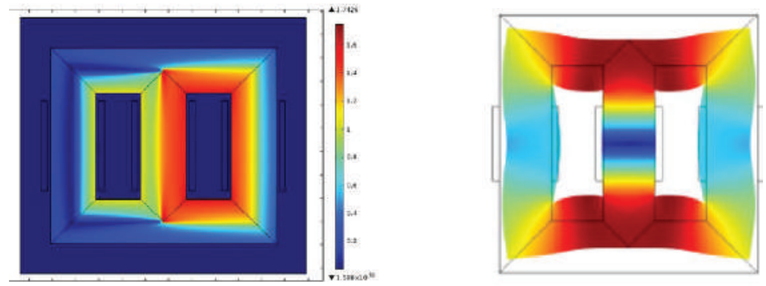


FIGURE 4. Left: COMSOL® software results showing levels of magnetic flux in the steel. Right: Results showing the resonance of the core. Deformations are exaggerated for visibility.

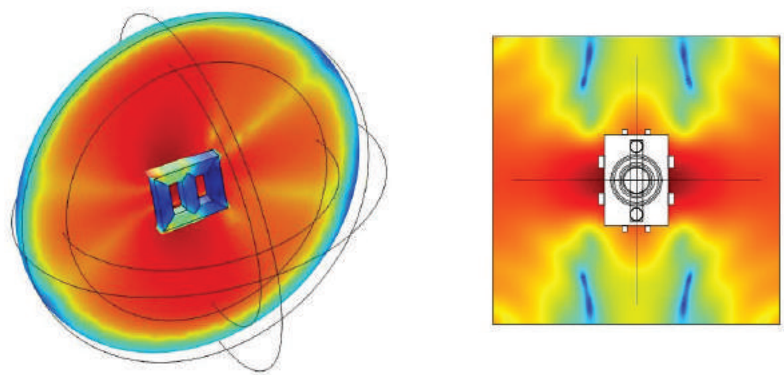


FIGURE 5. Results of the acoustic analysis showing the sound pressure field around the core (left) and around the transformer (right).

magnetic flux generated in the transformer core and windings (see Figure 3, left), Lorentz forces in the windings (see Figure 3, right), mechanical displacements caused by the magnetostrictive strains, and the resulting pressure levels of acoustic waves propagating through the tank.

They work closely with the Business Unit ABB Transformers, often relying on the experience and expertise of Dr. Christoph Ploetner, a recognized professional in the field of power transformers, to ensure that they satisfy business needs and requirements.

One simulation models the noise emanating from the core due to magnetostriction. The team began with an electromagnetic model to predict the magnetic fields induced by the alternating current, and then the magnetostrictive strains in the steel.

Their geometry setup included the steel core, windings, and an outer domain representing the tank. "We obtained the displacement from the magnetostrictive strains, then calculated the resonance for different frequencies using a modal analysis," said Kavasoglu (see Figure 4). "Resonances are easily excited by the magnetostrictive strains and cause high vibration amplification at these frequencies."

They were then able to predict the sound waves moving through the oil and calculate the resulting vibrations of the tank, implying sound radiation into the surrounding environment (see Figure 5).

They also simulated the displacements of the coil windings that cause load noise and determined the surface pressure on the tank walls due to the resulting sound field (see Figure 6).

> "We've also been using the COMSOL Server™ license to distribute our app to other offices for testing, which makes it easy to share it. This worldwide license is great; with a global organization, we expect users in our other locations around the world to benefit from these apps."

Including parametric studies that illustrated the complex relationships between design parameters (such as tank thickness and material properties) and the resulting transformer hum made it possible to adjust the geometry and setup of the core, windings, and tank to minimize the noise.

⇨ SPREADING SIMULATION CAPABILITIES THROUGHOUT ABB

The CRC team continues to use the COMSOL software to not only improve their understanding and their models, but to extend their knowledge to the rest of ABB's designers and to the business unit. Using the Application Builder in COMSOL Multiphysics, they have begun creating apps from their multiphysics models, which can be easily customized to suit the needs of each department.

These simulation applications simplify testing and verification for the designers

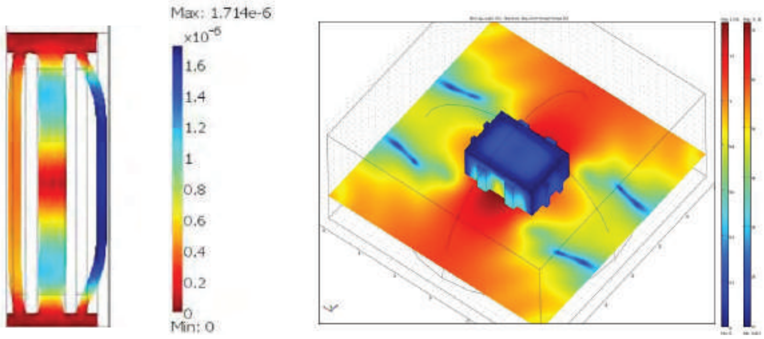


FIGURE 6. Left: Simulation results showing the displacement of the windings. Deformations are exaggerated for visibility. Right: Results showing the sound pressure levels outside the tank and the displacement of the walls.

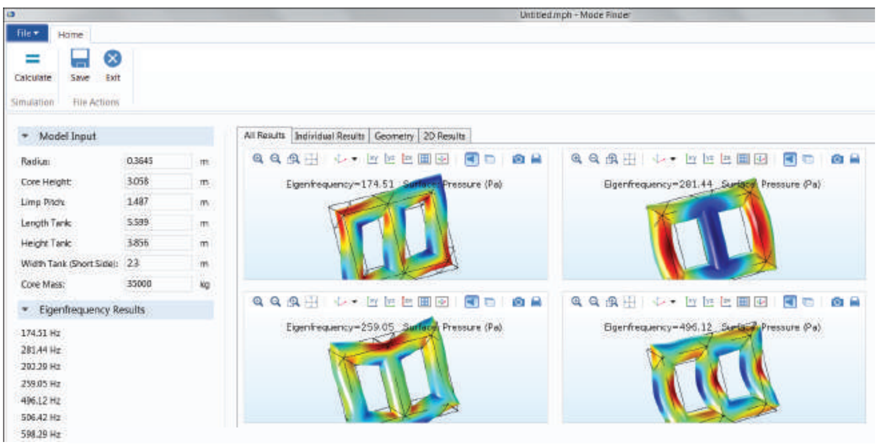


FIGURE 7. Cropped screenshot of the first simulation app created for calculating eigenfrequencies of the transformer core. At left, a tab in the app shows the model inputs; at right, results are shown for the calculated eigenfrequencies. Deformations are exaggerated for visibility.

and R&D engineers: "The designers have been using tools based on statistics and empirical models. We are filling the gaps by deploying simulation apps. The Application Builder allowed us to give them access to finite element analysis through a user interface without them needing to learn finite element theory," Haettel explained.

One application (see Figure 7) calculates the specific eigenfrequencies of the transformer core that can imply noise-related issues due to frequencies that fall within the audible range. This app includes both the physics model developed in the COMSOL® software and custom methods written in Java® code, programmed within the Application Builder.

"Our designers use standard spreadsheets that work well for the transformers they build frequently. But when new designs or different dimensions are introduced, they may run into problems with this approach, like error outputs showing less accurate data for noise levels. This can become quite costly if additional measures to reduce noise are required on the completed transformer," Haettel continued.

"Besides the cost aspect, there is the time aspect. The new app will make the designers' job easier and more efficient by using the precision of an FEA code."

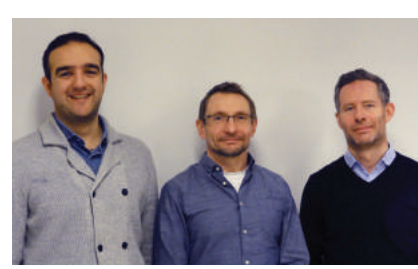
The custom application adds a level of convenience by letting users check how certain combinations of geometry, material properties, and other design parameters will affect the resulting transformer hum. "We've been deliberate about selecting which parameters we provide access to — focusing on the ones that are most important," Kavasoglu added.

With the wide range of industrial applications for which ABB designs transformers, this flexibility is immensely helpful for their design and virtual testing process. "ABB produces transformers for every industrial need. At the moment we're focusing on AC large power transformers commonly used by power companies that transmit and distribute electricity throughout cities," he explained.

"But the work we're doing can be translated to any type of transformer, and of course if we receive a specific request, we adapt the app to that need. This allows us to easily do additional development work. The Application Builder has made the transfer of knowledge and technology much easier.

"We've also been using the COMSOL Server™ license to distribute our app to other offices for testing, which makes it easy to share it. This worldwide license is great; with a global organization, we expect users in our other locations around the world to benefit from these apps." With a local installation of COMSOL Server, simulation specialists can manage and deploy their apps, making them accessible through a client or web browser.

The team is focusing on a second application that will calculate load noise. Once deployed to the business unit, this application will further remove the burden of tedious calculations, allowing designers and sales engineers to run more virtual tests without needing to work with a detailed model, and enable ABB to more quickly and easily produce the world's best transformers.



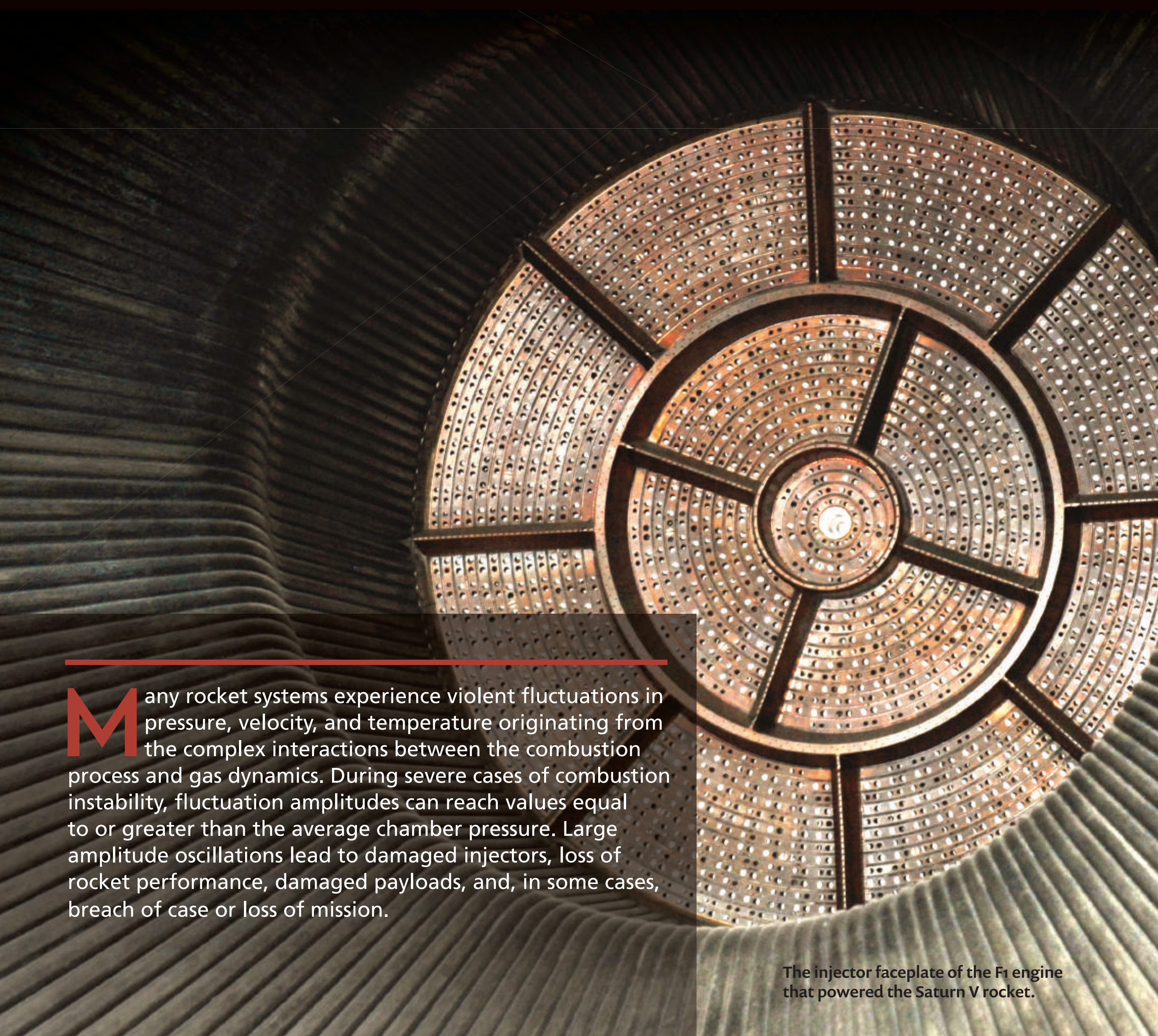
Left to Right: Mustafa Kavasoglu, Romain Haettel, and Anders Daneryd of ABB CRC.

NASA, AL, USA

MULTIPHYSICS SOFTWARE MODELS MEAN FLOW-AUGMENTED ACOUSTICS IN ROCKET SYSTEMS

Combustion instability in solid rocket motors and liquid engines is a complication that continues to challenge designers and engineers. The adoption of a higher-fidelity modeling approach supported by multiphysics analysis provides greater insight and predictive ability.

by SEAN R. FISCHBACH



Many rocket systems experience violent fluctuations in pressure, velocity, and temperature originating from the complex interactions between the combustion process and gas dynamics. During severe cases of combustion instability, fluctuation amplitudes can reach values equal to or greater than the average chamber pressure. Large amplitude oscillations lead to damaged injectors, loss of rocket performance, damaged payloads, and, in some cases, breach of case or loss of mission.

The injector faceplate of the F1 engine that powered the Saturn V rocket.

Historic difficulties in modeling and predicting combustion instability have reduced most instances of rocket systems experiencing instability to a costly fix through testing (see Figure 1), or to scrapping of the system entirely.

"A more complete depiction of combustion instability oscillations is achieved when a global energy-based assessment is used."

During the early development of rocket propulsion technology scientists and engineers were cued to the underlying physics at play through the measurement of vibrating test stands, observation of fluctuating exhaust plumes, and, most notably, the audible tones accompanying instabilities. These observations lead the pioneers of combustion instability research to focus their modeling efforts on the acoustic waves inside combustion chambers.

This focus on acoustics is quite logical given that the measured frequency of oscillation often closely matches the normal acoustic modes of the combustion chamber. But this narrow focus misses contributions made by rotational and thermal waves that are a direct result of, or closely coupled with, the acoustic wave. A more complete depiction of combustion instability oscillations is achieved when a global energy-based assessment is used.

Recent advances in energy-based modeling of combustion instabilities require an accurate determination of acoustic frequencies and mode shapes. Of particular interest are the acoustic mean flow interactions within the converging section of a rocket nozzle, where gradients of pressure, density, and velocity become large. The expulsion of unsteady energy through the nozzle of a rocket is identified as the predominate source of acoustic damping for most rocket systems.

Recently, an approach to address nozzle damping with mean flow effects was implemented by French[2]. This new approach extends the work originated by Sigman and Zinn[3] by solving the acoustic velocity potential equation (AVPE) formulated by perturbing the Euler equations[4].

Determining eigenvalues of the AVPE, where ψ is the complex acoustic potential, λ the complex eigenvalues, c the speed of sound, and M the Mach vector,

$$\nabla^2\psi - \left(\frac{\lambda}{c}\right)^2\psi - \boldsymbol{M}\cdot[\boldsymbol{M}\cdot\nabla(\nabla\psi)] - 2\left(\frac{\lambda\boldsymbol{M}}{c} + \boldsymbol{M}\cdot\nabla\boldsymbol{M}\right)\cdot\nabla\psi - 2\lambda\psi\left[\boldsymbol{M}\cdot\nabla\left(\frac{1}{c}\right)\right] = 0$$

is considerably more complex than the traditionally used pressure-based wave equation,

$$\nabla\cdot\left(-\frac{1}{\rho}\nabla p\right) + \frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} = 0$$

and requires numerical approximations of the chamber flow field and eigenvalues.

⇨ MODELING CHAMBER GAS DYNAMICS

The latest theoretical models for oscillatory disturbances in high-speed flows require a precise determination of the chamber acoustic eigenmodes. But first, a simulation of the mean flow properties of the combustion chamber must be performed.

COMSOL Multiphysics® software provides a numerical platform for conveniently and accurately simulating both the chamber gas dynamics and internal acoustics. This finite element software package provides many predefined physics along with a generalized mathematics interface.

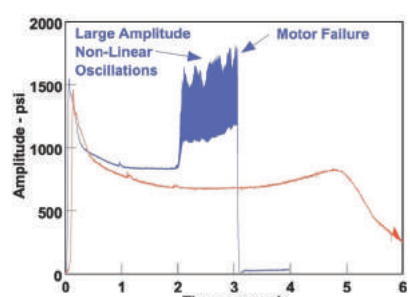


FIGURE 1. Pressure trace of a stable (red) and unstable (blue) solid rocket motor[1].

The present study employs the COMSOL finite element framework to model the steady flow-field parameters of a generic liquid engine using the High Mach Number Laminar Flow physics interface, which makes use of the fully compressible Navier-Stokes equations for an ideal gas together with conservation of energy and mass equations.

In order to account for the injection of hot gas due to the burning propellant, the injector face plate is modeled with a uniform inward flow of combusted propellant gas (see Figure 2). All other solid boundaries are modeled with the slip boundary condition, and the exit plane is modeled with the hybrid outflow condition, which means that both subsonic and supersonic flows are supported.

Results from the mean flow analysis are reviewed to ensure a valid and converged solution. Mean flow parameters such as pressure, density, velocity, and speed of sound are needed to model the AVPE. The values of the mean flow in the converging section of the nozzle, near the sonic choke plane, are of considerable interest. The sonic plane, where the Mach number is equal to 1, creates an acoustic barrier in the flow. In order to create an accurate geometry for the acoustic analysis, the sonic plane (pictured in magenta in Figure 3) is extracted from the mean flow analysis.

⇨ MODELING CHAMBER ACOUSTICS

The Coefficient Form PDE (Partial Differential Equation) mathematics interface of COMSOL Multiphysics is used to determine the complex eigenvalues of the AVPE. Mean flow terms in the AVPE are supplied by the solution from the mean flow analysis. Gas dynamics within the combustion chamber play a key role in defining the boundary conditions for the acoustic analysis. Within the converging and diverging section of the rocket nozzle, gradients of chamber pressure, velocity, and density grow theoretically infinite at the sonic plane where the Mach number is equal to 1. Downstream of the sonic plane, acoustic disturbances are convected with the mean flow at speeds greater than the speed of sound.

Originally published in the December 2015 edition of NASA Tech Briefs magazine.

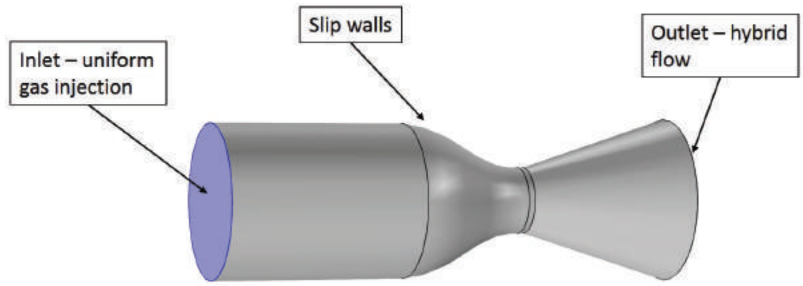


FIGURE 2. Simulated liquid engine geometry with boundary conditions.

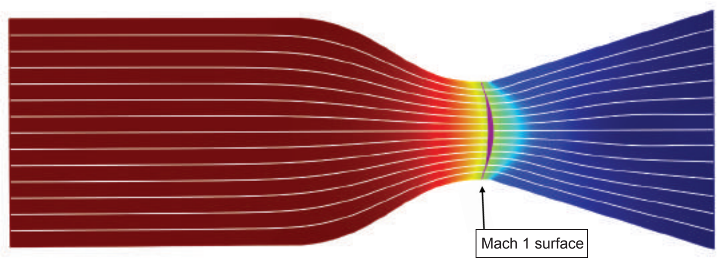


FIGURE 3. Velocity streamlines plotted over chamber pressure. The Mach 1 surface is plotted in magenta.

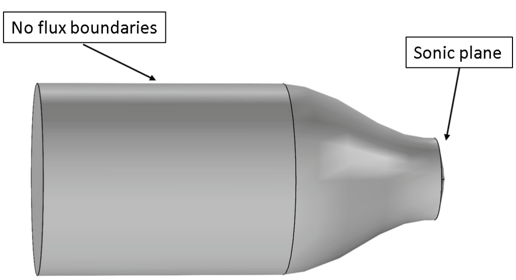


FIGURE 4. Acoustic analysis geometry with boundary conditions.

This condition prevents disturbances downstream of the sonic plane from propagating back upstream. The diverging section of the nozzle is acoustically silent and does not affect the chamber acoustics. The simulation geometry is truncated at the nozzle sonic line, where a zero flux boundary condition is self-satisfying (see Figure 4). The remaining boundaries are modeled with a zero flux boundary condition, assuming zero acoustic absorption on all surfaces.

The eigenvalue analysis produces complex eigenmodes and eigenvalues representing each acoustic mode and its complex conjugate. The real part of the complex eigenvalue represents the temporal damping of the acoustic mode, with the imaginary part defining the frequency of oscillation. The complex eigenvectors represent the spatial amplitude and phasing of the acoustic wave.

Comparing the acoustic mode shapes derived using the classic homogeneous wave equation (Helmholtz equation) to those derived using the AVPE demonstrates the benefits of higher-fidelity models that correctly represent the underlying physics (see Figure 5). Inclusion of mean flow terms in the AVPE accurately models the phase shift caused by the steady gas flow. Phasing is extremely important since combustion instability models make use of temporal and spatial integration of the acoustic eigenvectors.

Utilizing COMSOL Multiphysics to simulate the rocket gas dynamics and acoustic eigenmodes provides a more accurate mode shape over previous techniques. The higher-fidelity acoustic representation is easily incorporated into combustion instability models to give rocket designers and engineers greater predictive capabilities. The inclusion of damping devices, such as baffles, or changes in operating conditions, can now be more accurately modeled before testing.

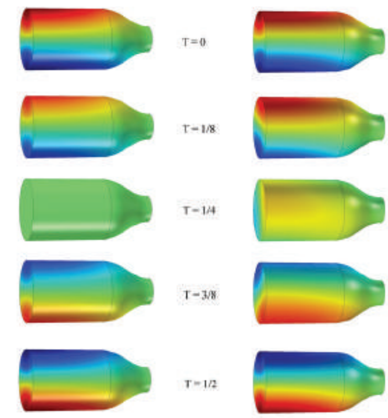


FIGURE 5. Comparison of the first tangential eigenmode calculated using the classic homogeneous wave equation (left), and the AVPE (right) of a half period (T) of oscillation.

⇨ CONTINUED WORK

A more complete depiction of combustion instability includes rotational oscillations and thermal oscillations in conjunction with chamber acoustics. Rotational oscillations occur as a direct result of the acoustic oscillation, where thermal waves can also be present in the absence of acoustic fluctuation. Continued work using COMSOL Multiphysics will focus on solving the viscous rotational wave that accompanies all acoustic oscillations. ❖

This article was written by Sean R. Fischbach, Marshall Space Flight Center/ Jacobs ESSSA Group, MSFC, Huntsville, AL.

REFERENCES

1. F. S. Bloomshield, *Lessons Learned in Solid Rocket Combustion Instability*, 43rd AIAA Joint Propulsion Conference, AIAA-2007-5803, Cincinnati, OH, July 2007.
2. J. C. French, *Nozzle Acoustic Dynamics and Stability Modeling*, Vol. 27, Journal of Propulsion and Power, 2011.
3. R. K. Sigman and B. T. Zinn, *A Finite Element Approach for Predicting Nozzle Admittances*, Vol. 88, Journal of Sound and Vibration, 1983, pp. 117-131.
4. L. M. B. C. Campos, *On 36 Forms of the Acoustic Wave Equation in Potential Flows and Inhomogeneous Media*, Vol. 60, Applied Mechanics Reviews, 2007, pp. 149-171.

Giving Furniture Testing a Leg Up

In the furniture world, quality standards rule. Chairs that pass a physical test are permitted to be sold on shelves, while a failed prototype means it's back to the drawing board for the engineers who designed the chair.

by LEXI CARVER

FIGURE 1. Setup of the chair test. Overlaid, geometry of the Continuum Blue chair model.

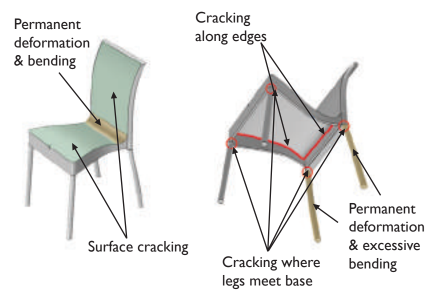


FIGURE 2. Schematic showing crucial points in the chair where failure typically occurs.

Behind every consumer product that hits the market is an iterative process of designing, redesigning, prototyping, and testing that points to the work of researchers, engineers, and specialists ensuring that an object is ready for retail. To meet requirements for safety and quality, manufacturers must pass an assessment to verify that their product can be sold in stores.

For a chair to be allowed on the shelves, this rigorous process involves confirming that it will hold a person's weight, retain its shape, and not buckle under certain conditions. A chair must survive thousands of cycles of repeated loading over a lifetime and cannot suffer cracking, breaking, or bowing.

⇨ TEST HOUSES LIGHTEN THE LOAD FOR DESIGNERS

Because this testing process can be costly and time consuming for clients needing certification, an independent assessment organization (a "test house") is striving to lessen the burden on companies involved in chair production. They test hundreds of different chairs every year, and each design that fails to meet the European and American quality standards (EN, BS, ISO, and ANSI standards) results in disappointment for chair producers and thousands of dollars in design changes and re-testing.

To reduce the burden on manufacturers, a test house turned to Continuum Blue, a COMSOL Certified Consultant helping companies develop numerical simulation apps, to create a tool the test house could offer clients for testing different chairs. This would allow them to virtually predict whether a chair design would pass or fail prior to the physical testing stage.

"Our customer wanted to explore whether they could develop simulation applications for their test services," explained Dr. Mark Yeoman, director of Continuum Blue. "Their clients would be able to log in remotely, upload various chair designs, and virtually test each one. Only once they were confident that a chair passed the virtual test would they manufacture the design and send a sample to the test house."

Such a test app would require a diverse range of functionality. It would have to incorporate the quality criteria, be intuitive and usable for engineers and designers at all stages of the development process, and be flexible enough to test different chair models, shapes, and materials.

⇨ SIMULATION PAVES THE WAY TO QUICKER FURNITURE ASSESSMENT

Yeoman developed a numerical model using COMSOL Multiphysics® software that would predict a chair's response to a standard loading test. His simulation included the chair geometry, floor, blocks to hold the legs in place, and loading plates for the back and base to mimic the physical test (see Figure 1).

Ultimately, the question the simulation needed to answer for many different designs was, "will this chair fail under the required load?"

"A chair has to hold up to scrutiny in quite a few areas," Yeoman remarked. "It must survive a lifetime without breaking, endure continued use with no cracking, and the legs should not bend, splay apart, or bow too much. If cracking occurs in the corners where the legs meet the base, the chair will fail."

⇨ MEASURING UP TO THE TESTING CRITERIA

"The first part of the test is a Static Load Test to see if the chair can hold a certain maximum load," Yeoman explained. "The test loads the chair over a 20-second period, initially loading the base and then the back, to mimic a large person sitting down and then leaning back. The second part involves a fatigue evaluation, where the chair is repeatedly loaded and unloaded over thousands of cycles. This is equivalent to the chair being used over many years."

Yeoman's model analyzed the deformation during a person sitting down, the mechanical stress, and the earliest and likeliest points of failure (see Figure 2). The model also assessed the contact pressure between the chair, loading plates, and floor, and predicted number of cycles to failure (see Figure 3).

"Setting up most of the model was straightforward," he added. "The structural mechanics, material properties, and load parameters were relatively quick to implement in COMSOL® software." Many mechanical factors influence the pass or failure of a chair test, but the contact study was the tricky part, as Yeoman explained: "Contact analysis is inherently complex; it is highly dependent on the material properties of the contact surfaces, the coefficients of friction, and is highly nonlinear in nature.

"There are several points of contact that make this an inherently unstable contact problem: four points of contact between the chair legs and the floor, then points between the seat and the bottom loading plate, and the chair back and the back loading plate," he continued. "We use the material stiffness, yield stress, and coefficient of friction to help us accurately define the contact parameters."

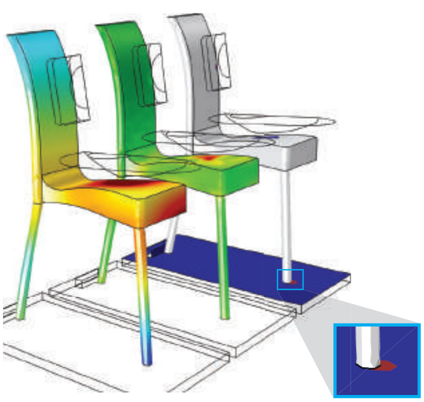


FIGURE 3. Simulation showing deformation (left), stress (center), and contact pressure (right) for a single chair during loading.

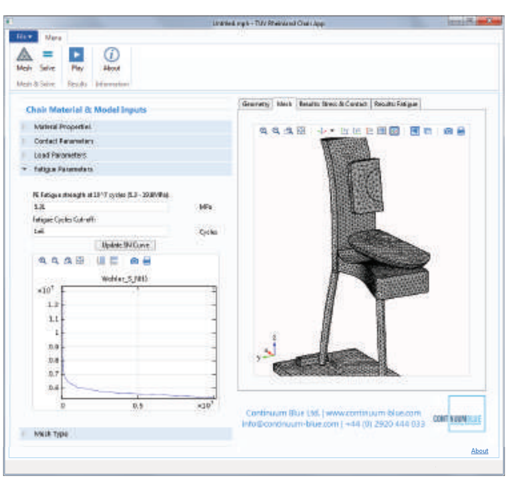


FIGURE 4. The Continuum Blue app showing mesh and input fields for a client to adjust according to their design needs.

⇨ MAKING VIRTUAL TESTING QUICK AND EASY

After completing and validating the COMSOL model, the Continuum Blue team went on to build the customized user interface, or application, that would allow a user to quickly run virtual tests on chair designs by changing certain inputs and parameters related to geometry, load conditions, and material properties.

"We used the Application Builder in COMSOL Multiphysics," Yeoman commented. The Application Builder allows the app designer to arrange entry fields, results tables, buttons, drop-down options, and graphics. The resulting app runs the full simulation, but the user — who may have no background in engineering or multiphysics modeling — has access to only the chosen inputs, not the model, physics, or analysis underneath.

"For the test application we built for our client, we wanted the user to be able to define various options, like nonlinear materials. We parameterized all the model features so that they were fully linked, and when someone changed one parameter the simulation would update accordingly."

"When the chair is loaded, stress levels are assessed against the yield and tensile strength of the material. If it goes into permanent plastic deformation, the relevant regions of the chair will be highlighted to let the user know that the chair is yielding. If the stress rises above the tensile strength of the material, then the chair has some form of catastrophic material failure, such as cracking and fracturing," Yeoman said. "The app will automatically show a pop-up window announcing that stress levels have gone above the material tensile strength and the chair has failed."

Mark Yeoman, Director at Continuum Blue.

This was exactly what the test house had been looking for, an application that allows chair manufacturers and designers to import their own chair geometry, select materials, define contact and loading conditions, and check the results against the test requirements to immediately see whether the design would pass or fail (see Figure 4).

Through a local installation of the COMSOL Server™ product, Yeoman was also able to distribute the app to different users, allowing them to log in to a database and launch specific apps. The test house, in turn, intends to use COMSOL Server in the future for sharing test apps with their own clients all over the world.

> "This will give them the ability to virtually assess dozens of designs before deciding on a single successful design to prototype and test."
>
> — MARK YEOMAN, DIRECTOR AT CONTINUUM BLUE

"These apps will make it much simpler for designers and test engineers to easily assess a chair's performance before production and physical testing," Yeoman said. "This will give them the ability to virtually assess dozens of designs before deciding on a single successful design to prototype and test, knowing with confidence that it will meet the test requirements of the quality standards, reducing the time and cost spent on building and testing physical prototypes." ❖

Increasing Productivity and Revenue with Computational Applications

With computational applications based on highly accurate multiphysics models, everyone wins. Both the app developer and its end users can benefit from innovation with reduced risk while minimizing production costs — or by introducing an entirely new revenue stream.

by **JENNIFER SEGUI**

HeatSinkSim is a cutting-edge computational tool, created by the simulation specialists at AltaSim Technologies, that until recently would not have been possible to develop in a realistic, competitive timeframe. The application's user-friendly interface extends multiphysics modeling and virtual prototyping capabilities to any engineer interested in optimizing their heat sink designs for power electronics.

Relevant to any product development process or design task, simulation applications can help resolve workflow bottlenecks in large companies and provide custom design capabilities to small companies and startups, where it is often impractical to keep dedicated simulation engineers on staff.

"Whether you're at a large company or a startup, if you're going to change the future and dominate the marketplace, you need to replace traditional design tools with something that is different, predictive, and represents the real world. Multiphysics modeling and simulation applications, such as HeatSinkSim, will enable and set the pace for engineering innovation," explains Jeff Crompton, cofounder and principal at AltaSim.

With professional insight and examples, this article will equip you with practical guidelines, and the inspiration to move forward with a simulation-driven product design workflow that reduces your development costs and time to market.

> "Multiphysics modeling and simulation applications, such as HeatSinkSim, will enable and set the pace for engineering innovation."
>
> — JEFF CROMPTON, PRINCIPAL AND COFOUNDER, ALTASIM TECHNOLOGIES

⇨ FROM DESIGN GOAL TO INNOVATIVE PRODUCT

The roadmap for integrating simulation into your product design workflow is straightforward with simulation applications, making expensive trial-and-error prototyping the primary design method of the past. To illustrate, let us consider common goals for heat sink design and optimization for power electronics, and the subsequent development and use of the HeatSinkSim app to meet and exceed those goals. Although you may not be working on heat sink design specifically, you can readily apply the methods and workflow described here to another product or process.

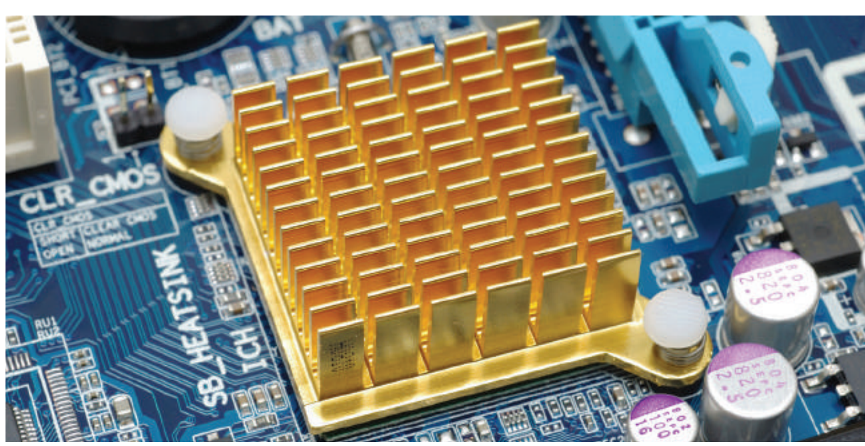


FIGURE 1. Vertical plate fin heat sink mounted on a PCB with power electronic components that drive critical continuous manufacturing operations.

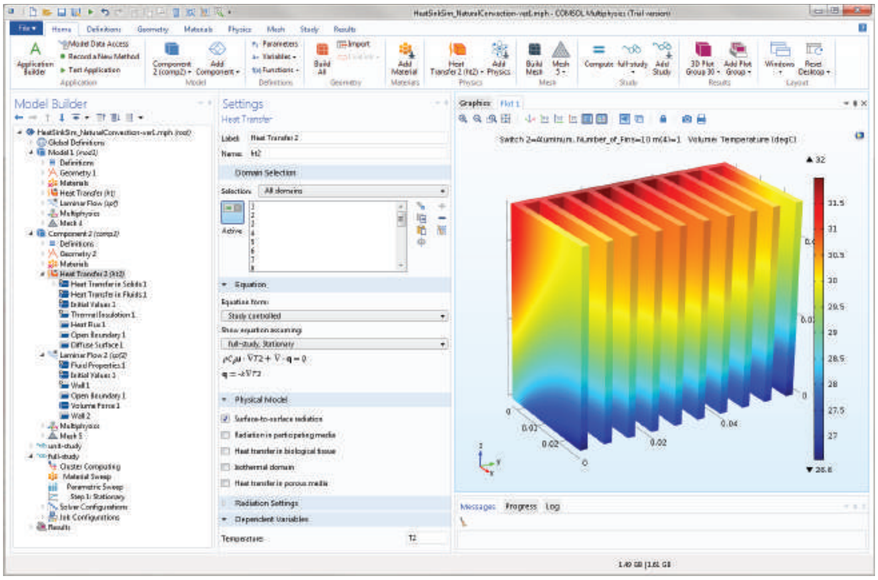


FIGURE 2. COMSOL Desktop showing the Model Builder through which you define your model geometry, materials, and physics. Then mesh your geometry, solve, and postprocess your results.

The overall objective in this example is the timely delivery of high quality products and services to market, through the reliable long-term operation of the power electronics that drive the automated manufacturing processes behind essentially all consumer goods. As higher production yields are required, increased performance demands of the electronics combined with their decreasing size results in higher power densities and operating temperatures, which can compromise extended operation.

To prevent higher operating temperatures from reducing the reliability of the control systems for manufacturing processes, the cooling of electronic components through passive heat dissipation and natural convection is necessary. Heat sinks are therefore integrated with the electronics on printed circuit boards (PCBs), as shown in Figure 1. The heat sink design shown in the figure uses an array of fins to increase the surface area available for heat to escape. The quantity, size, spacing, and thermal conductivity of the fins, for example, will affect the performance of the heat sink for a given applied power. Ultimately, there are many variables that need to be taken into account when optimizing the design of a heat sink to ensure the electronics remain well below their maximum operating temperature. But, what is the best way to optimize the heat sink design considering all of the variables involved?

Although physical prototyping has and continues to hold a necessary place in the product design cycle to ensure high quality, virtual prototyping via multiphysics simulation significantly cuts the time and expenses associated with the process by reducing the number of prototypes required and allowing assessment of viability early in the design cycle.

Using COMSOL Multiphysics® software, AltaSim developed and validated a multiphysics model of the heat sink shown in Figure 1, which serves as an indispensable tool for design optimization. You can get a good impression for the model setup process from Figure 2, which outlines the steps via the node order in the Model Builder window.

From the validated heat sink model, AltaSim developed a custom application using the Application Builder, a built-in feature in COMSOL Multiphysics shown in Figure 3. Their simulation app, HeatSinkSim, is shown in Figure 4. By using the Application Builder, they created an easy-to-use interface that allows any user to run complex engineering analyses, including those who are not simulation specialists. Central to the Application Builder are its Form Editor and Method Editor, which allow you to readily add Form objects to the app's user interface as well as

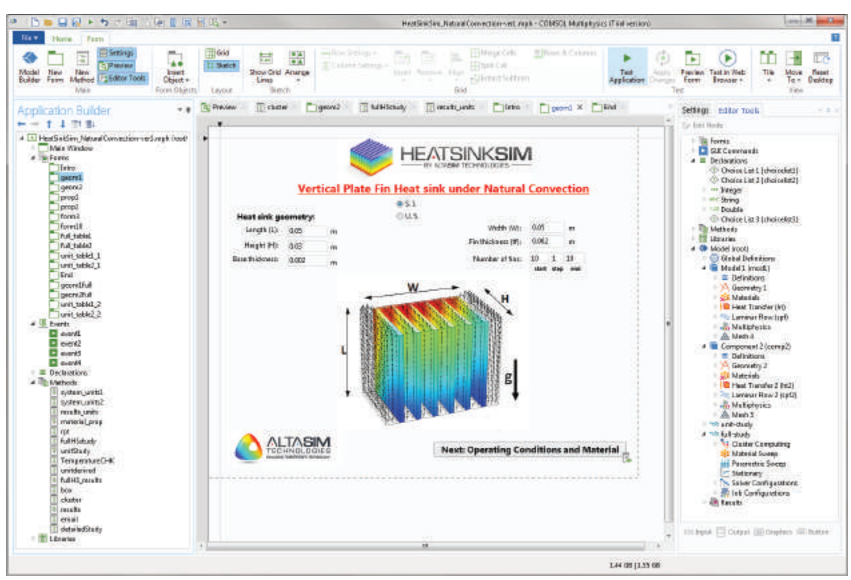


FIGURE 3. COMSOL Desktop® showing the Application Builder through which you design simulation applications based on your multiphysics models using the Form Editor and Method Editor.

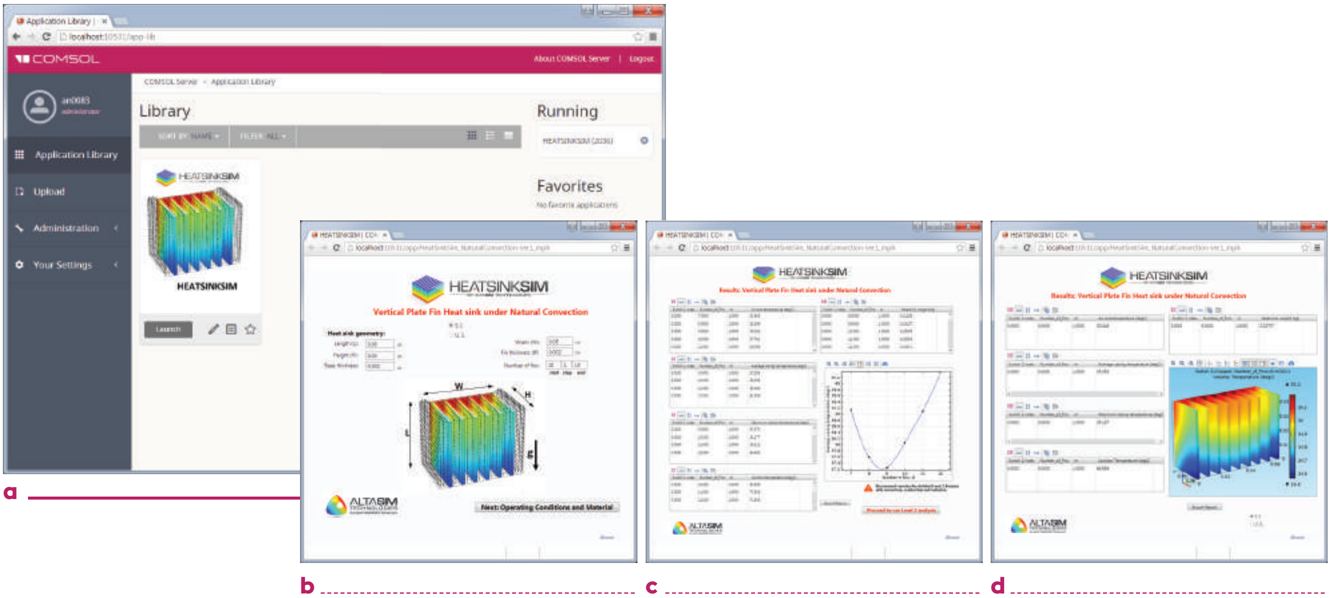


FIGURE 4. Log into COMSOL Server™ product using a browser. Then run HeatSinkSim from COMSOL Server in a separate browser tab or window (a). Through the simulation application in (b), you specify the geometry, materials, and operating conditions. The app offers two levels of analysis. Results from a Level 1 analysis (c) indicate that a Level 2 analysis (d) is recommended, which entails a full 3D conjugate heat transfer study.

customized functionality in Java® code methods that are run in response to user input.

Results from the HeatSinkSim app are shown in Figure 4, demonstrating the two different levels of analysis that are available. A *Level 1* heat transfer analysis solves the conjugate heat transfer problem and includes fluid flow with conduction, convection, and radiation. Results are shown in one-dimensional line plots of temperature vs. design parameters, such as the number of fins in the heat sink. A *Level 2* analysis performs a more detailed heat transfer simulation for a more accurate three-dimensional solution, but takes longer to run. If temperatures reach the user-defined operating limits during an initial Level 1 analysis, a Level 2 analysis is recommended to the user by HeatSinkSim.

In planning your simulation-driven workflow, it is important to consider logistics such as who will set up and validate the initial multiphysics model, design the simulation app based on that model, and ultimately perform design evaluation and optimization using the app. The ability to create an app based on a multiphysics model adds flexibility and makes planning your workflow easier since more people can run simulations through an app.

"If you recognize that you can develop simulation apps that are predictive and represent real-world situations with real dimensions and operational characteristics, then you can also look at the number of potential users that could benefit from simulation apps," says Crompton. "There are an estimated 80 million scientists and engineers around the world that are potential users of computational apps, compared to just 750 thousand that are currently using computer-aided engineering (CAE) tools."

Instead of running all of the analyses themselves, which can create a workflow bottleneck, simulation applications can help your simulation specialists make time to develop new tools and features that require their expertise. In case you do not have or intend to have simulation specialists on staff, you still have the option of outsourcing the model and app development to COMSOL Certified Consultants, such as AltaSim, or purchasing access to an existing simulation app — HeatSinkSim is just one example.

With simulation apps available to you and your colleagues, you can rely on highly accurate multiphysics models to predict the real-world performance of your product or process. By improving your understanding and reducing the time and cost associated with prototyping, you can get your innovative solution to market first.

Returning to the example of heat sink design optimization for power electronics, using HeatSinkSim offers a competitive advantage by improving access to simulation capabilities that help to ensure that safe operating

"There are an estimated 80 million scientists and engineers around the world that are potential users of computational apps, compared to just 750 thousand that are currently using computer-aided engineering (CAE) tools."

temperatures are maintained, and consequently, continuous manufacturing operations.

⇨ DISTRIBUTION AND REVENUE

Providing the means to access and run simulation applications is just as important as deciding who will be developing and using them to achieve your design goals. Through a local installation of the COMSOL Server™ product, your colleagues or customers can readily access simulation apps through a COMSOL client or web browser. Figure 4a shows a web browser with the Application Library for the COMSOL Server installation, through which you can launch the HeatSinkSim app. AltaSim uses the dashboard to manage app access and feature availability.

COMSOL Server is a product designed to run on anything from a standard laptop or desktop computer, to state-of-the-art supercomputing clusters. When choosing hardware that meets your specific needs, cost, availability, model complexity, and desired solution time are all factors to consider. If you would like results as fast as possible, but without the responsibility of managing hardware or the related security concerns, a supercomputing center may be an attractive option.

To provide access to HeatSinkSim, AltaSim partners with AweSim, an organization that aims to strengthen economic competitiveness by providing simulation-driven design capabilities to small and midsized manufacturers. Behind AweSim is the Ohio Supercomputing Center (OSC), which operates three major systems. HeatSinkSim runs on a node on the Oakley cluster, which is an HP Intel® Xeon® machine with more than 8300 cores for parallel computation. By submitting a request to AweSim to use HeatSinkSim, you are provided with an account at OSC, enabling you to log in remotely from a web browser on your own computer to run the app on their supercomputing clusters.

Through the AweSim distribution method, users can first try out HeatSinkSim, and then pay for the features and time that they use. By developing simulation apps, not only can you expand access to highly accurate multiphysics modeling capabilities, but pay-per-use access to apps can open up a new revenue stream for your organization as well.

⇨ A BETTER, FLEXIBLE PRODUCT DESIGN WORKFLOW

The HeatSinkSim app by AltaSim is not only an example of a solution to meet your design goals, but is also a product itself, developed and validated by experienced simulation specialists. By using the multiphysics modeling, application design, and distribution capabilities of COMSOL Multiphysics and COMSOL Server throughout the product development cycle, you can bring simulation to everyone, adding flexibility to your workflow that can improve quality and reduce risks and costs, while delivering your best possible product to market in a competitive timeframe. ❖

Executive Summary

HEATSINKSIM: BEHIND THE INTERFACE

To provide thermal design engineers with accurate tools, AltaSim Technologies is developing computational applications (apps) from detailed physics-based simulations. Apps include custom interfaces that allow the user to run multiphysics analyses by changing, for example, design parameters and operating conditions. HeatSinkSim is an app that examines the effect of heat sink design on thermal dissipation in power electronic components, potentially saving months of remediation measures later in the development process.

AltaSim is making the app available for general access and use with on-premise workstations or clusters as well as through secure connectivity to hosted parallel computing resources. Individual users can also perform further customization.

Benefits of app-based virtual prototyping

- Unified company approach
- Expert mathematical modeling knowledge accessible to designers and engineers
- Decisions based on predictive physics-based analysis
- Cost-effective

App management, deployment, and use

- Administrators can deploy, distribute, and manage apps with COMSOL Server™
- Users connect to COMSOL Server to access apps and run multiphysics analysis from a browser or COMSOL Client
- Apps are cluster computing enabled
- 24/7 worldwide access

Jeff Crompton, principal and cofounder, AltaSim Technologies

The AltaSim staff has over a century of collective experience in the development, application, and exploitation of multiphysics computational analysis and simulation. As founding members of the COMSOL Certified Consultants program, AltaSim combines its expertise in COMSOL Multiphysics® software and ability to extend its functionality with fundamental knowledge of physics, mechanics, computational science, and real-world processes.

Improving Gas Pipeline Squeeze-Off Standards with Numerical Simulation

In order to reduce the challenges associated with everyday gas pipeline maintenance, Gas Technology Institute (GTI) uses simulation-driven engineering to propose revisions to the ASTM squeeze-off standards.

by BRIANNE COSTA

Routine natural gas maintenance procedures often require digging into main roads, forcing drivers to navigate through a complex series of detours and backroads. But what if the process of repairing and replacing gas pipelines could be more efficient and less invasive? Gas Technology Institute (GTI), a leader in natural gas research, development, and training, aimed to investigate the industry standards for squeeze-off length in gas pipelines in an effort to make the pipes more accessible.

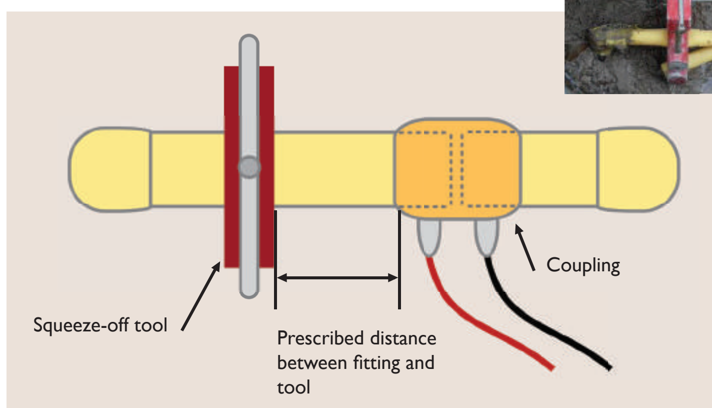


FIGURE 1. A polyethylene gas pipeline undergoing a squeeze-off procedure.

⇨ HOW CLOSE CAN YOU SQUEEZE OFF?

Illinois-based GTI is a research and development organization committed to the advancement of new energy and natural gas technologies. One sector of their research involves the investigation of squeeze-off distances for polyethylene (PE) gas pipelines. Squeeze-off is a seemingly simple procedure that involves compressing a pipe to completely stop the flow of gas (see Figure 1). This is commonly performed for maintenance tasks and to replace sections of pipe without shutting down the whole system.

After squeeze-off is completed, the pipe recovers much of its original shape, allowing gas flow to resume. There are natural benefits to polyethylene as a pipe material: its flexibility and ability to undergo large deformations; the absence of corrosion; its fusable and homogenous nature; and its resistance to harsh environmental conditions.

To avoid damage and malfunction, there are certain standards, such as the ones published by the American Society for Testing and Materials (ASTM), which must be met during the squeeze-off procedure. One particular standard involves the location of the squeeze-off relative to pipe fittings. The standard states that squeeze-off must be at a distance of either the length of three pipe diameters or twelve inches from the next pipe fitting, whichever is greater.

> “[We] trust the finite element method implementation in COMSOL for getting good results as proven by previous testing.”
>
> — OREN LEVER, PRINCIPAL ENGINEER, ENERGY DELIVERY & UTILIZATION, GAS TECHNOLOGY INSTITUTE

This rule was put in place to avoid strain and breakage in pipes. However, a number of gas utilities are interested in shortening the minimum required distance. The twelve-inch distance requirement is high for most pipelines (the majority of pipes used in housing and commercial applications are 2.375 inches in diameter or smaller). This means that although the three-diameter distance would be seven inches, the squeeze-off still has to be performed at the greater distance of twelve inches. The large length requirement for the small pipe diameters leads to more digging into the ground, rerouted roads, and more time and money spent.

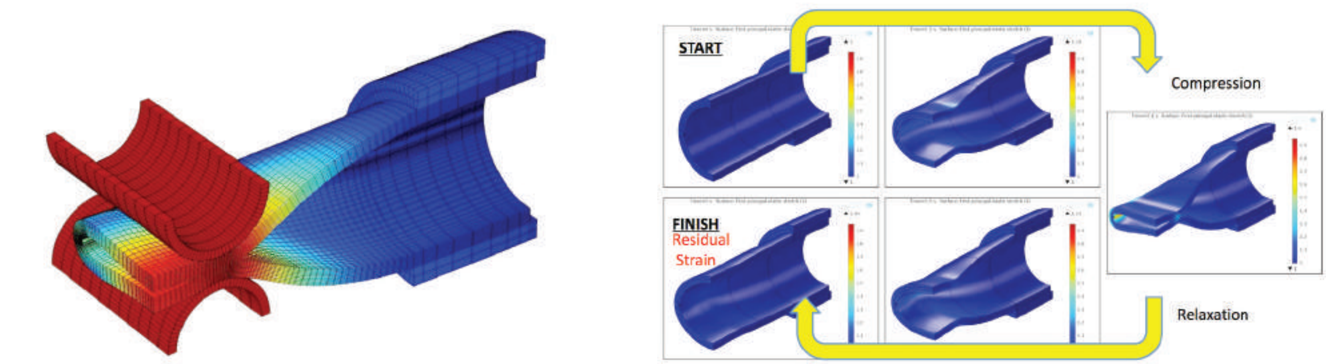


FIGURE 2. A simulation showing the total displacement (mm) of a pipe squeeze-off (left) and the full simulation of the squeeze-off process (right).

Funded by Operations Technology Development (OTD), a technology development partnership of natural gas distribution companies, GTI researchers Oren Lever and Ernest Lever took a closer look to see if the twelve-inch distance is really necessary for smaller pipes. Their goal was to see how close the squeeze-off of the pipe could get to a fused joint before it exceeded industry-accepted levels of strain and increases in stress concentration. To get answers, the team turned to the structural mechanics modeling capabilities of the COMSOL Multiphysics® software.

⇨ STRESS AND STRAIN ANALYSIS OF THE POLYETHYLENE PIPES

The GTI team set up a fully parametric time-dependent model using the Structural Mechanics Module and the Nonlinear Structural Materials Module in COMSOL. As Oren Lever of GTI says, their team "trusts the finite element method implementation in COMSOL® software for getting good results as proven by previous testing." They defined the mechanical and numerical properties of two sets of contacts to model the internal pipe-to-pipe and the external pipe-to-squeeze-off-mechanism structural contact, respectively.

They used the meshing capabilities of the COMSOL software to create a hybrid mesh by integrating structured and unstructured meshes. This approach, together with a custom constitutive model implemented in the software, allowed the team to accurately analyze the large deformations in the pipe. The simulation accounted for the different stages of the squeeze-off procedure shown in Figure 2: pressurization of the pipe, squeeze-off, hold, release, and relaxation.

Special attention had to be given to the meshing of the pipe under the squeeze bars to enable the simulation to analyze the very large deformations encountered when the pipe is fully squeezed off (see Figure 3). Thanks to the meshing capabilities in COMSOL and its parametric nature this particular meshing was easily scaled to different pipe sizes. While the stresses and strains in this region were not the focus of this project, they are of interest regarding the general effect of squeeze-off on the lifetime of the pipe. The COMSOL model will allow the GTI team to conduct further squeeze-off investigations.

⇨ HIGHLY NONLINEAR MATERIALS CALL FOR COLLABORATION

To capture the unique behavior of polyethylene, GTI needed a custom viscoelastic-plastic constitutive model. For this, they turned to Veryst Engineering, a COMSOL Certified Consultant, for help in implementing the chosen material model in COMSOL Multiphysics. To do this, explained Nagi Elabbasi from Veryst, they first selected the experimental material tests needed to calibrate the material law typically used for thermoplastics such as PE, then fit the material parameters to the stress-strain response of PE, and finally implement in COMSOL the set of ordinary differential equations (ODEs) needed for the custom material model.

The material tests have been carried out by GTI on medium-density polyethylene (MDPE) pipe materials and included tension and compression tests at different temperatures, strain rates, and strains, especially high strains, as well as loading and unloading tests (see Figure 4). The team at Veryst found actual values of material parameters that fit the experimental data. To calibrate, Veryst used MCalibration, an

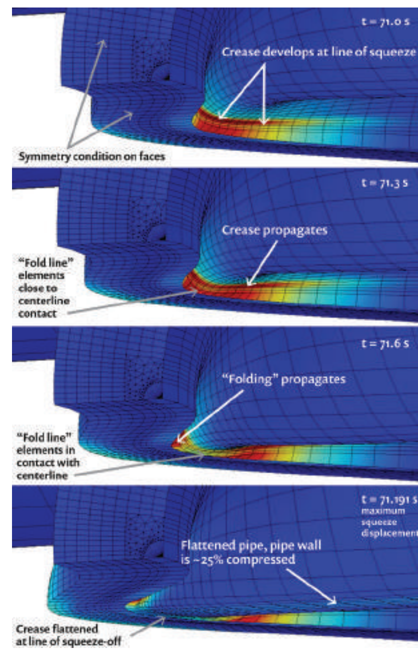
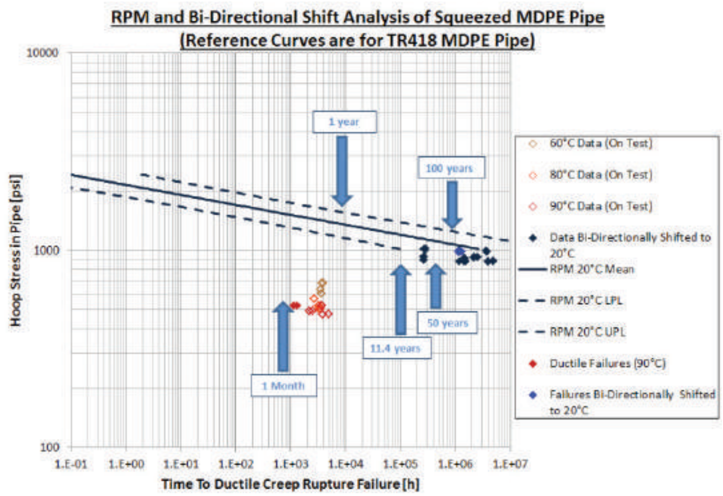


FIGURE 3. Deformation at the line of squeeze-off at different time steps, just before and at maximum squeeze displacement.



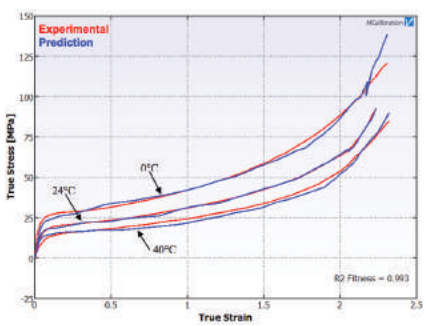


FIGURE 4. Example of the tensile response of PE and the model fit made by Veryst Engineering (bottom) and accelerated lifetime tests for the PE pipe squeeze-off performed by Gas Technology Institute (top).

optimization tool they developed to vary parameters until a very good fit of experimental data is discovered. To implement and verify the calibrated material law in COMSOL Multiphysics, Veryst relied on the software's flexibility by adding a system of ODEs representing additional states used to describe the custom constitutive model.

Another option, useful in those cases where a material model cannot be described by equations, would have been to use the External Material feature to access functions written in FORTRAN or C code and compile them into a shared library. In that case, the external material can be defined by the stress-strain relation, simply return an inelastic strain contribution to the material model in use, or link directly to a commercial external material library like PolyUMod® library from Veryst Engineering.

With simulation, GTI was able to determine that in the case of small-diameter pipes (smaller than 3.5”) the closer squeeze-off distance of three pipe diameters did not cause strains over current industry-accepted strain limits.

They used additional accelerated lifetime testing (shown in Figure 4) to validate these results. Squeeze-off was tested at two and four inches from the fused coupling at temperatures high enough to accelerate the creep rate in the polyethylene when compared to normal operation. Through this extra testing, the team at GTI discovered that the pipes would have an 80-year lifetime when operating at an average temperature of 20°C. This agrees with the standard industry lifetime of 80 years for MDPE gas pipelines.

⇨ EFFICIENCY IS KEY FOR FUTURE PLANS AT GTI

From their simulation work, GTI found that the squeeze-off location on small diameter pipes can be closer to the fitting than the current minimum distance allowed under the ASTM standard. This confirms that the current twelve-inch minimum distance requirement can be reevaluated for smaller pipe diameters, which are the most commonly used sizes for residential and commercial gas applications. Thanks to their research, GTI is helping set new guidelines that will make the current gas pipeline maintenance process less expensive, more efficient, and less invasive.

Ernest Lever, R&D director (left) and Oren Lever, principal engineer (right) at GTI.

Regarding plans for further research, Lever says their team plans to expand the constitutive model to include temperature and creep in order to describe relaxation effects more accurately. The goal is to be able to perform damage propagation and failure analysis to predict the behavior of the pipe for different load scenarios, such as the installation of reinforcement clamps.

GTI will also look into making their findings available to engineers not necessarily versed in numerical simulation. As COMSOL users they have access to the Application Builder, a tool that allows simulation specialists such as Lever to wrap a COMSOL model in a custom user-friendly interface. This means that engineers on the field and maintenance staff involved don’t have to second-guess their choices, especially in unusual operating scenarios. Meanwhile, simulation specialists save time and can focus on a new project. For now, GTI continues to devote their research and development efforts to natural gas and energy innovations. ❖

ADDRESSING OIL SPILL CLEANUP USING HYDROPHOBIC MESHES

Since current oil spill cleanup and containment methods are often costly and operate with only limited success, engineers at Amphos 21 have devised a numerical simulation app for testing new cleanup techniques using hydrophobic meshes.

by LEXI CARVER

Oil spills are notorious for being urgent and unexpected, known for the villainous damage they cause to aquatic environments and marine life, and must be contained swiftly before they cause long-term disaster. Techniques for containing and recovering spilled oil are readily available in the form of booms and skimmers that collect it, but these rarely fully rectify the problem.

Booms are used to curb the spread of oil into wider areas and keep the oil from reaching sensitive coastlines. Some designs absorb oil in an effort to remove it, while skimming techniques are also used to clear oil away. In other cases, controlled fires burn oil off of the water — though this creates another pollutant — or chemical dispersants are added to the water to accelerate the breakdown of the oil components.

These methods, while helpful, are not able to collect much of the oil during cleanup and are only effective if deployed very quickly to the site of the accident. Much of the oil sinks to the seafloor. For instance, cleanup efforts after the 1989 Exxon Valdez oil spill off the Alaskan coast were unable to recover most of the oil.

What is collected is often an oil-water mixture that is only partially usable. This means that in addition to the obvious environmental concerns, the wasted oil results in even more pumping after the cleanup attempts, in order to access the original amounts of oil.

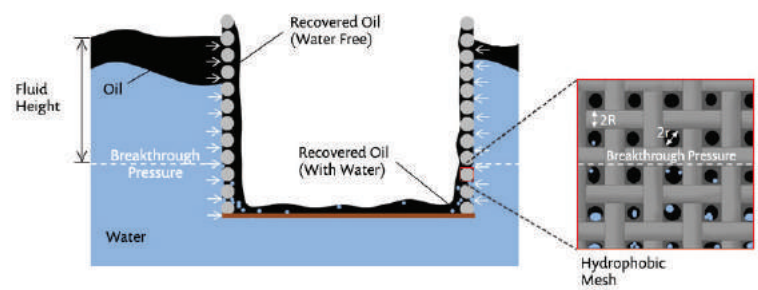


FIGURE 1. Concept of a hydrophobic mesh operation.

In response to the need for cleanup methods that prevent ecological disaster and oil waste, Amphos 21, a consultant group specializing in environmental technology, has developed numerical models of hydrophobic meshes, a concept for collecting spilled oil that is being studied by scientists and technologists. Their goal? An answer to this quandary that would be fast, easy to use, and environmentally friendly.

After conceptualizing the hydrophobic mesh as a porous medium, they developed computer simulations and custom applications to distribute to people in product development, disaster response, and environmental organizations. Through simulation apps, they intend to make virtual testing capabilities available for the engineers, researchers, and cleanup crews seeking the appropriate response for a given oil spill scenario as they race against the clock to stave off destruction.

⇨ POTENTIAL FOR NEW OIL RECOVERY METHOD

The meshes being studied at Amphos 21 are usually made of steel or copper and coated with a hydrophobic polymer to repel water and attract oil. They act like filters; water remains on one side, while oil passes through (see Figure 1). The rates of oil flux through the holes of the mesh depend on the water depth, oil properties (which can vary depending on where the oil is pumped), and the coating on the metal.

"In addition to being an effective option for clearing oil from the water, this offers the possibility of recycling

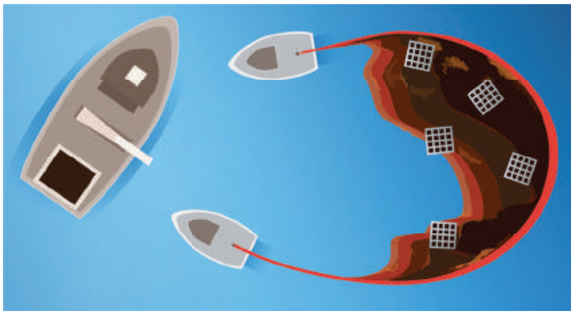


FIGURE 2. Concept: a boom towed by boats hems in the oil while hydrophobic meshes collect it.

the spilled oil without having to process it — which is often another costly step in the cleanup process," said Emilie Coene, consultant at Amphos 21. "These meshes could be used for continuous operation, are very clean, and potentially have good recovery efficiency and capacity."

Hydrophobic meshes could, hypothetically, be used alongside equipment designed for blocking the oil from spreading out. For instance, while a boom surrounded the spill, meshes and collection containers could be deployed directly into the oil to collect it (see Figure 2). The collected oil could then be pumped out of the container at regular intervals to ensure the correct pressure difference from one side of the mesh to the other.

"You can envision a large cylindrical container (the mesh) that could be left in the ocean until saturated with oil and then emptied," added Orlando Silva, senior consultant and project manager. "Or you could deploy the container and keep it connected to a pump for continuous extraction as oil is collected. They could also be tailored to many different operating conditions," he continued. "We're hoping to offer something to R&D engineers trying to design cleanup tools, environmental companies doing research, and the oil companies that have to figure out how to contain a very large problem very quickly."

One challenge arises with the hydrophobic mesh collection method: fluid height. At a certain water depth, hydrostatic pressure will reach a "breakthrough" level, causing water to intrude into the mesh and the collection container. If enough water gets mixed in with the salvaged oil, the oil needs to be treated and processed in order to be usable.

The retention properties of the mesh are a function of the oil-air and water-air surface tensions, and the contact angles of oil and water on the mesh surface. The coating on the mesh creates water repellency, but this is overcome at certain water depths. The flow of oil and water through the mesh depends on fluid properties such as viscosity, density, and surface tension — which vary for different types of oil — and the mesh porosity and permeability.

Given this, how could the team at Amphos 21 design a mesh for different water levels and oil types? Could different hydrophobic coatings work for different ocean depths?

The answers lay in finding the best combination of mesh properties for different oil spill conditions. The team set up numerical simulations to help them in their search to find the best designs to collect the most oil.

⇨ HOW DOES MESH DESIGN AFFECT OIL FLUX?

Seeking the solution to a problem no one has solved can sometimes feel like a needle-in-a-haystack hunt, even though you may have a general idea of where to start. But mathematical modeling techniques vastly simplify the process.

Coene, Silva, and Jorge Molinero, partner and modeling solutions director at Amphos 21, used COMSOL Multiphysics® software to set up a simulation of a hydrophobic mesh in order to analyze how different mesh designs would behave at different water depths and to assess factors influencing the mesh performance.

The success of a mesh is ultimately measured by the oil recovery rate and the purity of the recovered oil. The team's modeling work therefore involved testing different polymer coating properties and mesh operation at different ocean depths, and analyzing flow rates for different oils.

To see how different geometries would affect the oil flux, they also parameterized the mesh wire radius as well as the hole size and spacing. In COMSOL® software, they conceptualized the mesh as a porous medium coupled with two-phase flow to represent water

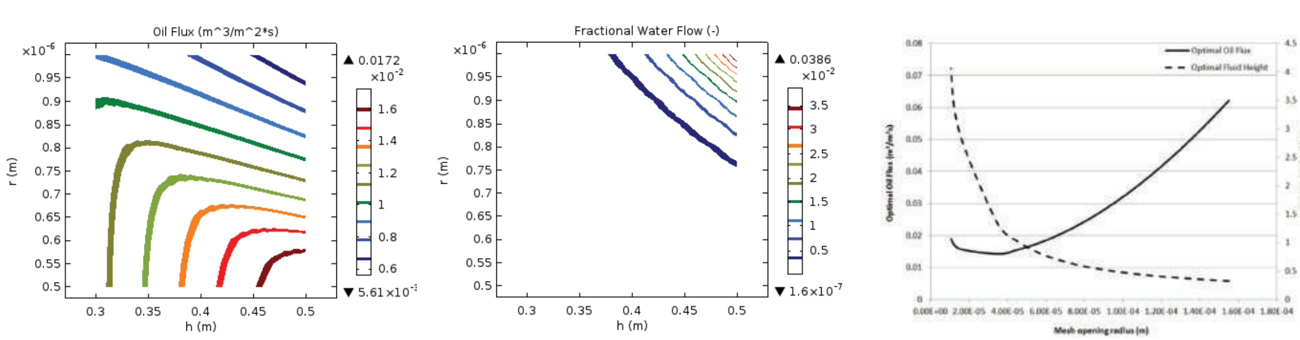


FIGURE 3. COMSOL® software results showing how oil flux through the mesh (left) and fractional water flow (center) vary with wire radius and water depth. Pore analysis showing the relationship between fluid height, pore radius, and oil flux (right).

and oil traveling through. From there, Coene, Molinero, and Silva were able to calculate the oil saturation level, the flux rates for the oil and water, and the fraction of water mixed with oil that passed through the mesh (see Figure 3, left and center). "These results are especially helpful for choosing a pore radius size when designing a mesh for use at certain depths," Molinero commented.

After studying the correlation between optimal oil flux and optimal fluid height, they also used their model to run a pore analysis (see Figure 3, right) based on a given maximum water fraction — the amount of water permitted in the mixture without requiring the oil to be treated.

"For different mesh opening sizes, the model results gave us the oil flux and the maximum fluid column height for the desired oil purity," continued Coene. "For example, imagine I want to operate the mesh at least half a meter below the water level, and I want a maximum of 1% water in my mixture. This shows me what radius I'd need for my mesh holes."

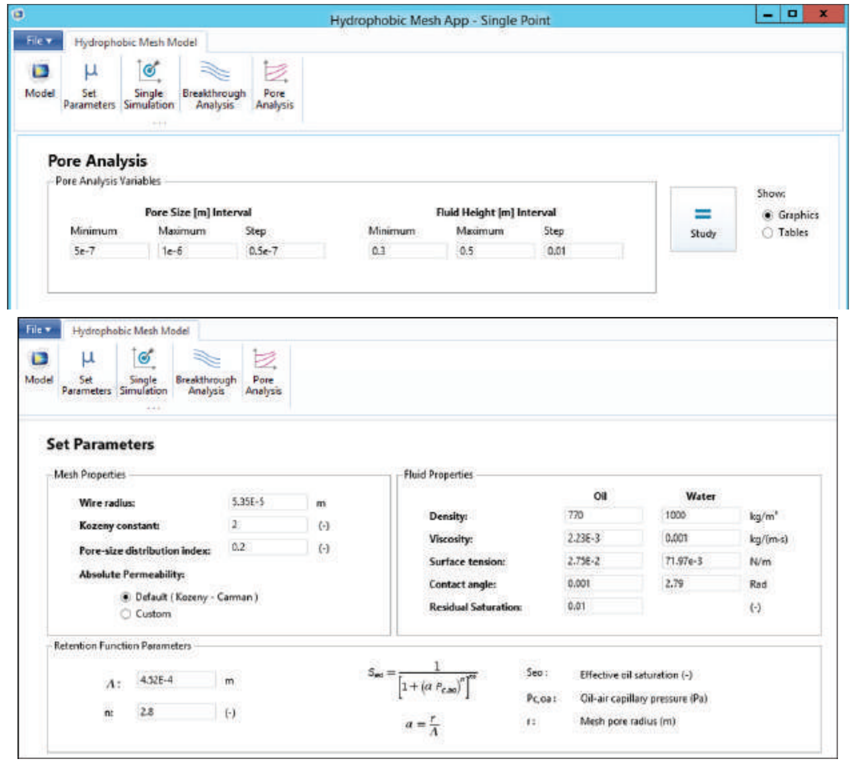


FIGURE 4. A cropped screenshot of the Amphos 21 hydrophobic mesh app showing inputs for a pore analysis (top) and mesh and fluid properties (bottom).

⇨ SIMULATION APPS OFFER DESIGN CAPABILITIES

One feature of the COMSOL software makes it possible to easily distribute the results of a simulation to others without sharing the entire model. "We used the Application Builder available in COMSOL Multiphysics to create a customized user interface around our model," Silva explained. "Simulation apps make it possible to distribute the simulation results to users but not give them the entire model. Companies can use these apps to reduce experiments, save money on testing, and they'll be able to simulate a small section of a mesh and then scale it in order to design the technology they need. This is a promising development in designing hydrophobic meshes."

The Amphos 21 app (see Figure 4) allows the user to predict the mesh performance and quickly check the quality of designs for different operating conditions by changing the mesh and fluid properties. In less than 30 seconds, he or she can run an analysis over a range of pore sizes and water depths to find the best mesh for certain depths based on a chosen maximum water fraction.

The app will also calculate different mesh properties, such as the absolute permeability and other quantities related to the breakthrough pressure, helping designers choose the ideal fit to suit the needs of different circumstances.

"We think this will be very appealing to R&D departments, environmental groups, and others working to solve this problem," said Molinero. "With the COMSOL model and a custom application built around it, we've made headway into a new way to clean up oil spills."

Next, they will distribute their app to engineers who can use the hydrophobic mesh model on a large scale to design the ideal mesh for a given situation, and ultimately deploy new tools to help in real time. ❖

> "With the COMSOL model and a custom application built around it, we've made headway into a new way to clean up oil spills."
>
> — JORGE MOLINERO, PARTNER AND MODELING SOLUTIONS DIRECTOR, AMPHOS 21

The Amphos 21 team working on their simulation app. Left to right: Orlando Silva, Emilie Coene, Jorge Molinero.

NGI, NORWAY

Shake, Rattle, and Roll

Norwegian researchers are tracking how low-frequency sound waves travel within buildings so that they can recommend design adjustments to alleviate annoying vibrations.

by **JENNIFER HAND**

Anyone who has slept near an airport will know the sensation — an early morning flight wakes you from sleep, not only because the engine is noisy but also because everything around you seems to be shaking. Likewise, people living near wind turbines, military sites, or hospitals with helicopter landing pads often complain that windows rattle and everyday objects buzz when there is external noise. More puzzling for them is the fact that even when they can discern no sound, they may still notice irritating vibrations.

If the response of the sound is 20 vibrations per second (20 Hz) or less, it is described as infrasound, meaning that the original sound is not usually audible to the human ear. The effects, however, are very easy to detect. As waves hit windows, spread to the floor, and affect internal walls, they induce a noticeable indoor vibration. Low-frequency sound waves are notorious for their potential to create annoying disturbances.

⇨ LOW-FREQUENCY SOUND WAVES IN BUILDINGS

Noise is part of modern life and there are formal standards that use sound pressure level measurements to recognize high-frequency sound waves at levels of sensitivity, intrusion, and danger for humans. According to Finn Løvholt of the Norwegian Geotechnical Institute (NGI), the generation of building vibration due to infrasound is an area of research that has not been explored extensively. For this reason, NGI, an international center for research and consulting within the geosciences, has been running investigative programs for several years on behalf of the Norwegian Defence Estate Agency.

"Low-frequency sound encounters less absorption as it travels through the air than higher-frequency sound, so it persists for longer distances. The amount of sound transmitted from the outside to the inside of buildings is greater. We are interested in what happens at the threshold of hearing," explains Løvholt. "We want to understand how sounds from external sources interact with buildings and generate vibration that is perceived by people. We can then recommend countermeasures to prevent vibration and may be able to propose standard units that recognize the need to account for the 'annoyance' factor."

> "We have never achieved this level of agreement with real-life testing before and it is all down to how we were able to model the different structural elements in COMSOL Multiphysics."
>
> — FINN LØVHOLT, NGI

⇨ SIMULATING THE SPREAD OF SOUND WAVES

Løvholt and his colleagues decided to create a computer model that would allow them to pick apart the mechanism of low-frequency sound waves hitting and penetrating a building. They used the COMSOL Multiphysics® software to simulate a wooden structure with two rooms separated by a wall (see Figure 1, top), closely mimicking the laboratory experiment setup. Within the model, they assigned a loudspeaker to one room, a microphone to the other, and placed various probes around the structure in order to monitor sound pressure levels and vibrations. Every component was

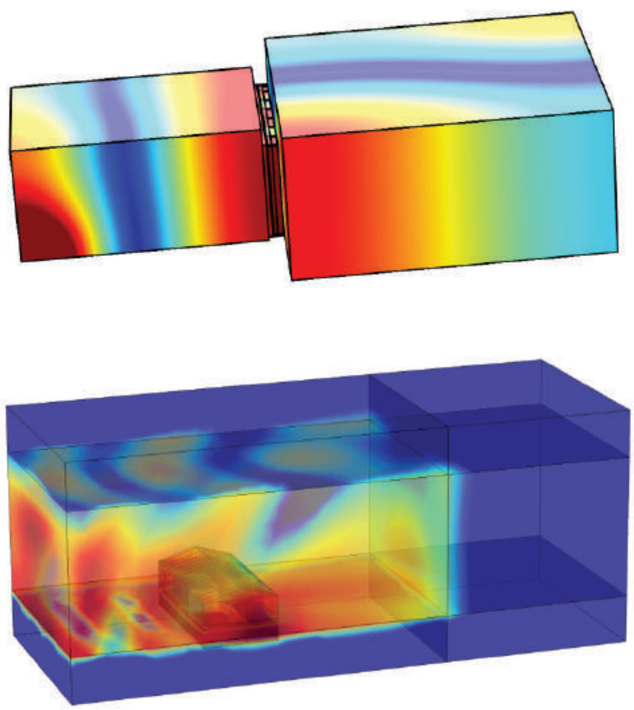


FIGURE 1. Top: Simulated sound pressure in a laboratory with two chambers divided by a wall. A loudspeaker is placed in the room on the left-hand side. The simulations show that the acoustic resonances within each room affect the sound insulation. Bottom: Simulated low-frequency sound originating from outside, around, and inside a building. In both cases, the colors indicate the variation in the sound pressure within the rooms and the wall cavities.

carefully modeled, including the steel frame, the air cavity and studs in the wall, the windows, the plywood sheet, and the plasterboard. "Each element has a resonance that depends on the wavelength of the sound wave and the pressure distribution. For example, there is high pressure in the speaker room and lower pressure in the microphone room, and the resonance of a wall will depend on its length, thickness, and stiffness," explains Løvholt.

The team also had to recognize compound resonances created when two components are joined, such as two pieces of timber that are screwed together. "The advantage of COMSOL Multiphysics is that it allows us to enter all the parameters we need to monitor. In particular, it enables us to couple physics, so we can, for example, look at the acoustics of open-air sound interacting with indoor structural dynamics. The coupling works both ways so we can identify feedback. This coupling is crucial for our analysis because sound waves can generate a huge range and variety of resonances. The model really allows us to see these."

The NGI team then verified their simulation with laboratory testing of low-frequency sounds as they were transmitted through a wooden construction with two rooms. Løvholt explains that the motion of the wall and the sound pressure level are the main quantities measured and results show very close correlation to the COMSOL Multiphysics model (see Figure 2). "The response of the real wall is very clear and the model mimics it almost perfectly. This is the most spectacular aspect."

The model shows that the transmission of sound within a building is governed by the way in which low-frequency waves interact with the fundamental modes of the building components, the dimensions of the room, and the way in which air leaks from the building envelope. Vibrations in ceilings and walls seem to be the dominant source of low-frequency indoor sound, with floor vibration driven by sound pressure inside the room.

⇨ CHEAPER AND QUICKER THAN PHYSICAL TESTING

"We now have a tool to predict sound and vibration at low frequencies," Løvholt says. "We can use it to design and test mitigation measures such as the lamination of windows and the stiffening of walls — if a wall or window moves less, sound transfers less. In addition, the model shows us the influence small details have on the system; for example, how the screw connection between studs and plasterboards can reduce the effect of a countermeasure, as they actually reduce the overall stiffness of the structure."

The next stage for the team is full-scale field tests on a real house in an area of Norway that is exposed to aircraft noise. Meanwhile, the team will continue to use and develop the model. "We have never achieved this level of agreement with real-life testing before and it is all down to how we were able to model the different structural elements in COMSOL Multiphysics," concludes Løvholt. "The model enables us to make decisions and assign countermeasures. This is much cheaper and quicker than physical testing. The model may then be expanded to simulate the sound propagation and vibration in an entire building" (see Figure 1, bottom). ❖

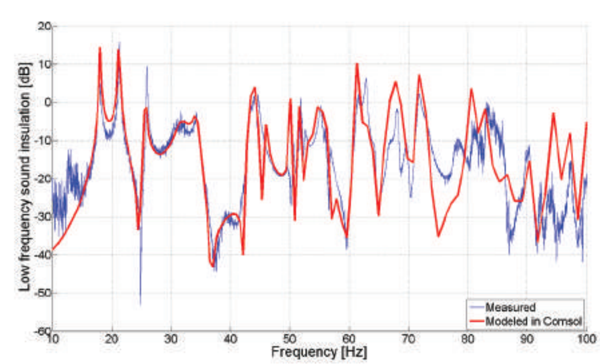


FIGURE 2. The model accurately captures the location of the resonances as well as the level within a few decibels. As the frequency increases, more modes in smaller and smaller structures will get excited. This shows as the increasing difference between the measurements and the model results.

Nuclear-Safety-Related SQA Procedure Automation with Custom Applications

Nuclear-safety-related procedures are rigorous for good reason. Small design mistakes can quickly turn into unwanted failures. Researchers at Oak Ridge National Laboratory worked with COMSOL to define a simulation app that automates the software quality assurance (SQA) verification process and provides results in less than 24 hours.

by NATALIA SWITALA

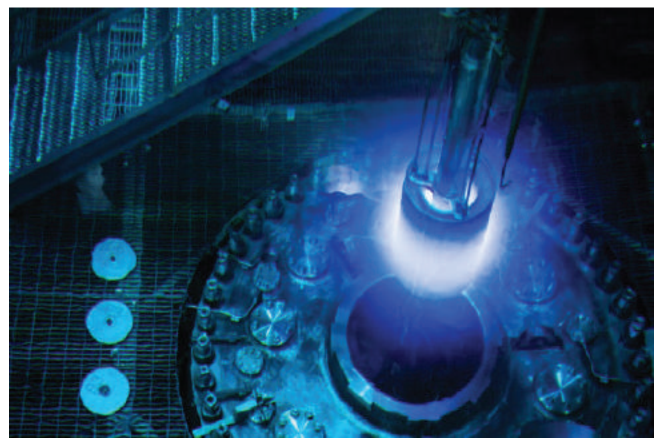
Software updates can feel like an old friend surprising you with insufficient notice that they will be coming to visit. You are equally excited and frantic. You hope everything will go smoothly, that the update is backward compatible with the version you are currently running, and that it passes all of the software quality assurance (SQA) requirements. This scenario is even more exaggerated when the software is used in a highly regulated environment, such as a nuclear research reactor operated for the U.S. Department of Energy (DOE).

⇨ SQA PROCEDURES KEEP US SAFE

When dealing with nuclear energy, there are many safety precautions in place to prevent failures, including SQA requirements that apply to all nuclear-safety-related components associated with the reactor facility.

One task that James D. (Jim) Freels and a team at Oak Ridge National Laboratory (ORNL) are focused on is research and development for the conversion of the High Flux Isotope Reactor (HFIR) fuel from highly-enriched uranium (HEU) to low-enriched uranium (LEU) fuel (Figure 1). In response to the Global Threat Reduction Initiative, many of the world's nuclear research reactors have already been converted. One primary design goal for the LEU conversion of the HFIR is that it remain the highest flux-reactor-based source of neutrons for condensed-matter research in the U.S. and, therefore, remain competitive in the world neutron source market. The unique fuel and core design, as well as the high power density of the HFIR, present a complex and challenging task for fuel conversion. These ORNL researchers use COMSOL Multiphysics® software to explore the impact that the fuel change will have on the HFIR's performance and on the neutron scattering initiatives.

The DOE requires rigorous compliance with SQA standards. Hence, procedures have been developed and are performed by ORNL to adhere to nuclear-safety-related practices. In order to comply, Jim and the ORNL team verify that any software they use behaves as expected by the code developers from the initial installation to the latest update.



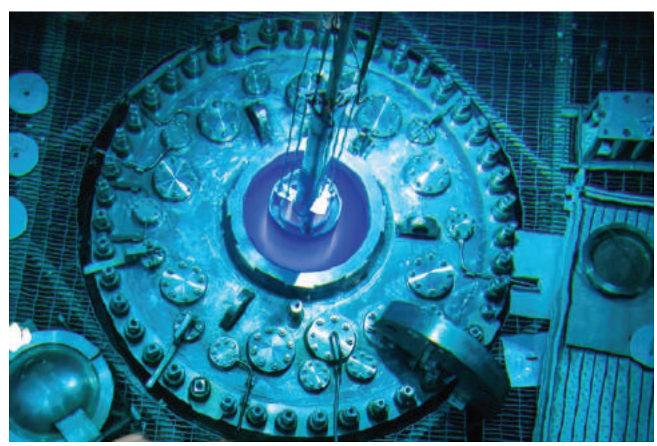


FIGURE 1. HFIR reactor core undergoing a defueling operation.

⇨ REPORTING REQUIREMENTS, SAFETY FIRST!

The SQA process is in place to ensure that the software used to perform an analysis is producing the intended results. "Verifying that a local software installation performs as the developer intends is a potentially time-consuming but necessary step for nuclear-safety-related codes," explains Jim, a senior research staff member of ORNL. ORNL separates their

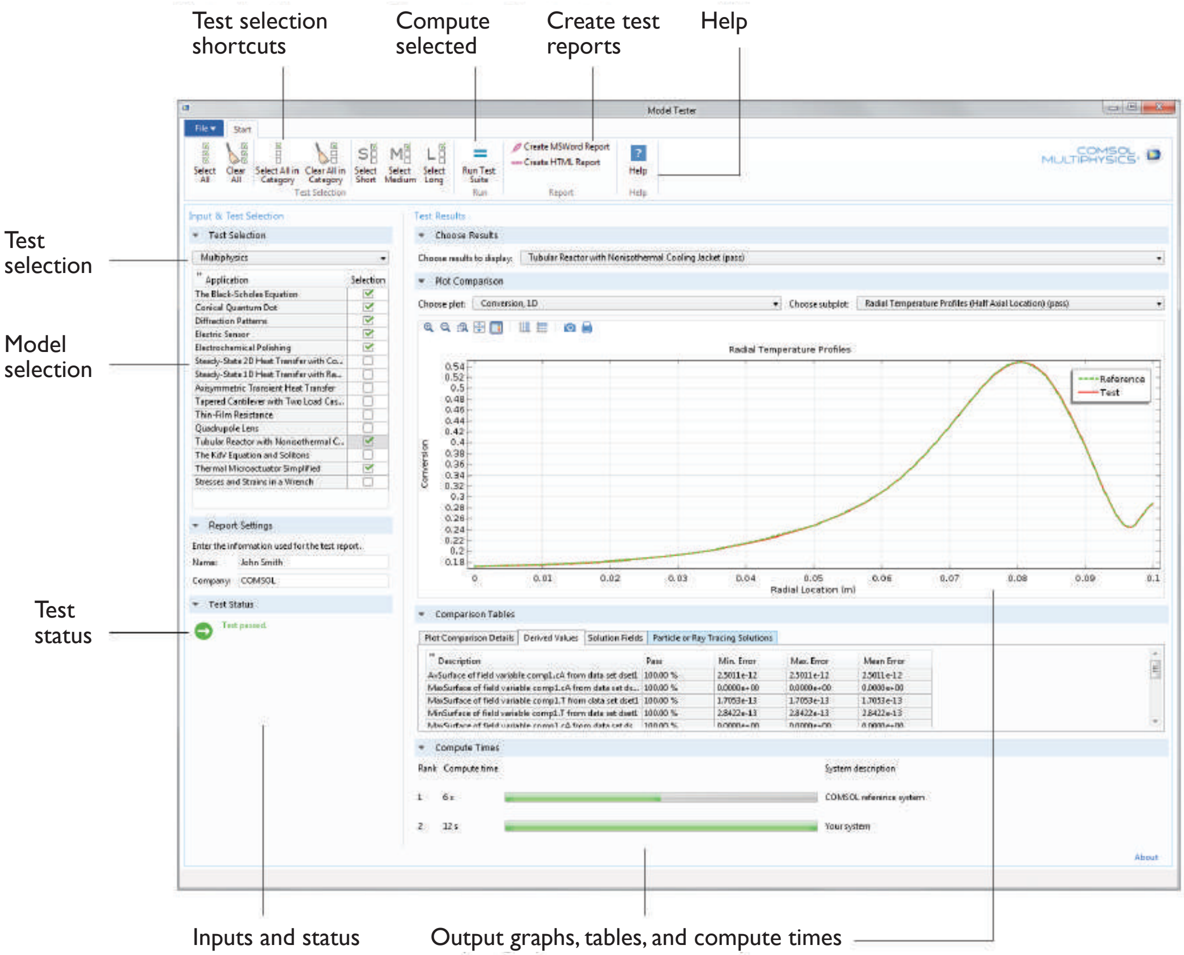


FIGURE 2. Model Tester app that will run a suite of models and compare results, such as temperature and electric potential, against the values provided in the product documentation. The user receives an automated report once the test is complete.

SQA efforts into two categories: verification and validation (V&V).

Compliance with the validation requirements can be the most difficult to meet since the ideal situation is to have experimental, test, or operational data to directly compare and measure the code accuracy. For some codes, particularly codes such as COMSOL Multiphysics that are new to the nuclear industry, the best approach, and the one that ORNL intends to carry out for COMSOL Multiphysics, is to produce a standalone validation report that demonstrates valid results for a number of simulations directly related to their research. Alternatively, the validation task is included as part of the formal nuclear-safety-related calculation process, as controlled and governed by a separate procedure.

While compliance with verification requirements is more straightforward, it can be very time-consuming without the appropriate software tools. DOE requires the team to produce a report that compares the results of a chosen set of COMSOL® software applications run by ORNL and included in the Application Library shipped with COMSOL Multiphysics against the results provided in the software documentation. The number of chosen applications can be large and every output variable reported in the results is required to be documented, resulting in significant resources needed to complete the verification procedures.

Jim says that "with the reporting requirements, qualifying a new version of COMSOL used to take around one to three months to complete, because we had to compare the results from several simulations by hand against the documentation provided by COMSOL."

⇨ STREAMLINING THE SQA PROCESS

This all changed when COMSOL introduced the Application Builder, and Michael W. Crowell (Mike) joined the ORNL team as a nuclear safety and experiment analyst and saw the opportunity to automate part of the SQA process. The Application Builder allows simulation specialists using COMSOL Multiphysics for their physics-based analyses to build a custom user interface for their models. This means that everyone on the team can access a COMSOL model and reap the benefits of the specialists' work without needing to have coding experience specific to COMSOL. In addition to being able to easily build a custom interface, simulation specialists can extend their models with methods written in Java® code that allow them to implement custom commands and link to legacy programs. This is exactly what the ORNL team needed.

The verification procedures require the researchers to demonstrate that the software has installed correctly on specific computers and produces the results intended by the COMSOL Multiphysics developers. Mike was looking for a solution that would allow the team to test whether they would get the same simulation results on different computers using different operating systems and mathematical libraries. "Because of the differences in machine architecture and libraries, along with the limitations of machine precision, we don't expect the included and local results to be identical to the final decimal point, but we expect them to be close enough," Mike explains. The reason behind any discrepancies may be due to, for example, how solvers and meshing algorithms are locally created and compiled, which could affect the final results.

> "The automated report has provided us with newfound time, as well as increased accuracy and reliability."
> — JIM FREELS, SENIOR RESEARCH STAFF MEMBER, ORNL

Prior to the release of the COMSOL-developed Application Builder "Model Tester", Mike had developed a custom MATLAB® software program that automated the verification process in a similar manner by selecting a subset of models from the Application Library that came with the COMSOL® software to run locally and extract and compare the results. This development was documented in a recent paper published through the COMSOL Conference 2015 Boston. The MATLAB® software program that Mike developed compares the results in the models and documentation provided by COMSOL against the locally generated results, and then reports the variance and highlights any out-of-bounds cases. Using Mike's new approach, ORNL was able to reduce the time needed for verification from months to days.

Jim was eager to share Mike's accomplishments with COMSOL, as well as with other facilities working on DOE projects. This shared knowledge led to a conversation with Ed Fontes, CTO of COMSOL, about developing an application to be included in the Application Library for all customers to easily verify a COMSOL Multiphysics installation. Ed explained that these kinds of verification tests are done during the development of COMSOL Multiphysics with several hundred models tested every night. He happily agreed to kick off the project and explained that "the app will allow customers to run a suite of models and compare several physics results, such as temperature and electric potential, against the values provided in the product documentation (and Application Library) and receive an automated report once the test is complete" (see Figure 2).

⇨ AUTOMATED REPORTS UNLOCK PRODUCTIVITY

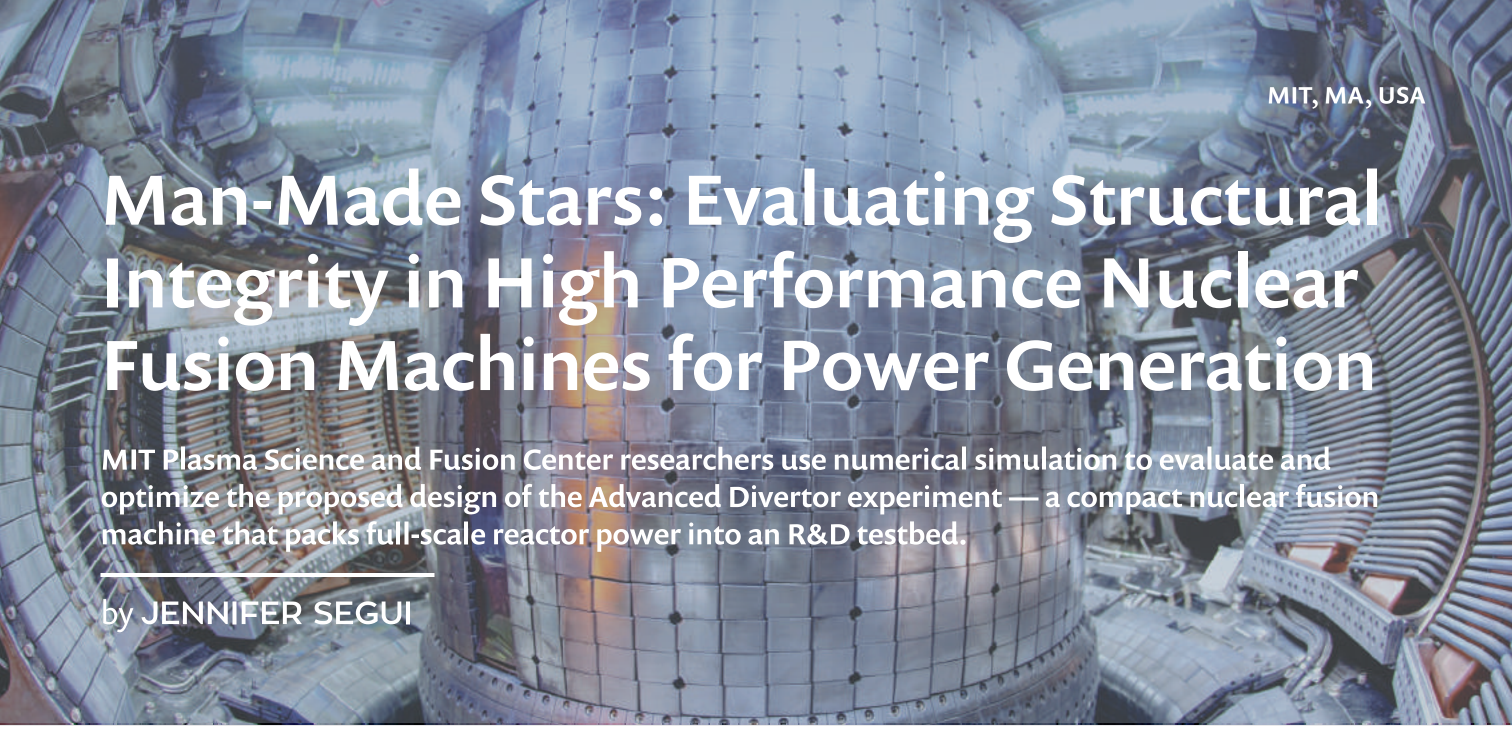
Customers will be able to run models in the Application Library according to their COMSOL Multiphysics license, with the flexibility to select which of the models they want to include in their own installation tests. Once the simulations have been run, the test app will tell the customers which of the models passed and which have failed, including the values where a failure occurred, and present the user with an automated report. The pass/fail criteria is set by default but may be changed to meet the customer's needs. Customers can also extend the tests with their own models and by entering their own reference values for the numerical solution.

"Customers can use the app to compare the results from the previous installation to clearly understand the impact of the possible product updates in a new installation," says Ed. "For example, if COMSOL changes a mesh algorithm or a turbulence model, you will be able to check how the results from your own models and from COMSOL's Application Library are influenced by the updates."

Upon testing a preliminary version of the test app, Mike exclaimed, "We were able to complete the entire verification process in about 24 hours!"

Jim added that "The automated report has provided us with newfound time, as well as increased accuracy and reliability. That allows us to direct our efforts on the work needed to convert the HFIR fuel from a high-performance HEU to an LEU fuel." ❖

Top: The High Flux Isotope Reactor Site at Oak Ridge National Laboratory. Bottom: The core COMSOL group within the Research Reactors Division of ORNL, from left to right: Christopher J. Hurt, Franklin G. Curtis, Prashant K. Jain, Michael W. Crowell, James D. Freels, and Emilian L. Popov.



Man-Made Stars: Evaluating Structural Integrity in High Performance Nuclear Fusion Machines for Power Generation

MIT Plasma Science and Fusion Center researchers use numerical simulation to evaluate and optimize the proposed design of the Advanced Divertor experiment — a compact nuclear fusion machine that packs full-scale reactor power into an R&D testbed.

by JENNIFER SEGUI

Nuclear fusion occurs naturally in the core of the sun, releasing enormous amounts of radiant energy as mass is lost when hydrogen nuclei fuse together to form larger helium atoms. We observe this energy here on Earth as sunlight, despite being on average nearly 93 million miles away.

Demonstrating the feasibility of hydrogen fusion as a clean, safe, and practically limitless source of energy has been the primary objective of over 50 years of international research efforts. At MIT the concept of a very high magnetic field approach to fusion has been the primary focus of research. At the MIT Plasma Science and Fusion Center (PSFC), experiment, leading-edge theory, and numerical simulation are combined to identify and understand the science and technology that can make fusion energy available sooner.

The Advanced Divertor eXperiment (ADX) is a nuclear fusion experiment, and more specifically a tokamak, proposed by researchers at the PSFC to provide heat fluxes, densities, and temperatures similar to what we expect to have in a fusion reactor, though with only short plasma discharges (see Figure 1).

In a tokamak, temperatures in excess of 150 million degrees Celsius cause electrons to separate from nuclei, forming a fully-ionized superheated plasma from gaseous hydrogen fuel. The core plasma is contained within a toroidal or donut-shaped vacuum vessel and maintained at high pressure to produce a dense plasma with high likelihood of collision. External magnetic fields confine and control the plasma in a manner analogous to the intense gravitational fields at the sun's core, thus producing nuclear fusion.

"Recent advances in high temperature superconductors could allow us to design a tokamak operating at higher magnetic fields, increasing the performance of the plasma to reactor levels," explains Jeffrey Doody, a mechanical engineer at PSFC. "The research focus then switches from improving the performance of the plasma to the support systems in the tokamak."

Using numerical simulation, Doody and his colleagues are designing the ADX structure to sustain reactor-level heat fluxes and magnetic fields, making it a suitable testbed for power exhaust systems and plasma-material interactions to support the development of next-stage fusion machines.

⇨ SURVIVING PLASMA DISTRIBUTIONS

The proposed design for the ADX vacuum vessel is innovative in that it is comprised of five separate axisymmetric shells, as shown in Figure 2, instead of a single cylinder. The modular design makes it possible to swap out magnetic coils and test different divertor configurations, where the divertor is a component that serves as the power exhaust system for removing fusion ashes from a tokamak. When ions escape confinement by the magnetic fields that control the plasma, the divertor collects and guides them out of the vessel.

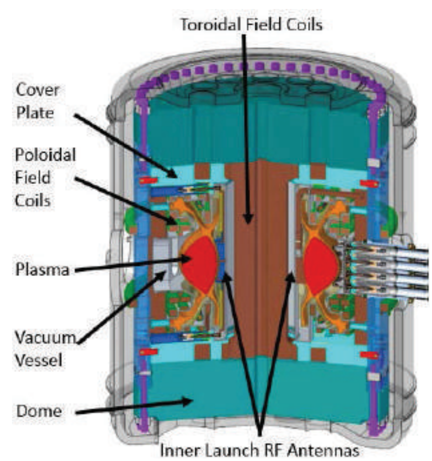


FIGURE 1. Schematic of the proposed ADX tokamak from MIT PSFC.

The modular vessel must not only withstand the high heat fluxes and magnetic fields needed to produce nuclear fusion, but also survive plasma disruptions, which are another source of stress in the vacuum vessel shell generated when the plasma collapses.

"To evaluate the proposed ADX vessel design, we perform numerical

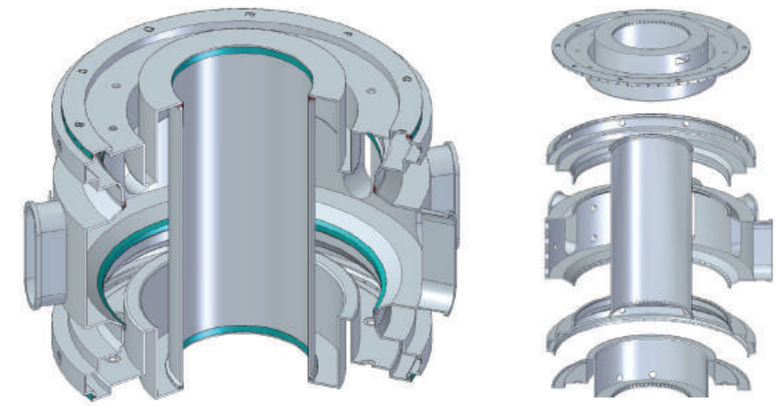


FIGURE 2. The ADX vacuum vessel features a unique design with five separate shells that are bolted together.

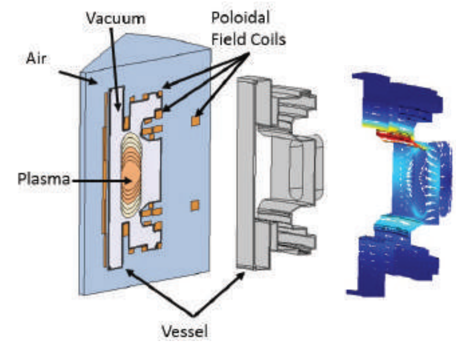


FIGURE 3. Model geometry, at left, used to determine the eddy currents in the ADX vacuum vessel walls, at right.

simulation in COMSOL Multiphysics® software to predict the magnetic fields, eddy currents, and Lorentz forces resulting from a plasma disruption," explains Doody. "The calculated loads are then applied to a separate structural model of the vessel in order to predict stress and displacement." Figure 3 shows the geometry for a cyclic symmetry magnetic model of the ADX, including the vessel, plasma, and poloidal magnetic coils, which are needed to hold the plasma in its equilibrium position.

A worst-case scenario exists for plasma disruptions in vertical displacement events (VDE), where the plasma drifts upward carrying 1.5 million amperes of current, stops moving after 10 milliseconds, and loses all of its current in a single millisecond. Rapidly changing magnetic fields surrounding the disruptive plasma produce eddy currents in the vacuum vessel shell. Lorentz forces are exerted on the vessel when the eddy currents cross both the poloidal magnetic fields, and the stronger toroidal magnetic fields of the tokamak that confine the plasma.

During a VDE, eddy currents are larger in magnitude because of how close the plasma gets to the vessel wall, and VDE is therefore the test case of choice in the computational model of the ADX. Figure 3 shows the eddy current distribution calculated from the numerical model. A second model was developed to determine the Lorentz forces due to the toroidal magnetic fields of the tokamak, where only poloidal fields were included in the first model of the ADX.

⇨ STRENGTHENING THE ADX VACUUM VESSEL

Plasma disruptions result in strong Lorentz forces that act on the walls of the ADX, particularly in the upper and lower pockets of the vacuum vessel during a VDE. In a structural model of the ADX vessel, shown in Figure 4, the top and bottom boundaries are attached to the vessel cover and cannot be displaced during simulation. Loads corresponding to the Lorentz force exerted on the vessel are applied to the

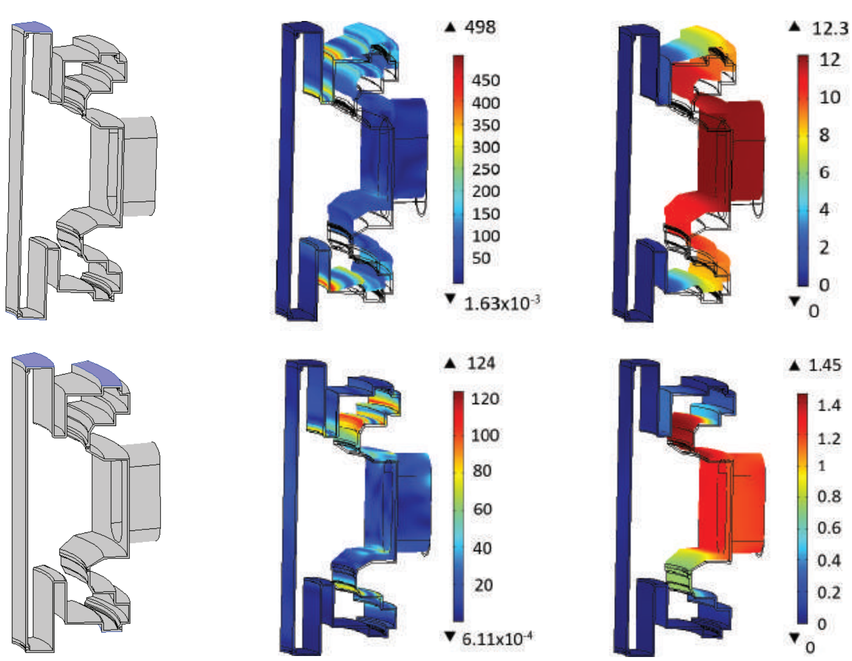


FIGURE 4. At top, the geometry for the structural model of the ADX shows purple boundaries where the structure is fixed. Stress and displacement simulation results indicate the design requires reinforcement. At bottom, the model geometry shows an additional fixed boundary corresponding to a support block added to the ADX design.

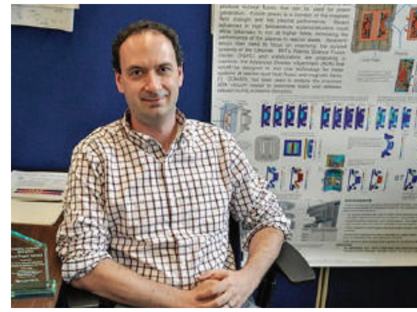
relevant boundaries. In this test case, the Lorentz force was determined for a tokamak operating with a 1.5 million ampere plasma current and 6.5 Tesla toroidal field strength.

The modular vessel components are made from Inconel 625, a strong nickel-based alloy that is also highly resistive to current flow, keeping eddy currents to a minimum. The yield stress for the material is 460 MPa, however the design criteria for the ADX stipulates that the vessel walls should not experience stresses exceeding 306 MPa, which is two-thirds of the yield stress value.

Numerical simulation shows that, without any design modifications, the Lorentz force due to a VDE leads to large stresses in the vessel approaching the yield value, and causing 1-centimeter deflections in the structure. To stabilize the vacuum vessel wall, a support block is added to immobilize an additional boundary as shown in the bottom row in Figure 4. Simulation results, obtained for the case with the support block in place, demonstrate significantly reduced stress and displacement of the vessel wall, indicating that the stabilized vacuum vessel can survive a plasma disruption and support ADX operation.

⇨ NEXT-STAGE NUCLEAR FUSION AND BEYOND

Simulation-led design of the ADX will help ensure its safe, successful operation at PSFC, where it will become the newest fusion machine to serve as an R&D platform to test the divertor concepts required for a fusion reactor. ❖



Jeffrey Doody is a mechanical engineer at the MIT Plasma Science and Fusion Center, pictured here at the COMSOL Conference 2015 Boston where he won an award for his simulation work.

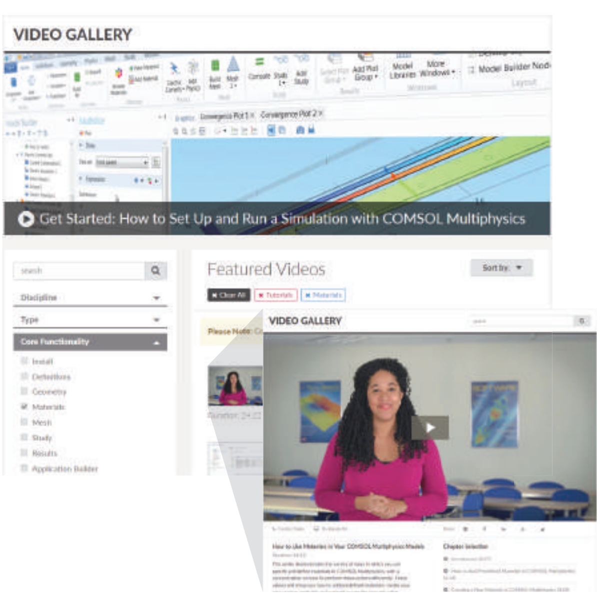
The COMSOL Video Gallery: an Interactive Learning Tool

by ANDREW GRIESMER

The COMSOL Video Gallery contains over 200 videos, with new additions every week. With a focus on supporting users of the COMSOL Multiphysics® software and COMSOL Server™ product, these videos help you get the most out of your numerical simulations.

TUTORIAL SERIES

Tutorial videos show how to use the software, from setting up geometry to postprocessing results and everything between. These videos are created for both beginning users and seasoned simulation specialists, spanning introductory features and advanced techniques alike. Each video is produced with insight from the COMSOL technical support team.



MODEL DEMONSTRATIONS

Want to learn about modeling specific physics phenomena? Model demonstrations take you through all the steps necessary to simulate them, including electromagnetics, structural mechanics, non-isothermal flow, acoustics, chemical reactions, MEMS, microfluidics, and more. These videos will give you a feel for how to create models in COMSOL Multiphysics® software and how to turn them into custom applications featuring a user-friendly interface for everyone to run by connecting to COMSOL Server™ product.

KEYNOTES

Hear from fellow engineers and scientists about how they are using numerical simulation in their industries. Each year at events worldwide, COMSOL users present their work, showcasing best practices and innovative designs. Watch this collection of presentations from the comfort of your desk.

COMSOL VIDEOS ONLINE

Browse the available videos in the COMSOL Video Gallery at comsol.com/videos.

PROMOTING INDUSTRIAL INNOVATION WITH CUSTOM SIMULATION APPS

A research group at the University at Buffalo is sharing simulation expertise across a wide variety of industries through custom simulation applications. These apps allow an end user to perform the desired analyses without needing the understanding and expertise that is required to develop the underlying computational model.

by BRIDGET CUNNINGHAM

Look at any industry today, from automotive design to consumer electronics, and you will find a common thread that binds them together: the demand for more innovative technology. The latest and greatest technologies are continuously surpassed by even more complex and intricate devices that offer advanced features and functionality.

Numerical simulation tools are a viable solution to the challenge of creating more elaborate devices quickly, delivering results with real-world accuracy without the need for building prototypes for each design modification. Some organizations, however, may not have the resources to bring a simulation expert on board to help create and modify models. This is where simulation applications come in. These customized user interfaces are built around numerical simulations of physics-based systems and allow an end user to run multiphysics analyses set up for them by simulation specialists.

With over 30 years of modeling experience in the industrial sector, Edward Furlani, a professor in the School of Engineering and Applied Sciences at University at Buffalo (UB) SUNY with joint appointments in the Departments of Chemical and Biological Engineering and Electrical Engineering, saw an opportunity to bring numerical simulation to a greater number of organizations. His idea: Organize a university group that could work together to develop mathematical models to analyze and design materials and devices for industrial purposes. Now, the potential for the team to extend the reach of their expertise to a broader range of industries is growing, thanks to the ability to create customized simulation apps using COMSOL Multiphysics® software.

⇨ BRINGING MODELING EXPERTISE TO MANY INDUSTRIES

For Furlani, multiphysics modeling has always been an important element in day-to-day work. As a research scientist at Eastman Kodak, he performed modeling in support of material and device development for commercial products, including inkjet systems and myriad digital imaging technologies. Furlani also developed models for a range of industry needs, from photonics and microfluidics to applied magnetics and microsystems technology. In his current role as a professor at UB, Furlani incorporates multiphysics software into the classroom to further help students learn and develop their engineering skills in an interactive way using computational tools.

The focus of Furlani's group at UB is highly interdisciplinary and reflects industrial and academic research. His graduate students develop multiphysics computational models for the development of products with functionality engineered all the way from the nanoscale to the macroscale. His group has created a number of COMSOL® software models with the goal of helping industries continue on the path of innovation and growth.

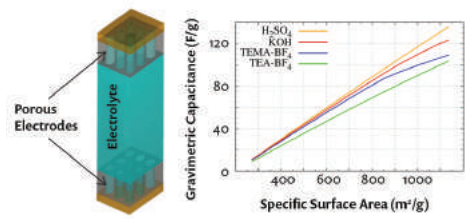


FIGURE 1. The geometry of an EDLC and a plot indicating the predicted capacitance.

When it comes to energy storage, for example, electrochemical-based devices like electric double-layer capacitors (EDLCs) are becoming more and more common. Favored over traditional capacitors for their significantly greater capacitance, these devices offer potential for new operating scenarios that require several rapid charge/discharge cycles. In the capacitors, ions separate and accumulate on the surface of the oppositely charged electrodes in response to an applied voltage. By developing models, Furlani's group, in collaboration with Professor Gang Wu at UB, has been able to predict ion transport and equilibrium charge accumulation within the devices, as well as their capacitance, to foster a better understanding of their behavior and how to optimize their designs (see Figure 1).

Membrane technology is another popular point of interest in the industrial community, with uses ranging from seawater desalination to CO_2 removal from natural gas. Designing thin-film composite (TFC)

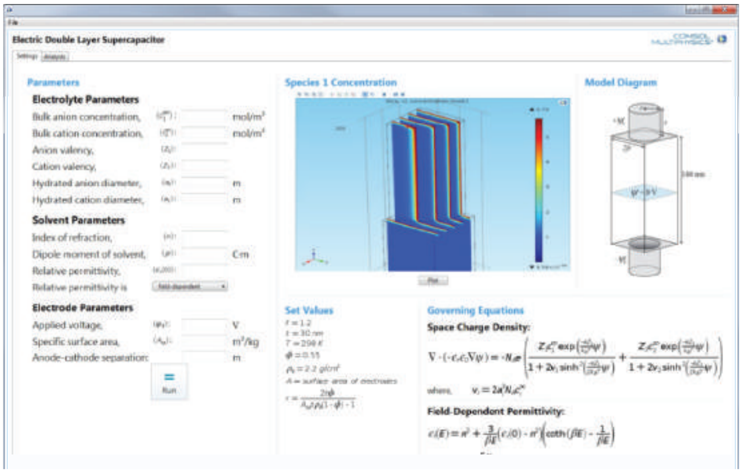


FIGURE 2. Cropped screenshot of an app for EDLC analysis.

membranes with high selectivity is the key to extending the use of membrane systems, keeping costs and the overall footprint at a minimum. The team at UB developed 3D models to see how layer thickness, selectivity, and porosity impact membrane performance. The models, which have been validated experimentally by Professor Haiqing Lin at UB, provide greater insight into how to further optimize these energy-efficient systems and pave the way for additional uses.

Furlani began turning these and other complex models into easy-to-use apps, as shown in Figure 2. Using the Application Builder available in COMSOL Multiphysics, he and his students were able to build a simplified interface on top of each model, customized to meet the specific needs of different companies, and empower a larger group of people to run their own simulation tests. "With custom applications, you have all of the power of COMSOL Multiphysics at your disposal without needing the expertise required to develop the underlying model," Furlani said. "Users can study details that are difficult to measure and can greatly expedite their product development life cycle by reducing costly and time-consuming trial-and-error engineering."

To further promote economic development via mathematical modeling, the team is running COMSOL Server™ product on a computing cluster through the university's Center for Computational Research (CCR) — a supercomputing facility that supports scientific computing, software engineering, and parallel computing. This setup creates a high-performance and high-throughput computing environment for running apps where end users can connect to COMSOL Server through a client or a web browser. Behind-the-scenes simulation specialists can easily manage and deploy their apps and make any updates available right away through COMSOL Server.

⇨ THE BUSINESS OF BUILDING CUSTOMIZED SIMULATION APPS

The group has also been involved in simulation activities for the startup world. At Vader Systems, a startup company in Buffalo, NY, founders Zachary and Scott Vader have developed an innovative process known as liquid metal jet printing (LMJP). Designed to mimic inkjet printing, this technology, which is based on magnetohydrodynamics (MHD), involves liquefying a solid metal feed and ejecting molten metal droplets via a pulsed electromagnetic field. This enables the printing of highly complex 3D metal objects. As they commercialize printers based on LMJP, Furlani's group has developed COMSOL apps to better understand the printing process and enhance its capabilities.

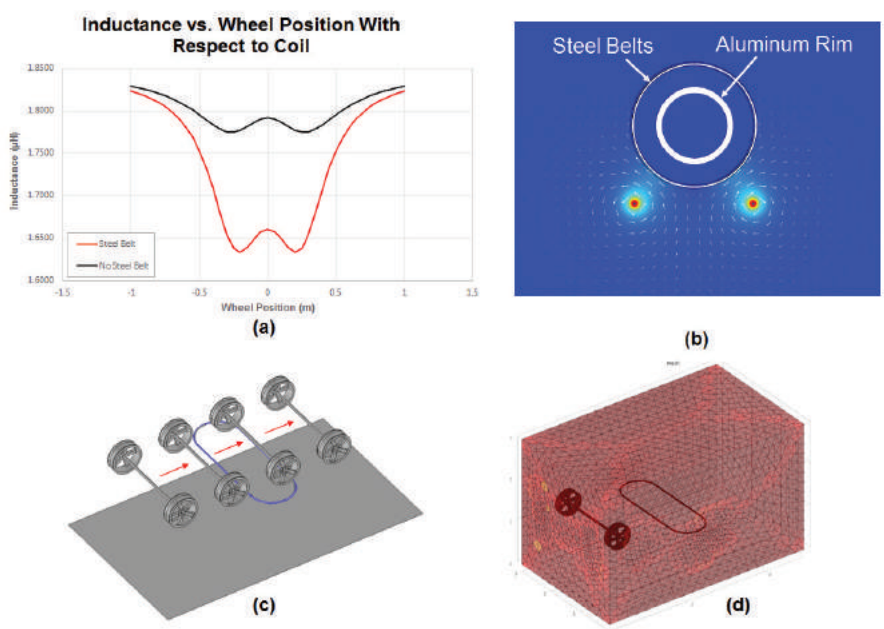


FIGURE 3. A simulation of an inductive coil coupling in a moving vehicle. The analysis examines the impact of steel belts in a moving tire on coil inductance (a) and the magnetic field distribution with a tire moving above an inductive coil (b). The vehicle axle motion is shown (c), as well as a 3D computational domain for simulating the detection of the axle (d).

Another important project has been carried out with scientists at Xerox led by Dr. Peter Paul, where they contributed to the development of novel inductive loop sensing systems that can be used in vehicle transportation projects. Along with monitoring and managing traffic control, the remote sensing technology could also prompt new techniques for collecting traffic data.

> *"With simulation apps, you can easily customize the user interface and include parameters that are of interest to different companies, which is a very useful feature."*
>
> —EDWARD FURLANI, PROFESSOR IN THE ENGINEERING SCHOOL, UNIVERSITY AT BUFFALO

Maximizing the performance of the systems requires an understanding of the electromagnetic coupling between the sensing coils that are embedded in the pavement and ferrous and metallic components of the vehicle, such as wheels and axles. Here the team used COMSOL Multiphysics to perform simulation studies to see how the coil inductance changes as a function of the position and motion of aluminum wheels and tires that contain steel belts (see Figure 3).

While working on these and other projects, the need to involve more people in the design workflow for each of the projects became evident very quickly. With the underlying models established, the team sought to create an interactive tool that could include more people while meeting their individual needs. The answer came, once again, in the form of building custom applications.

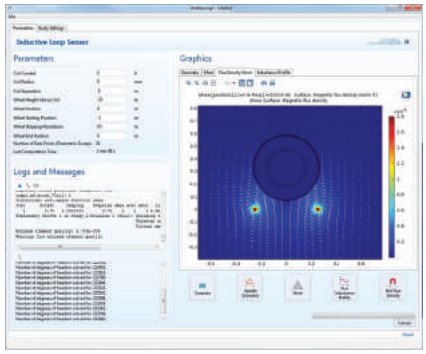


FIGURE 4. A screenshot of the app created for Xerox based on the underlying model for inductive loop sensing.

"With simulation apps, you can easily customize the user interface and include parameters that are of interest to different companies, which is a very useful feature," Furlani stated.

Such customization and flexibility is complemented by the new ease of communicating to and throughout organizations. The need for an end user to use COMSOL Multiphysics is essentially eliminated through the installation of COMSOL Server, which provides access to apps. By bringing simulation capabilities into the hands of more people within the organization, design changes can be addressed more efficiently with ensured accuracy. The teams at Vader Systems and Xerox (whose app is shown in Figure 4) have already experienced the benefits of using apps, such as creating a more collaborative and efficient product development cycle.

⇨ FROM STUDENTS TO ENTREPRENEURS

Simulation tools are also an engaging way to help students enhance their engineering skills. Because apps are designed to hide the complexity of the underlying model, they can serve as a helpful guide to students who are newer to simulation or the COMSOL software. As they become more acquainted with the different features and functionality, students can begin to delve deeper and learn how to utilize new tools.

Aside from serving as a helpful introduction to the software, students have the opportunity to engage with the business side of creating apps. Designing an app, Furlani notes, is not a one-step process. "There is also the work that goes into maintaining it," he said. "It is important that you stand by and provide support to customers, offering them help and engaging with them as they need it."

In this sense, the art and science of building apps develops business skills as students learn how to address questions and requests from customers in a timely manner. And by building apps on their own, students are adjusting to a new wave of simulation-led design, one that opens the door to greater freedom, flexibility, and entrepreneurship. ❖

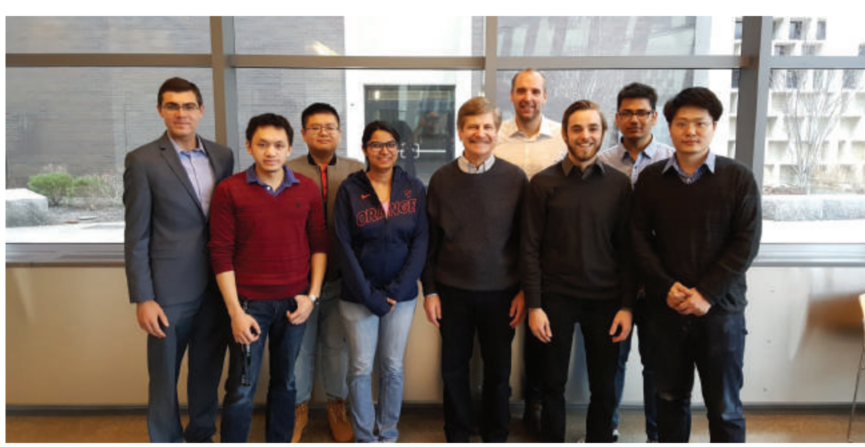


FIGURE 5. Members of Furlani's group (from left to right): Dante Iozzo, Mike Tong, Xiaozheng Xue, Aditi Verma, Edward P. Furlani, Ioannis Karampelas, Viktor Sukhotskiy, Gouray Garg, and Kai Liu.

Multiphysics Analysis Helps Preserve the Past

Students and consultants are collaborating to improve the built environment and to save historic structures and artifacts with the use of simulation apps.

by **GARY DAGASTINE**

FIGURE 1. From left to right, the images show different scales of structures of interest to the built environment, from large land areas down to the materials used in a window.

There's always room for improvement in the things we humans create, and nowhere is this more true than in the so-called built environment, the term for human-made physical surroundings in which we live, work, and play.

The built environment encompasses everything from large metropolitan areas to individual buildings, parks, roadways, and the infrastructure with which we interact over the course of our lives. Although it may not always seem dynamic, it is continually impacted by a myriad of physics-based processes, such as heat transfer, air flow, and moisture transport, which take place on many different scales (see Figure 1).

The ability to model and simulate these processes accurately can lead to significant improvements in many important areas such as energy efficiency, health and safety, operating costs, durability, and notably, historic preservation.

Deeply immersed in this work is Jos van Schijndel, founder of Netherlands-based consulting firm CompuToolAble and assistant professor at Eindhoven University of Technology, Netherlands, where he specializes in the mathematical modeling of building physics.

"What gives me joy and energy is to work on projects that not only incorporate leading-edge research, but which are relevant to society and can also inspire my students," van Schijndel said. "I formed CompuToolAble to offer clients our expertise in advanced mathematics and computational tools, and the ability to perform complex numerical experiments to create innovative designs and optimize their performance."

As a consultant and professor, van Schijndel often needs to think of ways to make complex ideas accessible to his clients who are not simulation experts or to students who are still learning about modeling and simulation work. As a COMSOL Multiphysics® software user, he takes advantage of its Application Builder, which enables him to build intuitive user interfaces powered by a COMSOL® software model. This allows him, as a simulation expert, to efficiently collaborate with clients and colleagues in other organizations and departments.

"Using COMSOL apps first gives students an entry point into numerical analysis and physics systems before they continue on to building their own models from scratch."

⇨ APPS FACILITATE THE USE OF NUMERICAL SIMULATION

To his corporate clients, van Schijndel offers numerical analysis and testing capabilities through these customized user interfaces, or applications, built from COMSOL models. These apps make it easy for an end user who has never used the simulation software before, or who doesn't have the technical background to create a model, to run virtual tests on different design changes based on their specific needs.

In the university setting, he uses the same software to introduce students to multiphysics simulation and application design. Using COMSOL apps first gives students an entry point into numerical analysis and physics systems before they continue on to building their own models from scratch. Once they are familiar enough with the concepts and the modeling techniques, they can eventually create their own apps using

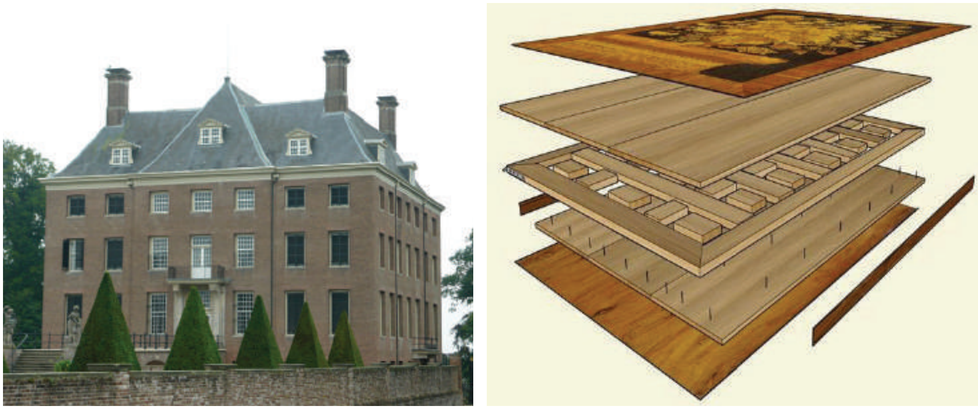


FIGURE 2. At left, Amerongen Castle is one example of an historic structure that is subject to the potentially damaging effects of heat- and moisture-driven stress and strain over time. This applies both to the building itself, and to the treasures the castle contains, such as the cabinet door at right (source: Rijksmuseum Amsterdam).

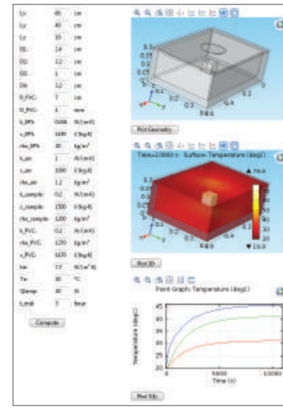


FIGURE 3. App users can explore a variety of designs and operating scenarios, e.g., different geometry parameters and material properties.

the Application Builder, further expanding their knowledge and the reach of their collective analysis capabilities.

"Application Builder is very important to me in two distinct ways," van Schijndel said. "At the university, it's an ongoing challenge to motivate students to use their creativity and ingenuity before they dive too deeply into mathematical modeling. But now, I can easily build engaging simulation apps with just the characteristics I want my students to explore. Then, only after they've had a chance to play with the apps to analyze the physics involved and to understand the effects when various changes are made, do we get into more detailed work.

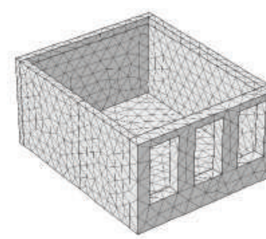
"On the corporate side," van Schijndel continued, "often there are people who need to use numerical simulations so that they can see and understand the impacts of physics processes on their products, but don't have enough experience or interest in building models themselves. Using the Application Builder I am able to create a specialized user interface based on advanced numerical models and provide them with only the parameters that they are interested in. This also ensures no errors are introduced, because we are all working from the same reference point." An app can be deployed for use by everyone who needs access to it, through either a web browser or a downloadable client and letting them connect to COMSOL Server™ product (as shown in a cropped screenshot of the app in Figure 3).

⇨ MULTIPHYSICS ANALYSYS AIDS HISTORIC PRESERVATION

When it comes to the preservation of historic buildings and the artifacts they contain (see Figure 2), it's imperative to examine the combined impacts of heat transfer, air flow, and moisture transport in order to blunt their deleterious effects. Fluctuating levels of temperature and humidity can increase the stress and strain on historic structures and objects, which in turn may lead to warping, cracking, dimensional changes, and other forms of damage. Having a better understanding of the dynamics of these processes and how they occur can lead to more effective preservation efforts.

Van Schijndel has created and overseen the creation of many COMSOL models related to the preservation of historic buildings and items in close collaboration with colleague Henk Schellen, associate professor in Physics of Monuments. For example, one of their PhD students, Zara Huijbregts, used the Heat Transfer Module, an add-on to COMSOL Multiphysics, to model how sunlight streaming through windows would heat the floor and walls of a room at different times over the course of a day (see Figure 4).

The simulation combined conductive heat transfer through the building envelope, convective heat transfer and measured indoor air, and radiant heat transfer. The model included different surfaces of



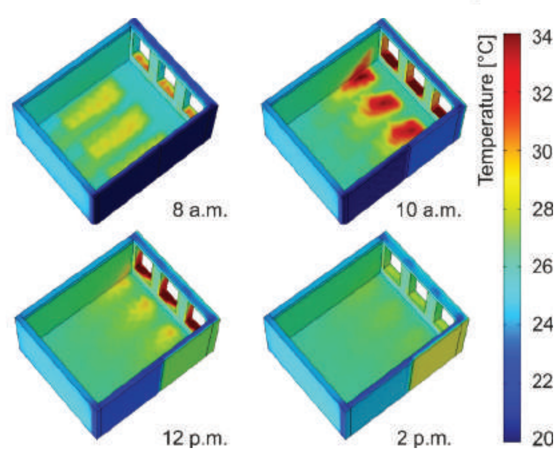


FIGURE 4. A numerical simulation in COMSOL® software of temperature distribution in the floor and walls of a room at different times of day. The heat comes from solar radiation streaming through the windows as the sun passes over the room.

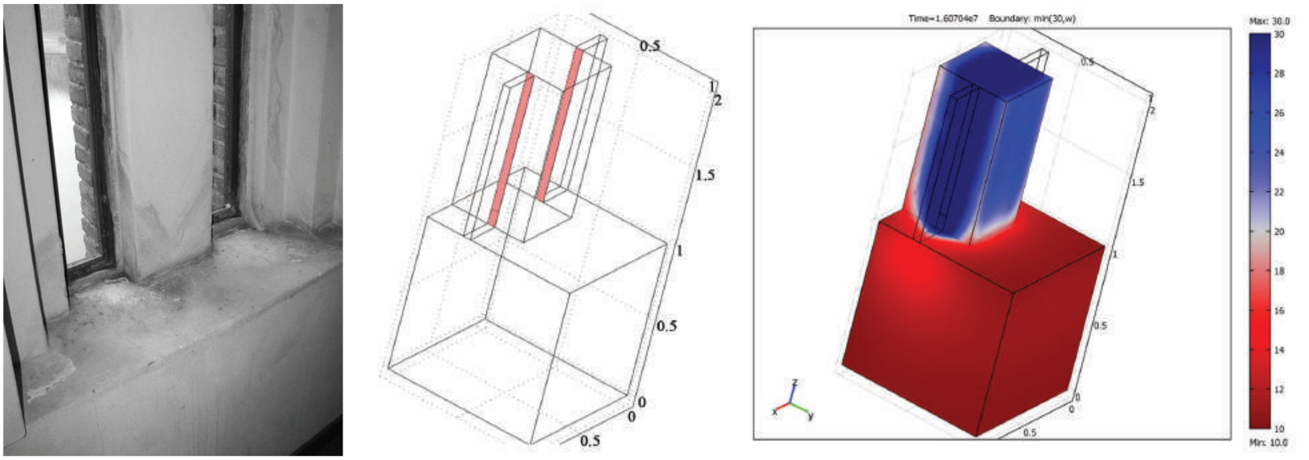


FIGURE 5. To understand the moisture transport, which can cause damage like that pictured above (left), van Schijndel created a COMSOL model that accurately couples heat and moisture transport through a wall (right).

the room and the building façades, and also accounted for the cooling effects of outdoor shade.

The results predicted the temperature on each wall at different times of day, indicating the best locations for protecting sensitive objects from sunlight and heat damage. "It is extremely important to know such precise temperature distributions so that, for example, paintings can be hung in locations where they will be less affected by solar heating," van Schijndel said.

Moisture damage is another major issue that can damage walls, buildings, roofs, and the contents of a room. Because in older structures it isn't always clear where the moisture enters a building or room, van Schijndel sometimes uses COMSOL Multiphysics as an investigative tool.

"The idea is to deduce how moisture might be entering by running multiphysics simulations that include measurements of relative humidity at many different locations. Areas of high relative humidity give clues as to how the mass transport of moisture occurs, and we couple this information with heat transfer simulations because heat drives the moisture distribution," he said.

Van Schijndel developed a COMSOL model of moisture transport in a wall with visible leakage damages near the window. His model enabled him to deduce where the moisture was coming from (see Figure 5). He and his students used this technique to model thermal bridges using different materials, analyze how the positioning of insulation influences heat transfer from the inside to the outside of a building, and to understand how moisture travels through different building materials such as concrete, stone, and insulation (see Figure 5).

Given that temperature and humidity are major drivers of stress and strain on precious objects such as historic paintings, the ability to accurately couple these together in a model can lead to better predictions of potential damage. Van Schijndel's model calculates the temperature distribution throughout the wall and painting, as well as the relative changes in humidity (see Figure 6).

"Historic structures are facing unprecedented threats to their integrity from climate change, increased urbanization, and other factors. At the same time, the regulations governing their preservation have never been stricter, because it is a societal benefit to preserve the legacy of the past. The ability to use sophisticated modeling and simulation tools such as COMSOL is an important part of the toolkit that helps us meet those requirements," said van Schijndel. ❖

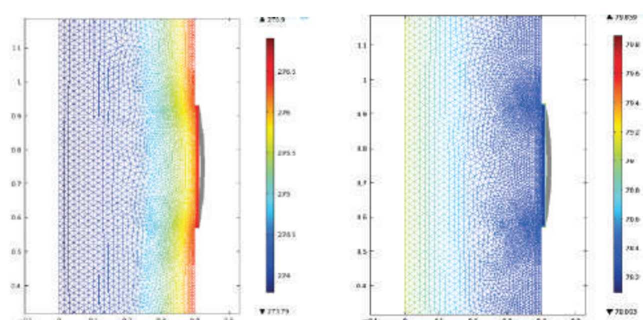
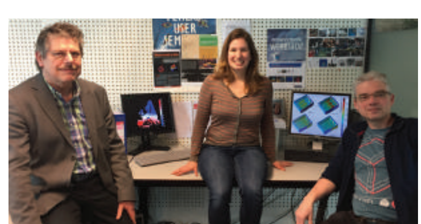


FIGURE 6. Both images show COMSOL simulations of a cross section of a wall on which a painting is hanging. The left image shows the temperature distribution in the wall and painting at a given point in time, while the image on the right shows the rate of change of relative humidity in the wall and the painting at that same point in time. This model can help predict stress and strain, which impacts historic buildings and artifacts, giving experts the information needed to take appropriate damage mitigation actions.



From left to right: Henk Schellen, Zara Huijbregts, and Jos van Schijndel.

BE CAE, ITALY

THERMAL CHARACTERIZATION OF AN ELECTRONIC DEVICE WITH A CUSTOM APP

By distributing high-fidelity multiphysics models as custom simulation apps, engineers at BE CAE & Test leverage their expertise and streamline the consulting process.

by GIUSEPPE PETRONE

Simulation consultants are using custom applications as an effective way to communicate their work to clients. Instead of delivering a static report, they can now deploy a product that contains the intricacy of an unabridged mathematical model, with the clarity and usability of an app. This lets clients run their simulations independently. At BE CAE & Test, we have created such an app to simulate a surface-mount device (SMD).

⇨ SIMULATION APPLICATIONS FOR ENHANCED ENGINEERING COMMUNICATION

In simulation consulting, apps are the next step in engineering communication: They're a streamlined way to communicate and work with clients. With a custom application, a client can access a complete simulation through a user-friendly interface. Using an app is advantageous for both the simulation specialist and the client, as the client receives an easy-to-use tool with which they can independently investigate their system, and the numerical expert is able to dedicate more time to the detail of the simulation rather than running computations for the client.

In this example, which describes the characterization of an SMD, the client has access to the numerical model through the app and can modify a few parameters and material choices.

⇨ THERMAL CHARACTERIZATION OF A SURFACE-MOUNT DEVICE

Whether devices use or convert energy, they must properly manage heat so that they continue to operate in a designated temperature range. An SMD is an example of one electronic system that clients ask us to model. We make use of COMSOL Multiphysics® software to investigate these systems due to the wide range of physics that can be taken into account and the ease with which one can couple them.

In our SMD model, the parts we're mainly interested in are the copper frame, lead-free solder layer, and the silicon die (see Figure 1).

The material of the solder layer and silicon die, the thickness of the solder layer, and the dissipated thermal power each have the potential to impact the maximum junction temperatures and junction-to-case thermal resistance. In our model, we investigate the effect of varying these parameters on heat distribution, as it ultimately affects the

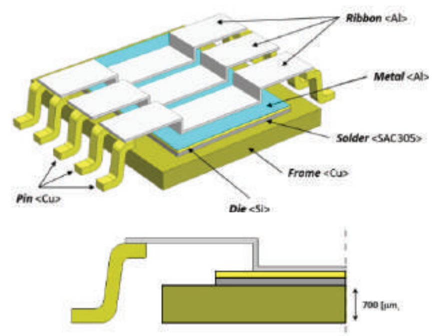


FIGURE 1. Top: Geometric details and materials used in the SMD. Bottom: Side view of the frame, die, solder, pin, and ribbon.

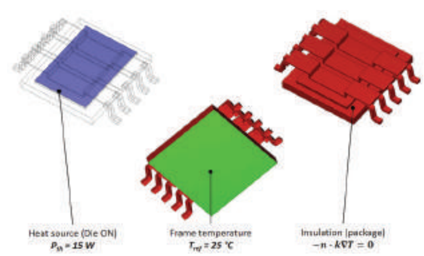


FIGURE 2. Highlighted regions of the SMD depicting the boundary conditions used in the multiphysics model.

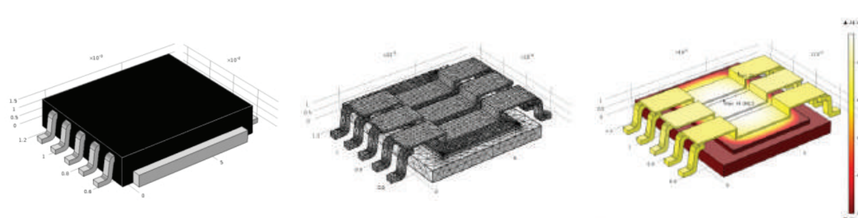


FIGURE 3. From left to right: 3D geometry, mesh, and simulation results from COMSOL Multiphysics® software.

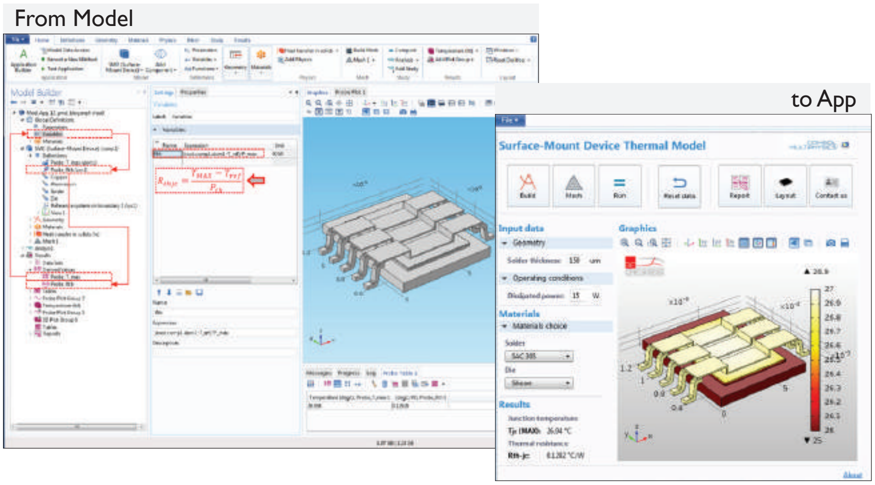


FIGURE 4. The Surface-Mount Device Thermal Model app, created with the Application Builder available in COMSOL Multiphysics® software. The user can change factors such as the solder thickness, operating conditions, and materials in order to analyze the thermal behavior of different SMD designs.

proper functioning of the SMD. In our test simulation run, the heat source is 15 W, the initial frame temperature is set to 25°C, and the remaining parts are thermally insulated.

With COMSOL Multiphysics, it is straightforward to model heat transfer through the device, as all modeling steps are carried out in the same environment. We were able to quickly build the geometry; add materials; use the Heat Transfer in Solids physics interface to set up boundary conditions; mesh; solve; and postprocess the results with the expressions we defined, such as the junction-to-case thermal resistance.

⇨ BUILDING A CUSTOM SIMULATION APP FROM A NUMERICAL MODEL

Once the COMSOL Multiphysics model is complete (see Figures 2 and 3), it can be wrapped in a user-friendly interface with the Application Builder tool. As the experts in the physics at hand, we consider both our mathematical model and our client's specifications in order to choose the parameters that the app user can access and modify within an acceptable range (see Figure 4).

The app user can see the geometry of the SMD, adjust the solder thickness, generate a mesh, launch the simulation, return to default settings, and generate a report. These features are easily created with the functionality available in the COMSOL Application Builder.

As the application user progresses through the steps of the simulation, graphical output is displayed: first the geometry of the SMD, then its mesh, and finally the computed temperature distribution. Thus, the interactive, dynamic nature of the model is preserved in the app. When the app user modifies a parameter, all visualizations are easily regenerated.

The final result is a clear and easy-to-use application that relies on the accuracy and predictive power of the mathematical model defined by simulation specialists, but does not overwhelm the user. The app described here allows the user to examine maximum junction temperature and junction-to-case thermal resistance as a function of the materials of the components, thickness of the solder layer, and dissipated thermal power. The app user can modify parameters and view the output quickly, confirming or contradicting predictions to make informed design decisions.

⇨ IMPROVING ENGINEERING COMMUNICATION

In our experience of creating simulation apps, clients have been pleased to be provided with an interactive tool with which they can investigate their system.

Previously, after providing clients with the outputs of simulations, there would be requests for further computational modeling with different parameters. Now, it is possible for the simulation expert to deploy a custom application with which the client can investigate all of their remaining uncertainties. This is optimal for the simulation expert as well as clients — the time spent on repeating simulations with different parameters is better spent adding complexity to the simulation and app.

Now, with simulation apps, upon receiving a request for parametric simulations, we can say, "Let us provide you with a COMSOL custom application, and you will be able to inspect your product yourself." ❖

> "In simulation consulting, apps are the next step in engineering communication: They're a streamlined way to communicate and work with clients."

ABOUT THE GUEST AUTHOR

Giuseppe Petrone is a cofounder and the sole administrator of BE CAE & Test, a COMSOL Certified Consultant. He received his master's degree in mechanical engineering from the University of Catania in Italy and later earned his PhD in energetic and process engineering from the Université Paris-Est in France. Before starting BE CAE & Test, Petrone devoted his time to academic research ventures, which included exploiting numerical methods in fluid dynamics and thermal analyses. He has been a user of the COMSOL Multiphysics® software since 2005.

COMSOL Multiphysics® Brings Innovation to Academia and Industry

by CARL D. MEINHART

I use COMSOL Multiphysics® software in both academia and industry to solve complex multidisciplinary problems. The reason is simple: What would previously require a custom numerical algorithm, from advanced multiphysics applications to optimization analysis, can be easily solved by COMSOL® software.

At the University of California, Santa Barbara, COMSOL has been incorporated into several graduate and undergraduate classes. For example, in the Introduction to Multiphysics Simulation class, we teach predominately mechanical engineering seniors and graduate students. This course provides seniors with the necessary skills to use COMSOL for the design and optimization of their year-long capstone projects. It allows students to iterate through a variety of design ideas in the fall quarter of senior year, before they ever build hardware. This process saves significant time and energy, and teaches students modern industrial design practices.

> "With COMSOL, we are able to quickly simulate and analyze many design concepts, and optimize the designs, before committing to actual hardware fabrication."

We're also planning to use custom applications created with the Application Builder. Apps will most likely first be used to teach undergraduate laboratory classes. Students can use pre-built applications to simulate physical experiments and predict the expected results before actually performing them in the lab. During the analysis stage, students will compare simulation and experimental results to understand the physics as well as experimental and numerical errors.

Many projects are multidisciplinary in nature, spanning the fields of engineering, chemistry, physics, and biology. One unique advantage of COMSOL is that vastly different physics can be easily coupled to solve research problems or to answer scientific questions that previously required years of developing dedicated numerical algorithms. This flexibility allows researchers to spend more time thinking about the physics and less time worrying about the numerics.

One example is in the field of microfluidics, where AC electrothermal flow can be used to drive heterogeneous chemical reactions. These models require us to consider electrostatics, heat transfer, fluid flow, and chemical reactions. To further increase the high fidelity of our numerical simulations, our mathematical models also take into account the fact that material properties are temperature-dependent. The models allow the physics to be fully coupled and interact exactly as they do in the real world.

I founded Numerical Design, Inc. in 2012 and since its inception, it has been a COMSOL Certified Consultant company. We serve a variety of industries in technical fields such as microfluidics, electromagnetics, chemical engineering, and many others.

We work on many microfluidics applications occurring on length scales ranging from tens of nanometers to tens of microns, and time scales on the order of a few microseconds. Because of the small length scales and short time scales, it is often very difficult to instrument devices for experimental performance characterization. Fortunately, COMSOL Multiphysics allows us to simulate these devices and numerically design them for optimal performance.

We routinely design microfluidics devices that would be impossible to create by instrumenting hardware prototypes. Furthermore, the microfabrication process for many microfluidic devices can cost tens of thousands of dollars per microfabrication run and may take, say, six months to complete. This large cost in both time and money can make multiple design iterations impractical. With COMSOL, we are able to quickly simulate and analyze many design concepts and optimize them, before committing to actual hardware fabrication.

There are many optimization tools available commercially. However, for multiphysics-based shape optimization, COMSOL is the ideal tool for the design of complex structures that cannot be engineered strictly from first principles. In one industrial example, we recently conducted a 10-parameter optimization of a nonlinear multiphysics problem. If all possible parametric combinations were studied and each parameter was swept with, say, 10 steps, the result would be 10^{10} simulations. Assuming each nonlinear sweep took approximately 30 minutes to solve, it would take approximately 570,000 years to complete. By using COMSOL, we can optimize the problem within approximately one day. The resulting geometric structure is non-intuitive and could not have been predicted *a priori*.

ABOUT THE AUTHOR

Dr. Carl Meinhart is a professor of Mechanical Engineering at the University of California, Santa Barbara. He earned his PhD from the University of Illinois in 1994. Since coming to UCSB in 1996, his research has focused on developing microfluidic devices and investigating their fundamental transport mechanisms. Dr. Meinhart is also founder and CEO of Numerical Design, Inc., which is a COMSOL Certified Consultant company based in Santa Barbara, CA. Dr. Meinhart is a fellow of the American Physical Society.