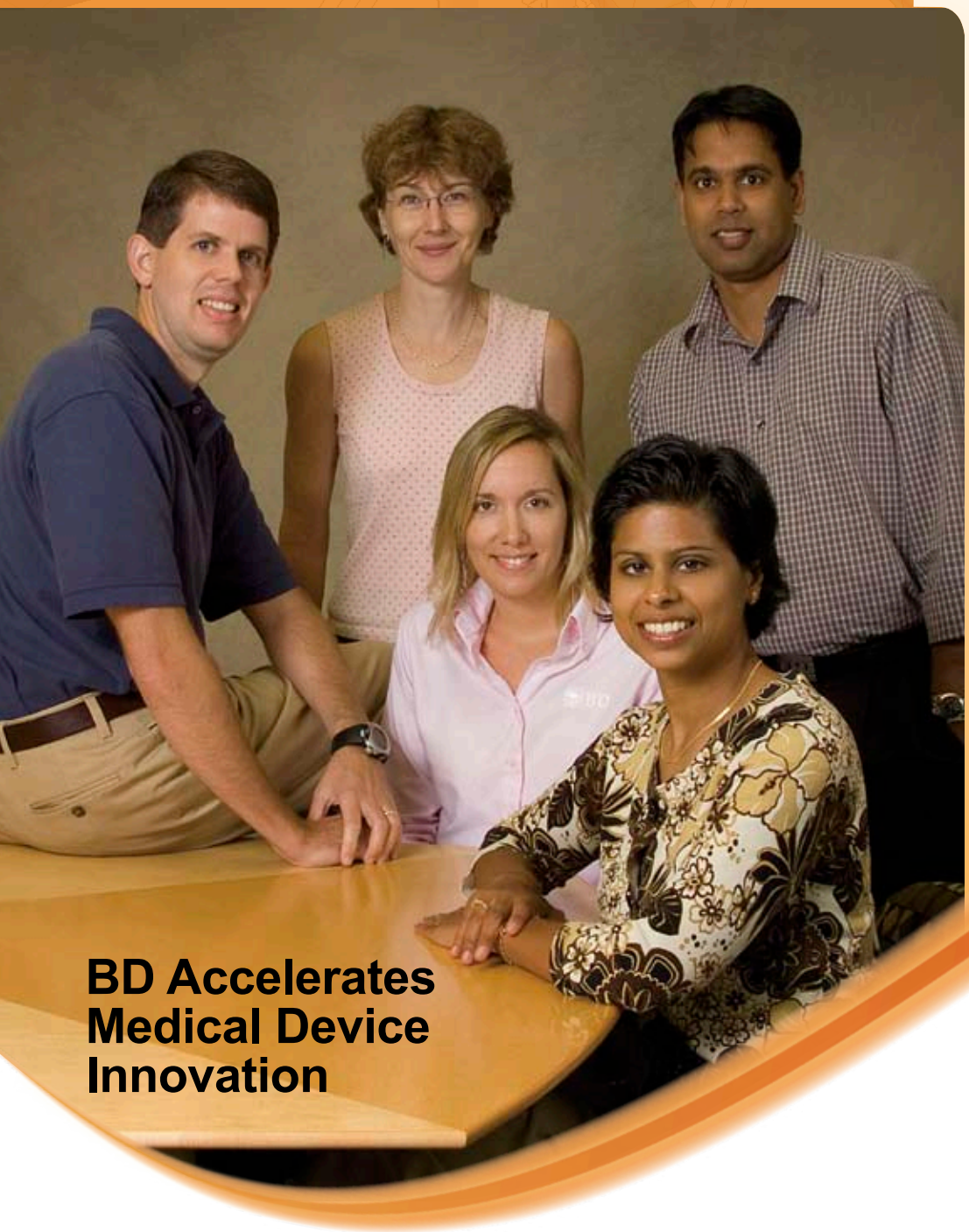
INSIGHTS

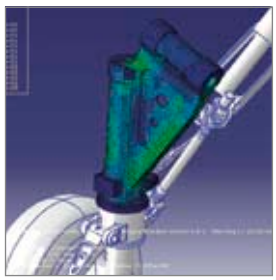
5
10 2008
Dassault Systèmes Realistic Simulation Magazine



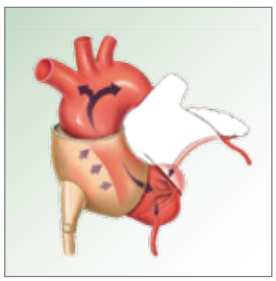
BD Accelerates Medical Device Innovation



GN ReSound
Improving Hearing Aid Performance



Isight for Abaqus
New Product for
Design Exploration

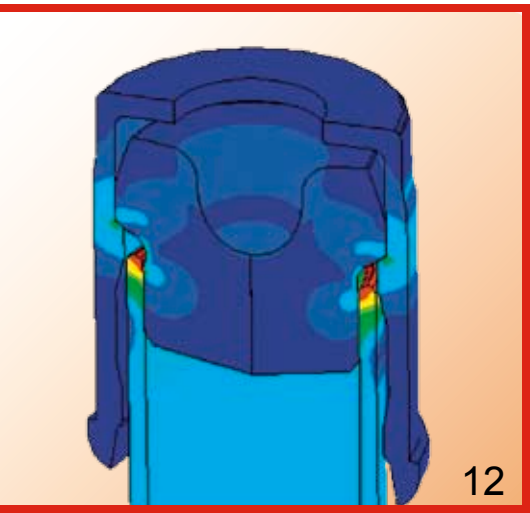


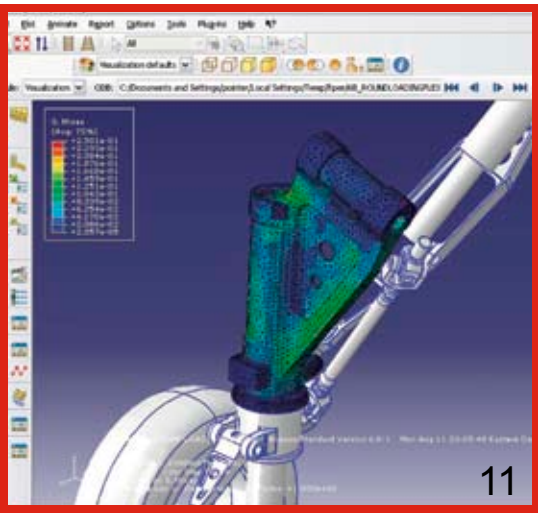
Sunshine Heart
Simulating Heart Pump Behavior

SIMULIA

INSIGHTS

Sept/Oct 2008







Inside This Issue

12 Cover Story
The Role of Simulation in Product Development at BD

8-11 Product Update
- CZone for Abaqus
- Abaqus for CATIA V5
- SIMULIA SLM New Release
- Isight for Abaqus

6 Customer Spotlight
Hearing Aid Design is a Resounding Success with Finite Element Analysis

In Each Issue

3 Executive Message
Scott Berkey, CEO, SIMULIA

4 In The News
- Industry Press Coverage
- Nonlinear Analysis of I-35 Bridge Gusset
- Cambric Expands Solutions Offering
- Lenovo Uses Abaqus

10 Product Update
Design Exploration, Components, and Process Integration Technology Extend SIMULIA SLM

14 Life Sciences Strategy
Subham Sett, Medical Industry Lead, SIMULIA

16 Customer Case Study
Sunshine Heart Optimizes Unique Pump Device

19 Alliances
Simulayt Provides Enhanced Composite Modeling and Simulation

20 Academics
- Western Michigan University Canoe Team Stays Afloat with Abaqus
- Abaqus Student Edition 6.8
- Flexible Licensing for Academics

22 Services
Accelerating Analysis with HPC

23 Events
- 2009 SCC – Call for Papers
- 2008 RUM Schedule

INSIGHTS is published by
Dassault Systèmes Simulia Corp.
Rising Sun Mills
166 Valley Street
Providence, RI 02909-2499
Tel. +1 401 276 4400
Fax. +1 401 276 4408
simulia.info@3ds.com

www.simulia.com

Editor:
Tim Webb

Associate Editor:
Julie Ring

Contributors:
Upul Attanayake (Western Michigan University), Graham Barnes (Engenuity), Ryan Becket, Scott Berkey, Dale Berry, Anita Bestelmeyer (BD), Rob Bray, Wei-Shan Chang, Matt Dunbar, Kyle Indermuehle, Bill Klug (UCLA), Rob Miller, Scott Miller (Sunshine Heart), Parker Group, Marc Schrank, Subham Sett, Louise Short, Morten Birkmose Søndergaard (GN ReSound)

Graphic Designer:
Todd Sabelli

The 3DS logo, SIMULIA, and Abaqus are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries. Other company, product, and service names may be trademarks or service marks of their respective owners. Copyright Dassault Systèmes, 2008.

Customer-Focused Strategy — Our Commitment to You

I am honored to be writing to you as the new CEO of SIMULIA. I believe this is an exciting time in our 30-year history. SIMULIA is well-positioned to increase the business value of the simulation technology that we deliver to you, and we believe that sustained business momentum is based on fundamental principles of technology innovation and customer satisfaction.



Scott Berkey
Chief Executive Officer,
SIMULIA

Our recent acquisition of Engineous Software allows us to complement the capabilities of Abaqus and accelerate the development of SLM. The professionals from Engineous bring expert knowledge in process integration and design exploration. The iSIGHT and FIPER products have been successfully deployed across multiple industries, and our customers are enthusiastic about the technical advances and business benefits they will gain from this new venture.

Our newest product is a direct result of the acquisition. Isight for Abaqus provides design exploration and optimization capabilities to users of Abaqus (see *INSIGHTS,* p. 11). Our development teams are also in the process of extending the functionality of SIMULIA SLM with iSIGHT and FIPER technology (see *INSIGHTS*, pp. 9-11). These developments, combined with the technology resources of Dassault Systèmes, are enabling us to strengthen our technology-leading position in Simulation Lifecycle Management.

These achievements would not be possible without you, our valued customers. The customer case studies in this issue of *INSIGHTS* from GN ReSound, Sunshine Heart, and Becton, Dickinson (BD) demonstrate that SIMULIA is focused on providing solutions for industries where the use of simulation is still emerging, as well as those where simulation usage is more mature. Our solutions are being used effectively in a range of industries beyond auto and aero, including civil engineering, electronics, energy, and medical devices. As Anita Bestelmeyer, CAE Manager at BD, points out: "When you use simulation for concept definition, and development of a product, you decrease product development costs and get to market quicker." This view makes it clear that the benefits of applying realistic simulation—throughout the product lifecycle—can be gained in all industries.

Even as we expand our portfolio to include Multiphysics and Simulation Lifecycle Management solutions, our focus on enhancing core Abaqus FEA technology continues. This is achievable due to customer loyalty, feedback, and collaboration. Our 2007 Customer Satisfaction Survey shows consistent results over the past three years for quality, support, and innovation. These results are testimony to our commitment of meeting your needs.

We continually strive to understand your business goals, engineering challenges, and technology requirements so that we can develop simulation solutions that meet real-world demands. For this reason, your participation at our international conference and Regional User Meetings is more valuable than ever. These gatherings are your opportunity to learn from your peers, gain a deeper understanding of our strategy, and share your requirements with our professional staff.

I would like to thank each of you personally for your commitment to making realistic simulation an integral part of your design, engineering, and research processes. Together, we are making a positive impact on the future of realistic simulation and innovative product development.



2007 CUSTOMER SATISFACTION SURVEY RESULTS

Compare SIMULIA to other vendors you are familiar with:



Industry Press Coverage











European Automotive Design
February 2008, p. 8
Software Tools Slash Door Latch Testing
The strength of a car door latch is critical to passenger safety during an automobile accident. Kiekert, one of the world's largest vehicle-latch manufacturers, has a virtual validation program that combines CATIA, Abaqus FEA, and in-house software to speed up latch design, testing, and development. Beginning with CAD models, Kiekert follows a process of FE modeling, multibody dynamic simulations, and tolerancing that has significantly reduced the number of physical tests for product design validation and cut time-to-market.

Offshore
May 2008, pp. 152–154
High-Performance Flexible Pipe Can Be Designed to Fit
As oil & gas companies move ever deeper offshore in search of energy resources, the need for flexible fiber reinforced pipe (FFRP) has become more pressing: FFRP is well-suited to withstand the greater hydrostatic pressures, wellhead pressures, and temperature extremes that accompany deepwater recovery. DeepFlex Inc., working with MMI Engineering Inc., uses Abaqus FEA models to tailor their composite pipe construction to multiple environmental variables. Abaqus FEA allows performance analysis of each layer of composite to meet exacting specifications for burst, collapse, axial extension, bending, and torsion.

Mechanical Engineering
May 2008, pp. 32–35
Offshore Analysis
Pelamis Wave Power was featured in this article about how FEA is being used to design energy-producing machines built to withstand the rigors of life in the ocean. Pelamis makes an energy converter that floats on the ocean surface and generates electricity from waves; a wave farm of multiple units can provide electric power for thousands of homes. Adapting Abaqus FEA applications to analyze both small components and larger assembly simulations, Pelamis is studying fatigue and durability in their product, with three real-world offshore projects underway in the U.K. and Europe.

Desktop Engineering
June 2008, pp. 46–47, 71
Abaqus Enables Full-Body N&V Simulation
Automakers need to rapidly assess noise and vibration (N&V) levels in the early stages of car design as they seek to increase the customer-pleasing aspects of a vehicle's ride. Abaqus FEA now makes it possible to simulate the N&V characteristics of a full automobile body—including tires—according to SIMULIA senior engineering specialist Charlie Chin in this bylined article. By using FEA to model the response of a vehicle to both structural and airborne input from the road and powertrain, engineers can improve the design and evaluation of new vehicles while minimizing physical prototype testing.

Mining Equipment and Supplier News
June 20, 2008, online
Realistic Simulation Accelerates Safety Evaluation of Mine Designs
Abaqus FEA software is being used to enhance the design and engineering of a number of large underground mines around the globe. Beck Arndt Engineering (BAE), working closely with SIMULIA engineers in Australia, is helping the world's largest miner, BHP Billiton, simulate a full, three-dimensional analysis of a mine's life cycle. The FEA models are used to study mining-induced seismicity, deformation, and collapse to ensure the safety of miners and achieve productivity objectives.

Product Design & Development
July 2008, Cover and pp. 38–39
Hearing Aid Design
The miniaturization of hearing aids has been a boon to consumers, but it has challenged designers in many new ways. The primary goal is to maximize amplification of sound, but the close proximity of very small, complex parts makes feedback an ongoing hazard. Until quite recently, physical testing of multiple prototypes was necessary to refine the qualities of an ideal hearing aid. But now, GN ReSound has both deepened and streamlined their product design process by using Abaqus FEA to simulate a number of complex tests and shorten the development time cycle (see *INSIGHTS,* p.6).

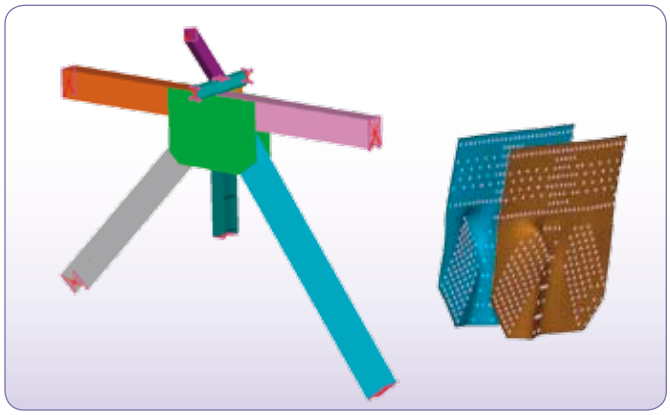
For More Information
simulia.com/news/media_coverage

To share your case study, send an e-mail with a brief description of your application to simulia.info@3ds.com.

SIMULIA Central Services Performs Nonlinear Analysis of I-35 Bridge Gusset

An interim report on the finite element modeling effort investigating the collapse of the Interstate 35 bridge in Minneapolis, Minnesota has been made public by the National Transportation Safety Board (NTSB). Supporting the NTSB's Modeling Group, the SIMULIA Central region services organization worked with the Group's members—the NTSB, State University of New York (SUNY) at Stony Brook, and the Federal Highway Administration (FHWA)—to perform nonlinear analyses on the bridge gusset of the U10W joint. The local gusset model is embedded into a larger global model of the bridge, provided by FHWA. The modeling is focused on predicting the stress distribution in the gussets of the U10W joint.

A link to the full interim report is available at: simulia.com/services/services_cust_references



Cambric Expands Solutions Offering with Abaqus Unified FEA

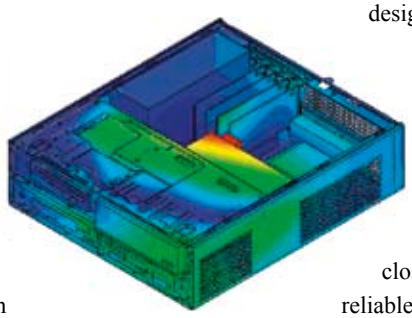
Global engineering services provider Cambric Corporation has selected Abaqus Unified FEA products to enhance their simulation service offering. Cambric provides engineering services to manufacturing companies in industries such as automotive, aerospace, heavy equipment, and consumer products. By implementing realistic simulation technology from SIMULIA, Cambric is enhancing collaboration with their customers and improving their process of evaluating accurate product performance.

"Our selection of Abaqus Unified FEA over competitive simulation products was based on better alignment with our customer's FEA tools, improved efficiency—both technically and financially—and SIMULIA's respected portfolio of nonlinear and multiphysics capabilities," stated Paul Spangler, Vice President, Cambric Corporation. "By leveraging SIMULIA's realistic simulation solutions, Cambric will be able to address a larger market for FEA services—by supporting our current customers, and developing new customers in emerging industry domains."

Spangler also noted that the selection of Abaqus Unified FEA was driven by scalability in terms of reduced solution times, broader parallel computing platform support, and SIMULIA's commitment to advancing their FEA technology.

Lenovo Uses Abaqus in New Simulation Technology Center

World-leading IT and Personal Computing (PC) company Lenovo is using Abaqus Unified FEA software in conjunction with the establishment of a new simulation technology center within their award-winning Innovation Design Center (IDC). The new center is using Abaqus FEA to evaluate realistic product performance during the design of Lenovo's innovative PC and portable electronics products.



"Reliability and quality are major challenges for information technology and personal electronic products," says Zhifeng Xin, manager of Lenovo IDC. "We are using Abaqus to perform design analysis on all of our products, including personal computers, notebooks, cell phones and servers. By partnering with SIMULIA to establish our new simulation center, we will deploy efficient analysis methods to accelerate the development of high-quality, market-winning products."

Lenovo uses Abaqus during the design process to simulate dynamic impact, heat transfer, vibration, fatigue, and other realistic performance characteristics of their products. Virtual tests help to identify appropriate design changes, if necessary, and to determine whether the product will meet the performance requirements.

Realistic simulation solutions from SIMULIA are enabling companies such as Lenovo to lower costs, reduce time-to-market, and improve product quality—resulting in higher levels of customer satisfaction. "At SIMULIA our goal is to work closely with our customers to help them make better, more reliable products in less time," states KC Jen, General Manager, Asia/Pacific, SIMULIA. "We are extremely pleased that Lenovo has chosen SIMULIA's technology for their Innovation Design Center. Our realistic simulation solutions are playing an increasingly important role in the fast-paced and innovative electronics industry."

Hearing Aid Design is a Resounding Success With Finite Element Analysis



The hearing aid has come a long way from bulky mechanical devices like 18th-century ear trumpets and speaking tubes. Early electrical hearing aids were based on telephone technology invented by Alexander Graham Bell in the late 19th century, but they used heavy batteries and provided limited sound amplification with poor-to-moderate fidelity.

Technology advances led to progressive miniaturization of devices during the 20th century so that today's hearing aids are extremely light and comfortable to wear—and even come in "cool" colors and styles. But designing such hi-tech hearing aids can be a challenge.

Imagine you are working in an office at an average noise level of 40 decibels (dB). Now put the loudest rock band in the world (about 130 dB) next door and try to carry on a normal conversation. Seems impossible. But that is essentially the problem that miniaturization has created for behind-the-ear (BTE) hearing aid design engineers.

The goal is gain without feedback

"When a person is wearing a hearing aid, there is a distance of only 2-3 mm between the microphone and the receiver (or loudspeaker) inside the device," says Morten Birkmose Søndergaard, Senior Acoustic Engineer at GN ReSound. "We are trying to produce up to a 90 dB gain (the difference between the office and the rock band) between the two without exceeding the feedback limit."

Going beyond the feedback limit results in the output from the receiver looping back into the microphone: the instrument will squeal (at about 100-145 dB, depending on receiver size and applied gain—not a pleasant sound level). This fundamental performance limit must be accounted for in every BTE hearing aid design.

The GN ReSound Group is one of the world's largest providers of hearing instruments and diagnostic audiological instrumentation. The Group is a part of GN Store Nord, founded in 1869 as a telegraph company, and now a global enterprise with over 4,500 employees. Sondergaard and his colleagues perform design analysis and testing in a high-tech acoustics laboratory at the company's corporate headquarters in Copenhagen, Denmark.

Just a few years ago, numerous hearing aid prototypes were physically tested in the lab, and modifications in their design and composition were made according to the results. But now the company's engineers have deepened, yet streamlined their testing—greatly reducing the number of prototypes they need to build, and significantly shortening the development time cycle—by adding finite element analysis (FEA) to their R&D arsenal.

GN ReSound's current test equipment includes a laser vibrometer to measure velocity on a vibrating surface, a 3D acoustic holography robot that measures radiated sound of the hearing instrument, general electro-acoustic measuring equipment, and a shaker/exciter to analyze different velocity/sound pressure stresses.

Looking inside the "black box" with finite element analysis

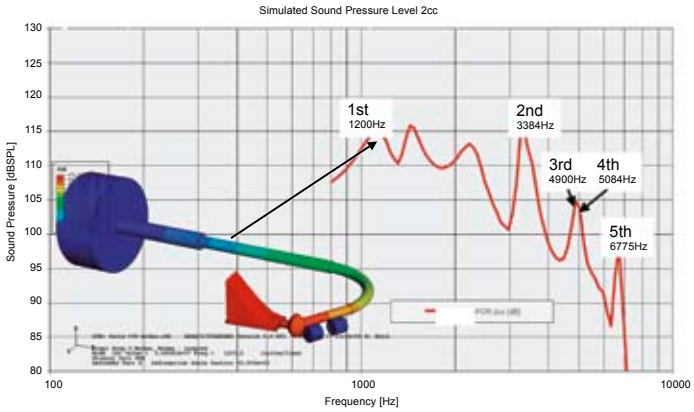
"Before simulation, we were limited to a trial-and-error approach for all our hearing aid design and testing," Søndergaard says. "We were essentially working with a 'black box' we could only measure from the outside to get information. Now, with simulation, we can look inside the black box and evaluate and alter its behavior."

GN ReSound uses Abaqus FEA software to ensure the stability of their device designs, improve hearing aid performance, and experiment with new materials and geometries.

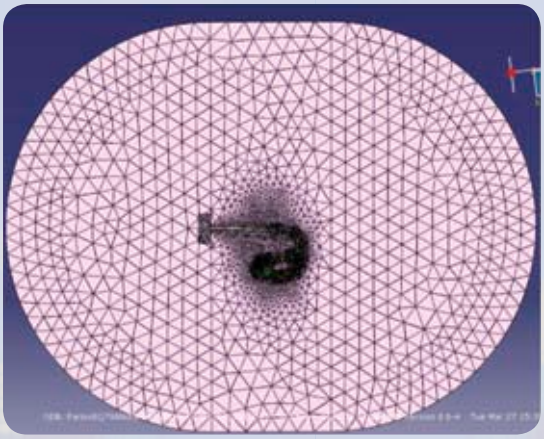
"We use Abaqus because the hearing aid feedback path consists of many parameters that can be difficult to assess accurately with traditional measuring equipment," Søndergaard says. "In some cases it is even impossible to measure or visualize certain vibro-acoustic behavior without FEA."

Modeling the hearing aid

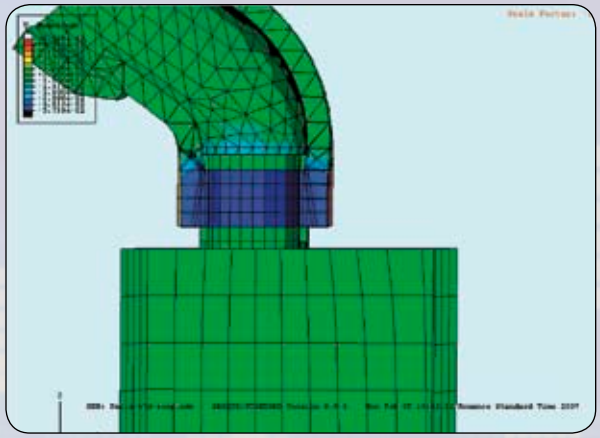
Abaqus software enables GN ReSound engineers to make computer models of all the critical elements of a hearing aid (a BTE hearing instrument connected to a PVC sound tube, along with a coupler that represents the ear canal). They run their



Modal analysis of a hearing instrument (at 1st through 5th modes) performed with Lanczos eigensolver, using Abaqus FEA structural-acoustic coupling where applicable, helps engineers determine areas of maximum vibratory stress (red) in the model.



FEA model of hearing instrument (center), with mesh representing the air both inside and surrounding the device, demonstrates Abaqus' multiphysics capabilities enabling structural/acoustic analysis.



FEA shrink-fit model of the receiver suspension and receiver shows the critical sound tube connection (blue) that is an important component in feedback control and sound transmission. Note the tetrahedral elements above and the quadrahedral ones below.

models through virtual vibration and sound pressure stresses that approximate real-world conditions, assess performance, and then validate the results with laboratory tests of actual units.

To model a hearing instrument, the engineers start with a simplified geometry of the device. They then use the Pro/ENGINEER Associative Interface to automatically transfer parts and assemblies into Abaqus/CAE, which enables the definition of model attributes, meshing and results visualization. The associative interface then allows for quick, automatic updates of designs in Pro/ENGINEER.

Within Abaqus, models of critical connections, such as that between the steel receiver housing and the rubber tube that goes over the receiver sound port, are a particular focus for simulation. The "shrink-fit" function in Abaqus is employed to model the important pre-tension in the part of the rubber tube that stretches over the underlying receiver sound port. Most models are composed primarily of tetrahedral elements, but other shapes are used where applicable. An average model has about 200,000 to 300,000 elements and 1 million degrees of freedom.

Acoustic resonance frequencies are of obvious importance in a sound-amplifying device: engineers study these using an FEA modal analysis, which incorporates both natural vibration frequencies and the specific vibration patterns of the structure being studied.

Another important challenge when modeling the structure of a hearing aid is to account for the air, both around and inside it, that conducts the sound—and then to analyze the interaction between the air and the unit itself. This is where the multiphysics capabilities within Abaqus come to the fore: a model of the hearing aid structure can be quickly and automatically coupled to the air, using surface-based tie constraints, without the need for matching meshes between the two.

Tough tests validate real-world results

Once a model is set up, GN ReSound engineers put it through its virtual paces. Using four Intel Xeon CPUs and averaging two runs overnight, they assess vibration velocity, sound pressure levels both inside and outside a hearing instrument, and acoustic holography in both two and three dimensions—enabling direct comparison with real-world data. The engineers also experiment with different types of rubber, plastic, and other materials to evaluate damping and stiffness.

Close agreement between FEA models and lab tests gives the engineers the confidence and design freedom to adjust components and materials in their models for high stability—no feedback or squeal—and maximum sound gain. "We now have a greater understanding of the causes of instability so we can eliminate them in the early design stages," says Søndergaard. "This leads to improved performance, and also faster development times. Today we're using FEA for all our hearing instrument products, evaluating geometry, materials, and performance. Once we have a working model, we can optimize it to make it even better before it goes into production. Then, using our existing simulation data, we can give our researchers advice and guidelines for developing future designs."

The prevalence of hearing impairment in the global population is estimated between 1.5 and 5 percent, and it can be unilateral, bilateral, slight, moderate, or severe. The cause can be congenital, or the result of accident, disease (viral or progressive), or drug toxicity. Total lack of hearing is actually rare, but when hearing loss occurs within the normal frequencies of human speech, it can create significant challenges at any age.

While implantable devices for the middle ear and cochlea have been developed for moderate-to-severe deafness, most cases of hearing loss can be ameliorated with externally-worn, behind-the-ear (BTE) hearing aids.

For More Information
www.gnresound.com

CZone for Abaqus

Simulating Crush in Energy-absorbing Laminated Composite Structures

Overview

Crushable structures that absorb energy during impact are used in automobiles, helicopters, aircraft, trains, and other transport vehicles to help protect occupants and cargo from shock and injury during a crash. Composite materials hold great potential for providing increased energy absorption in lower-weight crushable structures as compared to conventional, heavier metallic designs. A lack of commercial, industry-standard methods to simulate and accurately predict the crushing of these materials during impact has impeded the widespread application of composite materials in crushable structures.

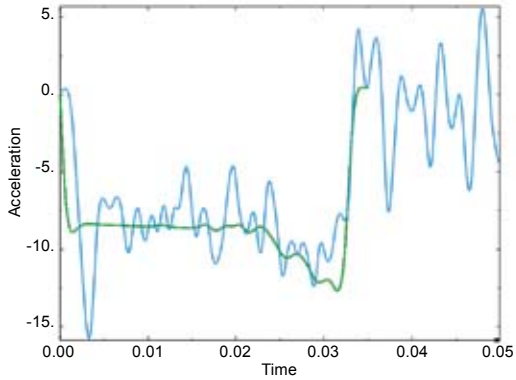
CZone for Abaqus is a new add-on capability to Abaqus/Explicit that provides access to a state-of-the-art methodology for crush simulation. Based on CZone technology from Engenuity Ltd., and targeted toward the design of composite components and assemblies in the Automotive and Aerospace industries, CZone for Abaqus provides for inclusion of material crush behavior in FEA simulations of composite structures subjected to impact.

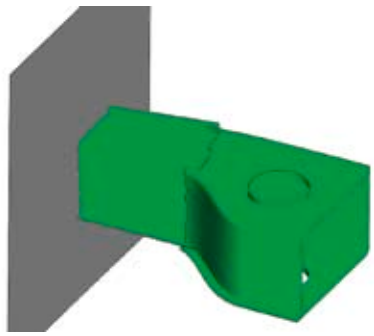
Features & Benefits

CZone technology provides direct implementation of crush-based element force generation and failure in defined "crush zones," typically located at the forward edges of the structure in direct contact with the impactor. CZone for Abaqus simulations determine the extent of material crushing and other modes of composite failure, the energy absorbed in the crush zone, and the forces generated by material crushing. The behavior of the composite structure outside the crush zone is simulated utilizing existing Abaqus capabilities to account for possible delamination, damage, fracture, and buckling. In this way, CZone for Abaqus "unlocks" the power of Abaqus to help design the back-up structure to properly support the crush zone.









A mass of 1150 kg. moving at an initial velocity of 9.1 m/s impacts a complex composite cone structure in an experimental sled test. Crushing of the cone progresses to a point where a large fracture develops suddenly in the transition region between the cone and its backup structure. CZone for Abaqus results predict well both the crushing response and the sudden fracture outside the crush front. Acceleration histories of the sled mass correlate well when comparing experimental data (blue) against simulation results (green).

With CZone for Abaqus results as a guide, a proposed design can be altered to optimize the placement, thickness, construction, and geometry of crush structures to maximize their energy-absorbing capacity. Crush properties for candidate materials can be obtained in a cost-effective manner from coupons cut from flat panel specimens. Such crush testing and calibration services are available directly from Engenuity; other test laboratories are also developing this capability. This information can also help in screening and selecting appropriate materials to use in a structure and evaluate whether candidate materials behave well or poorly during crushing.

Crash Simulation with Abaqus

As a foundation for integrating the CZone technology, Abaqus provides extensive capabilities to address crashworthiness and occupant safety simulation for the Automotive and Aerospace industries. This has been an ongoing strategic focus for several years, and Abaqus has been adopted as the primary tool for such design simulation at the OEM level.

CZone for Abaqus represents the next step in extending crash simulation capabilities with Abaqus to include the prediction of the crushing behavior of composite structures. It also complements existing Abaqus capabilities for composite failure analysis, including damage mechanics for material degradation and failure, VCCT for brittle delamination, cohesive element technology for failure in adhesively bonded regions, and specialized woven composite material models.

For More Information

Contact your local SIMULIA office or representative.

Abaqus for CATIA V5R19

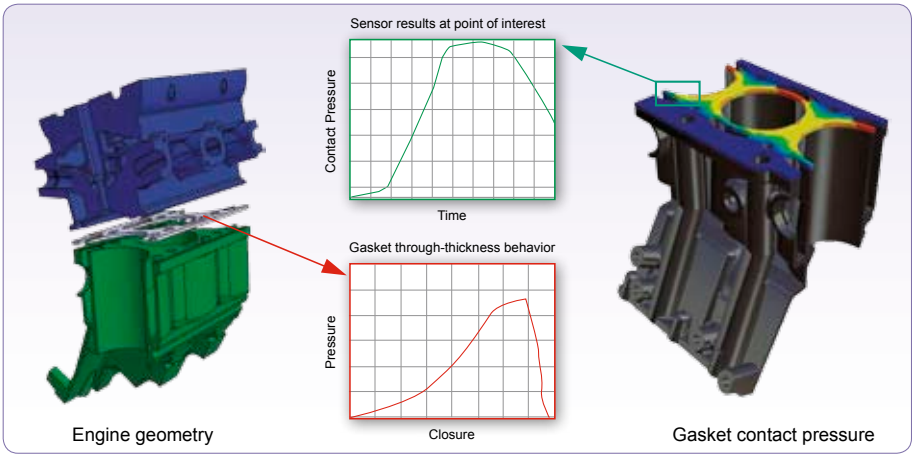
Deploying Proven Analysis Workflows Throughout the Enterprise

The latest release of Abaqus for CATIA V5 offers new and improved usability and design analysis capabilities directly in CATIA V5 that will help users accelerate their product development process.

New and enhanced capabilities include:

- Dedicated gasket elements to accurately model thin, flat gaskets
- Simulation continuation capabilities to enable a preloaded model to be used as the base state for subsequent simulation
- Interactive simulation diagnostic tools to enable enhanced troubleshooting

For More Information
simulia.com/products/afc_v5



In engine design, the head gasket's behavior must be modeled with great accuracy so that the sealing effectiveness can be evaluated following bolt-up and service loadings. Abaqus gasket elements now available in Abaqus for CATIA V5 are specifically designed for this type of simulation.

SIMULIA SLM V6R2009

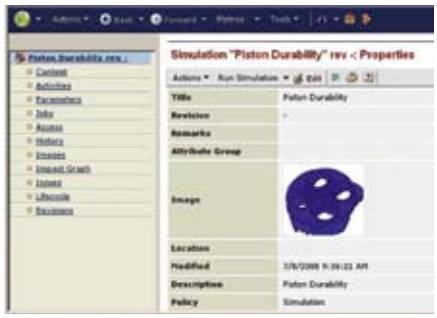
New Release Extends V6 Online Collaborative Platform for Managing and Securing Simulation Intellectual Property

SIMULIA SLM V6R2009 provides the ability to govern and manage simulation IP based on Dassault Systèmes' V6 platform, the online collaborative environment for PLM 2.0. The new release simplifies the capture, re-use, and deployment of approved simulation methods and best practices. It provides companies with the ability to improve simulation data quality and traceability, increase productivity, improve confidence in simulation results, accelerate distributed decision-making, and secure intellectual property.

SIMULIA SLM V6R2009 offers significant out-of-the-box functionality for rapid and economical deployment. The software includes secure online storage, search, and retrieval of simulation-specific data. It provides distributed and multidisciplinary teams with an environment for live collaboration and real-time decision-making based on the most current simulation results. SIMULIA SLM V6R2009 can be used as an integral part of a complete PLM system implementation or as an independent and scalable SLM system.

Highlights of SIMULIA SLM V6R2009:

- The open V6 platform provides proven technology that enables SIMULIA SLM to work with any simulation data, whether it originates from applications developed by other CAE vendors, customers, or Dassault Systèmes.
- Simulation templates capture and facilitate the rapid reuse and deployment of approved simulation processes and best practices to a range of users—from designers to expert analysts—who can then perform simulations with confidence and repeatability.
- Parameters can be created, edited, and used to update and manage critical variables related to a simulation. They can be used to represent the physical or performance attributes of the product being simulated, or the key variables needed in order to process and execute the simulation and its activities.
- The simulation job execution framework boosts performance by minimizing the network flow of large files.



This image shows the top level view of a Simulation collector for a piston durability simulation in SIMULIA SLM V6R2009. The tree structure at left indicates the collection of objects and capabilities managed by the Simulation collector.

- An application programming interface (API) provides customers with the ability to create custom user interfaces and vertical applications. This allows them to leverage the SIMULIA SLM capabilities for data management, job execution, collaboration, and simulation process automation.

For More Information
simulia.com/products/slm

Design Exploration, Components, and Process Integration Extend SIMULIA SLM

SIMULIA has completed the acquisition of Engineous Software, a market leader in process automation, integration, and optimization. The technology of iSIGHT and FIPER, along with the expertise of the Engineous professional staff, will extend and accelerate SIMULIA's leadership in delivering the industry's most robust and economically deployable Simulation Lifecycle Management solution.

The industry-proven process integration, automation, design exploration, and decision support functionality in iSIGHT and FIPER complement the technology-leading data management and collaboration capabilities in SIMULIA SLM to provide the most comprehensive suite of products available for Simulation Lifecycle Management. Our aggressive development plan to integrate and enhance these capabilities will ensure that customers gain even greater value in the near future.

iSIGHT is used at the desktop level and enables designers and engineers to integrate software applications. It also provides robust design exploration tools for performing Design of Experiments, Monte Carlo studies, or design optimization. The FIPER framework enables engineering processes to be captured and published as best practices across the enterprise using web-based interfaces. This allows engineering teams to collaborate on decision-making at a workgroup or enterprise level.

"We have enjoyed considerable success in deploying solutions from both Dassault Systèmes and Engineous as part of our overall digital engineering capability. The integration of Engineous and SIMULIA solutions within the open Dassault Systèmes environment represents an exciting opportunity for CAD/CAE system improvements, which we look forward to with great interest as our enterprise digital engineering strategy evolves. We are very pleased with this unification from our most dependable partners."

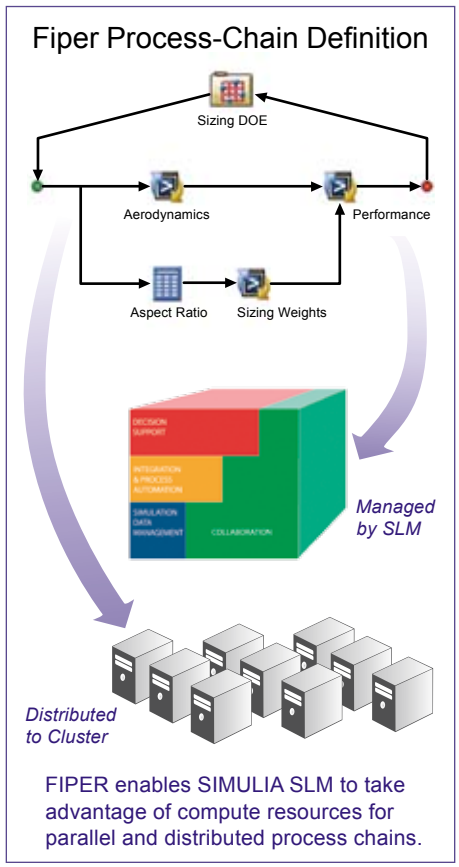
—Dr. Byungsik Kang, Director of CAE, Vehicle Technology Center of Hyundai-Kia Motors Corporate Research & Development Division

Extending SIMULIA SLM Capabilities

A robust Simulation Lifecycle Management solution must deliver a core set of capabilities for collaboration, data management, process automation, and decision support. The technology in iSIGHT and FIPER, together with our SIMULIA SLM solution, provides the following capabilities:

- Centralized simulation repository with full associativity to related data to enable collaboration
- Framework for standardization, process automation, and methods deployment to non-experts to increase reliability and repeatability
- Design exploration with Six Sigma principles for multi-disciplinary optimization and probabilistic analyses
- Scalable solution from workgroup to enterprise that leverages existing computing assets
- Open Architecture that allows the use of the simulation tools of your choice and integration with critical business systems
- Decision Support to provide deeper, intuitive insight to program and product status
- Total lifecycle management to enable full traceability and history assuring data quality, integrity, and reuse

The proven Product Lifecycle Management (PLM) technology from ENOVIA, our sister brand within Dassault Systèmes, provides the innovative data management and collaboration tools as the core foundation in the SIMULIA SLM solution. FIPER complements these capabilities by providing technology to enable engineers to create simulation processes and applications that can be executed from web browser interfaces. The underlying simulation data management functionality, combined with FIPER's process automation and design exploration capabilities, extends the benefits of Simulation Lifecycle Management from the workgroup to the wider enterprise.



FIPER enables SIMULIA SLM to take advantage of compute resources for parallel and distributed process chains.

Process Integration and Automation

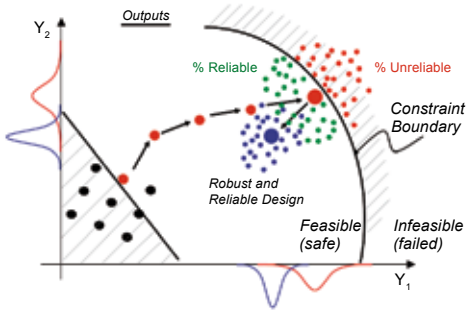
Using the open, component-based process integration framework in iSIGHT and FIPER, methods developers can create simulation process chains. This provides a powerful and flexible way to define the process being performed, its sequence, and the application and data set being used. This ensures that methods used to achieve simulation results can be captured, standardized, and repeated accurately.

Users are also able to integrate simulation process chains with business process workflows from ENOVIA. This provides confidence that the product performance

predictions are being calculated using the right product data—and the right performance specifications—at the right time in the product development process.

Simulation-based Decision Making

To maximize the impact of design exploration studies, iSIGHT and FIPER provide capabilities that enable users to add intelligence to an automated process, such as optimization for goal searching, Design of Experiments (DOE) for trade-off studies, and Monte Carlo analysis for uncertainty studies. These techniques accelerate design exploration and assist in effectively meeting performance objectives. SIMULIA SLM leverages these design exploration capabilities to enable local and distributed product development teams to share results and make performance-based decisions more effectively.



iSIGHT allows stochastic effects to be considered when designing real-world products using optimization.

Integrating iSIGHT and FIPER technology within the SIMULIA SLM portfolio provides the most robust environment for the integration, automation, and management of realistic simulation. The complete Simulation Lifecycle Management solution will enable our customers to develop and deploy approved simulation methods, perform more trade-off studies, ensure data traceability, and accelerate decision-making by providing a collaborative and secure simulation environment.

"SIMULIA SLM and the FIPER framework are key components in our future enterprise simulation strategy. This combination of people, technology, and vision is an exciting development for us and supports our vision for widespread deployment of realistic simulation throughout our organization, which will accelerate innovation and drive design decisions within P&G."

—Mike Telljohan, Director of PLM, Procter & Gamble

For More Information
simulia.com/products/slm

Isight for Abaqus

New Product Provides Design Exploration and Optimization to Abaqus Users

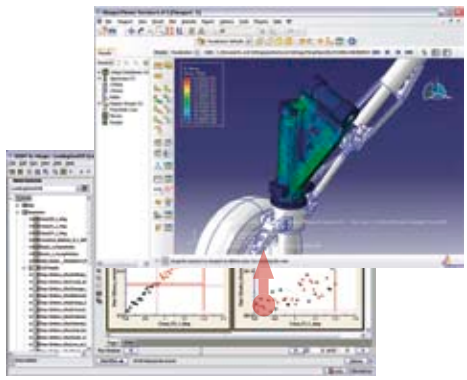
Isight for Abaqus is an add-on product for Abaqus FEA software that provides design exploration and optimization technology. This enables designers and engineers using Abaqus to perform rapid trade-off studies of real-world structural behavior—accelerating the development of innovative products in a variety of industries.

The add-on product allows a user to quickly explore thousands of design options in a well-understood space of competitive choices. It supports parallel submission of optimization, Monte Carlo, and Design of Experiments (DOE) jobs on multiprocessor machines or with third-party job scheduling software. An Isight for Abaqus user can collaborate with colleagues and partners to rapidly find the best design via an intuitive, interactive graphical user interface.

"The release of Isight for Abaqus marks a major milestone for SIMULIA as we leverage technology from recently acquired Engineous Software to expand and enhance our realistic simulation portfolio," said Steve Crowley, director of product management, SIMULIA. "With the rapid delivery of this new product, SIMULIA is bringing market-leading optimization technology directly to our Abaqus users. This will dramatically increase the number of design options they can evaluate to enhance their products' performance, reliability and quality, while reducing time and cost."

Highlights of Isight for Abaqus:

- A complete suite of DOE techniques to explore design space
- Gradient and genetic algorithms to optimize structural design performance
- Quality and Six Sigma methods take into account the manufacturing and operational variability in product design
- Multi-run scatter plots offer one-click visualization of the virtual prototype in Abaqus/Viewer
- Correlation maps show the influence of design variables on product performance
- Response Surface Method (RSM) and Radial Basis Function (RBF) approximations interactively trade off product performance



"As we have worked to accelerate our development of medical devices, Medrad has benefited from the simulation software and consulting expertise provided by SIMULIA. Now, the use of Isight for Abaqus to automate simulation exploration will enable us to perform multi-factor Design of Experiments analysis—painlessly."

—Ned Uber, Ph.D., Medrad Fellow, Medrad, Inc.

For More Information
simulia.com/products/i4a

The Role of Simulation in Product Development

An interview with Anita Bestelmeyer, CAE Manager, BD

BD is a leading global medical technology company that is dedicated to improving people's health throughout the world. BD develops, manufactures, and sells medical devices, instrument systems, and reagents. The Fortune 500 company, founded in 1897 and headquartered in Franklin Lakes, New Jersey, employs approximately 28,000 people in 50 countries throughout the world. In recent years, BD has boosted the pace of its R&D spending to focus on growth through innovation, and has begun implementing a company-wide product development system. The company's use of computer-aided engineering (CAE), as part of these efforts, is enabling it to accelerate the development and delivery of high-quality, robust products.



BD has been using Abaqus FEA software for over two decades. CAE Manager Anita Bestelmeyer joined the team 17 years ago and has been its head for five. She leads the simulation group in supporting product development and also works toward integration with BD's company-wide product development system, educating the customer base within the company about how her group can positively impact R&D efforts through simulation. Bestelmeyer recently talked to INSIGHTS about how collaboration and sharing of simulation results help drive design and business decisions at BD. A structural engineer with an aerospace background, Bestelmeyer finds the complexity of medical problems "a lot more interesting" than those in her previous field.

INSIGHTS: What are the major challenges in your industry right now, and how is BD evolving to meet them?

Bestelmeyer: There's a lot of competition to develop new, innovative medical applications using the latest technology advancements within the current economic environment. To maintain and grow BD's position, we have to ensure that our products are innovative and of the highest quality. To achieve this, it's critical to ensure robust performance despite inherent product variations, and maintain functionality throughout the product shelf life. We are also dedicated to protecting product users and healthcare workers through safety devices. At BD, we are constantly exploring ways to optimize product and instrument designs and reduce costs. Simulation has an important role to play in that process.

"When you use simulation for concept, definition, and development of a product, you decrease product development costs and get to market quicker."

—Anita Bestelmeyer, CAE Manager, BD

INSIGHTS: What are the key groups/ projects for which BD uses simulation?

Bestelmeyer: Our group receives the majority of requests for simulation (75%) from our BD Medical business segment, with the second largest number from our BD Diagnostics segment (20%) and the fewest from our BD Biosciences segment. These requests come from product development groups worldwide, although the largest number of requests comes from BD's headquarters in Franklin Lakes, New Jersey.

We are seeing an increased trend across BD toward using simulation earlier in the product development process as an integral part of product development instead of at the latter stages. We believe that this is due to ongoing education of our product development community about the value of simulation tools and our continued success stories, as well as the roll-out of Design for Six Sigma (DFSS) and the BD company-wide product development system process within the organization.

When you use simulation for concept, definition, and development of a product, you decrease product development costs and get to market quicker. Our BD units are increasingly internalizing this message and voluntarily coming to the simulation group more and more. Among the services we provide are tolerance stackup analysis, virtual design of experiments (DOE), and simulation of product shelf life to help identify the critical drivers that will ensure correct performance and robustness over time.

INSIGHTS: What types of simulations does your team perform, and what simulation challenges do you face?

Bestelmeyer: Our core BD CAE Analysis Services group performs simulations in three focus areas. The first and largest area is structural analysis to optimize part

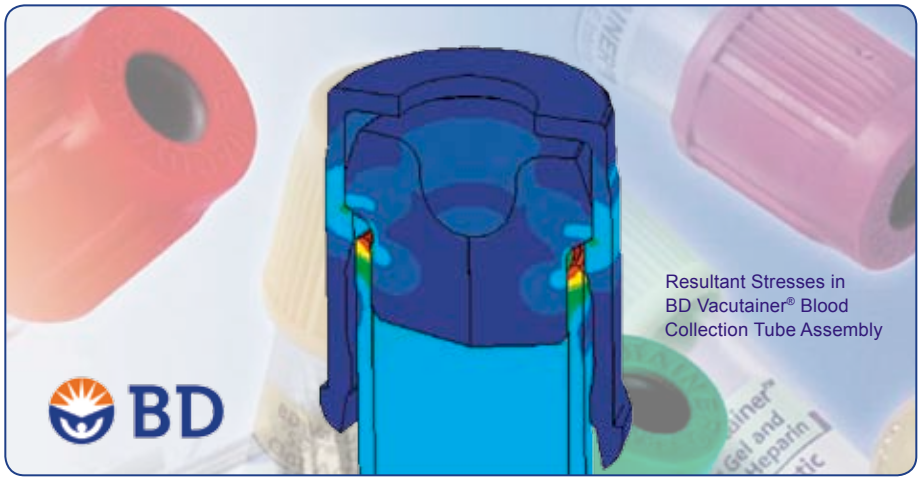
performance and structural integrity. The second area is injection molding simulation to predict part manufacturability and ensure the highest part quality.

The third area is computational fluid dynamics, where we can use our simulation capabilities to optimize fluid flow and product packaging; these are probably the most complex simulations we do. An example of this would be modeling fluid flow through an entire catheter, from the IV bag to the point where a drug is infused into the body.

Recently we have received requests that explore untapped or newly developed capabilities in the software codes. This involves frequent communication with our software partners to understand their product's features. We also collaborate with them to develop new capabilities, improve efficiency, or extend code functionality. For example, we are working with SIMULIA to investigate problems such as device interaction with biological tissue which require coupling of analysis capabilities for multiphysics simulation. These are challenging technology development initiatives for cutting-edge applications.

INSIGHTS: You've been using Abaqus for almost 20 years. How has the software evolved along with your technical needs? What are the main advantages of using it?

Bestelmeyer: The Abaqus software code has evolved quite significantly over the last 20 years; we are now performing difficult and complex simulations that would not have been possible previously. We also utilize Abaqus/CAE for setting up and postprocessing our structural analysis requests. The new functionalities in contact, material models, and the Abaqus/Explicit solver, among other improvements, have enabled us to simulate many more applications and grow the use of simulation within the organization. We believe that the long-term relationship is important, and we will continue to work together to develop additional capabilities that are of specific value to BD.



Resultant Stresses in BD Vacutainer® Blood Collection Tube Assembly

INSIGHTS: How do you currently manage your simulation processes, data, and workflows?

Our simulation requests have been managed through an IBM® Lotus Notes® CAE Jobs database for over 10 years, and we have recently upgraded this by launching a new web-enabled CAE Jobs database.

Our CAE Analysis Services group keeps all information associated with a simulation request in the database on the job profile form. Pertinent information regarding the simulation request, including related analysis approach and process, important data, and workflow, is all kept in one place. Our group uses this database for weekly meetings to discuss new requests and prioritize workflow to meet critical project timelines effectively.

BD's product development community can use the web-enabled database to upload their simulation requests and related files and also specify the simulation objective, material, and geometry, for example, to initiate a new request. BD associates worldwide have their own dashboard that provides a customized view of information regarding their simulation requests and those of their business unit. They can also search all of the BD requests to obtain information about past and ongoing requests to learn from previous experiences and encourage collaboration within the organization.

At the end of each month, a detailed summary of ongoing project work is automatically generated to update key BD R&D heads on the individual usage for their business unit and information on ongoing requests for the entire organization.

INSIGHTS: What is your future vision for managing simulation collaboration? In what ways does partnering with SIMULIA facilitate your work?

My future vision for managing simulation collaboration includes continuing to share and enable access to key initiatives of importance within the organization. Our CAE website has recent case studies that help BD users understand new ways to utilize simulation, and we hold ongoing information sessions to provide an overview of CAE capabilities, as well as understand changing R&D challenges.

As simulation is rolled out to more distributed BD sites, we plan to organize forums to share methodologies and best practices and provide technical guidance based on past experience and learning. We are also investigating the possibility of using Simulation Lifecycle Management or Product Lifecycle Management to accelerate the sharing of information and tying the simulation directly to the native solid model geometry.

Working with SIMULIA is important to our future vision since we use the Abaqus structural analysis code and are investigating multiphysics capabilities for actual BD applications. Our long history using the Abaqus code effectively and working with key SIMULIA representatives will enable us to develop simulation technology capabilities that address future BD challenges.

For More Information
www.bd.com

Rx for Accelerating Medical Device Innovation:

Realistic Simulation and Simulation Lifecycle Management

Subham Sett, Medical Industry Lead, SIMULIA Technical Marketing



Innovation in the medical industry is being driven by the need to provide improved healthcare options to patients—both those in emerging markets and those in the aging, yet physically active populations of developed countries. However, the variety of physiological and clinical conditions of patients and the complexity of the human anatomy pose considerable challenges to the design and performance evaluation of medical devices.

Material modeling for understanding human tissue response and in-vivo loading conditions is a critical tool for effective medical device development. In addition, the heavily regulated nature of the industry puts stringent demands on controlling the processes, quality, and reliability of the devices being manufactured. Physics-based computer modeling tools such as finite element analysis and computational fluid dynamics are now playing an increasingly important role in the product development process.

To meet industry demand for more realistic simulation in the development of medical devices, implants, and surgical procedures, SIMULIA is focusing on developing new capabilities and enhancements within the Abaqus Unified FEA product suite. With the recent acquisition of Engineous Software, we are also delivering a new add-on product to Abaqus to enable rapid and extensive design exploration. Isight for Abaqus allows a user to quickly compute thousands of design simulations with the software's support of parallel submission of Monte Carlo, DOE, and optimization jobs on multiprocessor machines or with Load Sharing Facility (LSF).

The realistic simulation capabilities in Abaqus include complex material models, contact, multiphysics (such as fluid-structure interaction), and parallel processing, among other powerful features. Abaqus also provides the ability to work with model data captured by CT Scans and MRIs. Such capabilities provide a robust environment for analyzing the physical behavior of mechanical systems in medical devices as well as in interaction with human tissue and bone.

Our customers are using Abaqus for a broad range of applications, including: the simulation of implanted devices (stents, pacemakers, and heart valves), orthopedics (knees, hip, and shoulder implants), drug delivery systems (syringes, auto-injectors, inhalers), diagnostics and monitoring tools,

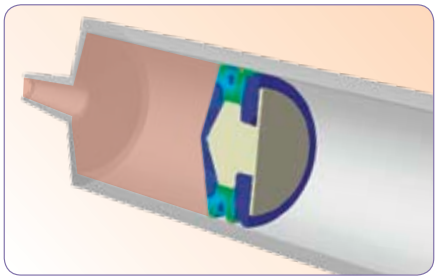
biomechanics (gait and motion simulation), and brain injury due to head impact. In all of these applications, the stakes are high for ensuring that the medical device functions correctly and reliably before being put to use in the patient population. Simulation is playing a key role in investigating the response of these products under realistic loading conditions.

Medical Application Examples

In the area of stents, the superelastic, shape memory, biocompatibility, and fatigue properties of nitinol, a nickel-titanium alloy, have made the material an attractive option. Abaqus can simulate the complete lifecycle of the stent and stent insertion equipment including manufacturing (laser cutting, annealing, insertion, and crimping), insertion (bending, torsion, and extension), expansion/deployment (lumen shape and diameter), and cycling. Simulating such processes reduces testing and time to market. The constitutive models for superelastic alloys are available as user subroutine libraries for both Abaqus/Standard and Abaqus/Explicit.

Pacemaker leads are used to carry electrical signals between the heart and the pacemaker. The system is a sophisticated assembly of conductors that carry current and multiple helical coils for torsional stiffness. These conductors and coils can be separated by sheaths of polymers. The assembly presents challenges in the form of complex contact conditions between the coils and the polymer sheath and goes through severe deformation of the coils and the polymer during the testing process. Abaqus provides the complete tool set to allow the engineer to assemble the entire pacemaker lead, use our robust contact capabilities, and create material models for polymers to investigate the areas of fracture and failure of leads and kinking of the polymer sheaths.

For orthopedics, medical innovators are developing new and improved artificial joints and implant processes. Joint materials can include stainless steel, titanium, cobalt-chrome, and ultra-high molecular weight polyethylene (UHMWPE). Artificial joints also have complex ranges of motion to mimic the flexibility, strength, and durability of natural joints. Abaqus provides a complete platform for simulating the kinematics of the joint assembly, mechanics including contact, durability, and wear of the joints, and possible failure modes.



During the design of a syringe, the fluid-structure interaction analysis capabilities in Abaqus can be used to evaluate possible fluid leakage around the seal due to various loading conditions.

The need for accurate and convenient drug delivery has led to the growing use of prefilled syringes. Patients are now using auto-injectors for illnesses beyond diabetes. Market demand has completely transformed product development requirements for syringes, which are a critical component of complex drug delivery systems. The disposable nature of these devices requires innovative products to be brought to market extremely quickly yet with an eye on product quality, safety and ease of use. While Abaqus has been traditionally used to look at the mechanical components of syringes, including safety features, recent advances in multiphysics technologies enabled with our CEL capability allow modeling of the operating characteristics of the syringes as well, such as leakage due to pressure buildup and fluid flow including non-Newtonian effects.

Managing and Protecting Intellectual Property

The medical device industry is one of the most heavily-regulated industries in the world. With the U.S. FDA and other regulatory agencies around the globe encouraging the use of modeling and simulation, Abaqus simulation data is being employed for more effective premarket notification [510(k)] or premarket approval (PMA). As medical device manufacturers place a stronger emphasis on using realistic simulation during the development process, it is expected that the volume of data, analysis methods, and intellectual property generated from simulation will increase dramatically. Medical companies need to be able to trace the history of their decision-making processes. SIMULIA is leveraging technology from Dassault Systèmes' new V6 platform and the ENOVIA brand to provide a complete solution for Simulation Lifecycle Management (SLM). SIMULIA SLM maximizes the value of company-generated IP through the capture, re-use, and deployment of simulation best practices for collaborative product development. The newest release of SIMULIA SLM delivers unique capabilities to integrate and control the execution of simulation applications, carry out operations such as query and version control, administer access privileges, and perform and review simulations in a distributed, collaborative environment that provides significant value to our medical customers.

Customer-focused Solutions

With significant advancements in our realistic simulation technology, as well as increasing simulation use within our medical-related customer base, it is clear that our strategy of providing robust nonlinear FEA and multiphysics solutions for the entire range of the medical device development process—plus the tools to manage and secure the resulting IP—is resonating strongly within the industry. It is our goal to meet with our customers regularly to understand industry requirements and deliver solutions that address their product development challenges today and in the future.



Subham Sett – *Medical Industry Lead*, SIMULIA

Subham is focused on developing simulation roadmaps for the biomedical industry as well as SIMULIA's multiphysics and fluid-structure interaction solutions. He began his career at SIMULIA as an engineering specialist. Prior to joining SIMULIA he was involved in the design of MEMS switches at Coventor Inc. and holds several patents in the area. Subham holds an M.S. in Mechanical Engineering from the University of Colorado, Boulder and a BTECH from the Indian Institute of Technology, Kharagpur.

For More Information
simulia.com/solutions/life_sciences



The Beat Goes On

Sunshine Heart Optimizes Unique Pump Device Design with Abaqus FEA from SIMULIA

Heart failure is a debilitating, progressive disease characterized by the organ's inability to provide sufficient blood flow to the body. Some five million U.S. patients are currently suffering from heart failure (HF), with 500,000 new cases diagnosed each year. HF can result from coronary artery disease, heart attack, high blood pressure, diabetes, heart muscle infection, lung disease or valve disorders. Symptoms, which can become life-threatening, include difficulty breathing, swelling limbs, weight gain, and lack of energy and stamina.

Treatment for HF can range from drugs to defibrillators to internal heart pumps, with transplant as the final option. No single therapy works for everyone, and side effects and mechanical issues can arise for the implanted pump devices. Dr. William Peters, a cardiothoracic surgeon and research fellow at Auckland City Hospital in New Zealand, thinks there has to be a better way.

"I've always had a strong interest in devices to support the failing heart," he says (he has also invented a commercially-successful minimally-invasive bypass system). "Because of concerns about existing technologies, I was looking for a device that would not involve contact with the blood." Common implanted blood-contacting devices such as left-ventricular assist devices (LVADs), while lifesavers for people awaiting transplants, require that the patient remain on blood thinners (which themselves can be a stroke risk) to prevent clots. Reliability has also been an issue with some heart-assist device designs.

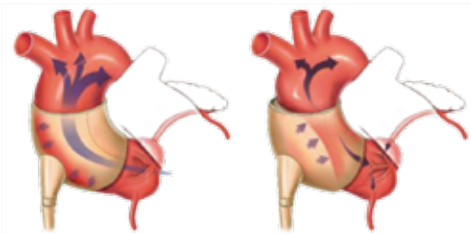


Image 1: Sunshine Heart's novel C-Pulse heart pump design consists of a cuff that encircles the aorta, inflating and deflating to enhance blood flow and decrease the heart's workload.

Novel pump works from outside the heart

Dr. Peters conceived of a novel idea for a pump system that works inside the body but outside the bloodstream, called the C-Pulse™. It consists of a cuff that wraps around the aorta (the main blood vessel that carries oxygenated blood from the heart to the rest of the body) and inflates and deflates a membrane (balloon) against the vessel's external walls (see Image 1). The positive and negative pressure of the balloon make the aorta pulsate in time with the heart, augmenting blood flow through the circulatory system, thus reducing total work and strain on the entire heart. A battery-powered pump worn outside the body powers the device (see Image 2).

Peters patented his pump idea and formed a company, Sunshine Heart, to develop and test the device, initially on the bench and in sheep. But once animal trials were successful, when the balloon was ready to be scaled up to a human model, the company decided that they needed a more sophisticated approach to the design and development process than the empirical, build-and-test approach. The goal was not only to reduce lead time, but to provide a level of confidence that long-term performance would satisfy product requirements established by physicians for an acceptable medical device.

FEA optimizes fatigue performance

"The average human heart rate of 80 beats a minute equates to 42 million inflation cycles a year," says Scott Miller, manager of mechanical engineering at Sunshine Heart. "The accumulated stress, especially on a polymer, was the design challenge—and C-Pulse is essentially a permanent implant. To ensure that our physical design solution was optimized to give us the long term fatigue performance required, we decided to look at it from a computational perspective using finite element analysis (FEA)."

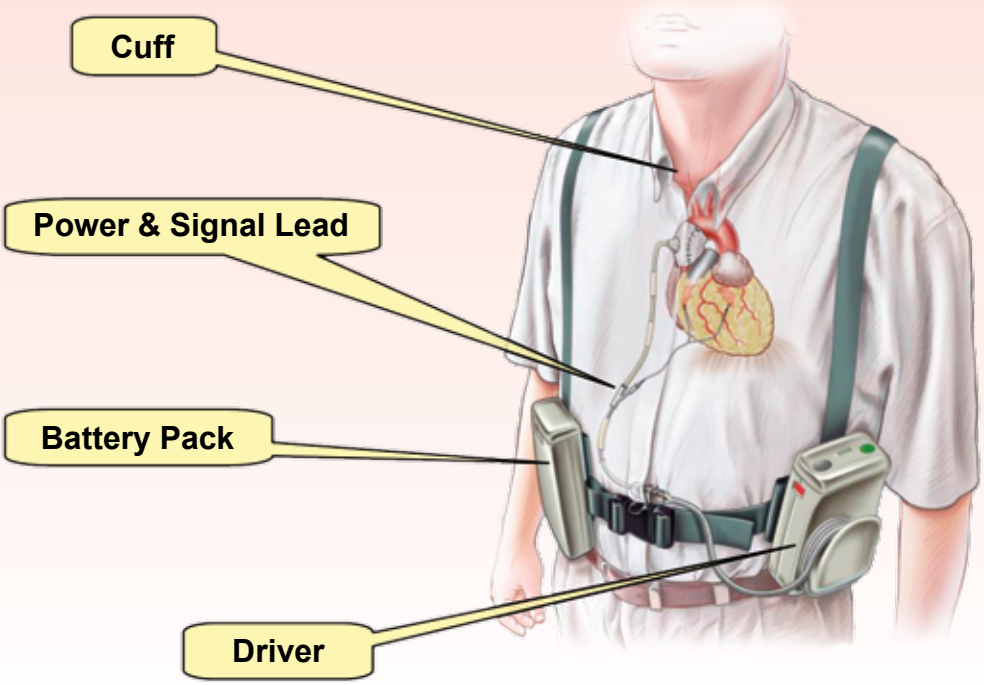


Image 2: Patient wearing the C-Pulse System. A lead from the external power source [Driver] connects to a catheter inside the body attached to the implanted device [Cuff], which is wrapped around the exterior of the heart's ascending aorta.

Miller and his product development team worked with Matrix Applied Computing Ltd for technical engineering software services. Matrix used Abaqus/Standard software from SIMULIA to model the behavior of the C-Pulse cuff and balloon interacting with the aorta.

"The FEA analysis was an iterative process that required some very unique approaches because of the way our device worked, the materials we were using, and how the device is actually assembled," says Miller. The balloon had to be easy to manipulate during implant surgery; conform to the shape of the aorta; have the strength and flexibility to "snap through" from concave to convex and back again repeatedly; compress the artery; and perform reliably from initial inflation through years of use—all within a very limited space. The goal of the FEA modeling was to accurately represent the real-world behavior of the device in order to guide design decisions and optimize the C-Pulse's performance through every stage of this process.

Element and material choices are critical

As a starting point for the FEA analysis, Sunshine heart provided Matrix with concave and convex Pro/E models of the device (see Image 3). According to Don Campbell, Principal Engineering Analyst for Matrix, "It was an interesting challenge. Our analysis involved modeling hyperelastic material; a fabric membrane; simplified biological material for the aorta; contact, large strain; and a staged assembly process."

To determine what kinds of elements (the geometric shapes mathematically representing physical units that make up an FEA mesh) to use for modeling the artery, cuff and balloon, Matrix created a series of test models. Quadrilateral shell elements turned out to be acceptable for the bulk of the parametric design studies (including determining the all-important optimum thickness of the balloon). But for modeling surface strains affecting the balloon in the fillet radius region (a critically important area where failures of the very earliest designs had occurred), hexahedron solid brick elements were chosen for more precise results using substructuring techniques with results from the shell model driving the solid element analysis (see Image 4).

The material modeling portion of the analysis was constrained by physiology and anatomy studies that had already been conducted. "We were given pre-existing data for the biocompatible material (a polymer approved for medical device applications) from which the device would be manufactured," says Campbell. "The Ogden hyperelastic material model in Abaqus provided an excellent fit with the experimental data." The Ogden model is often used to model rubberlike materials such as polymers, and biological materials.

(Story continued on page 18)



Image 3: Pro/Engineer Geometry of C-Pulse unit on which Matrix's Abaqus FEA models were based.

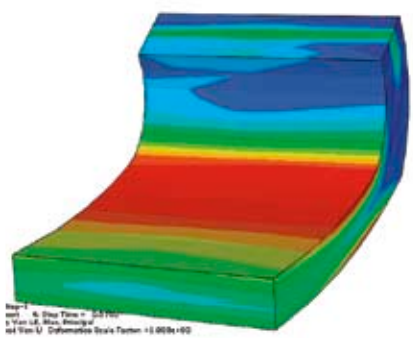


Image 4: Abaqus FEA submodel solution showing variation of strain through thickness of balloon wall.

Modeling the "snap through" function

With the FEA models of the C-Pulse set up, Matrix ran simulations to determine what shape the device's balloon should be during surgical implantation (starting with a convex configuration turned out to be most effective at minimizing strain). Next they simulated the complete balloon "snap through" motion of convex to concave and back again (see Image 5). "The complexity of the analysis was less in its geometric difficulty or problem size, but more in the simulation of the continuous, alternating process," says Campbell. "The strain on the balloon varied from the outer to the inner surface of the material as it snapped through, so the total strain we were analyzing was a combination of stretching and bending. During the simulation cycle, the location of peak strain in the fillet actually moved from the minor to the major axis of the oval-shaped balloon."

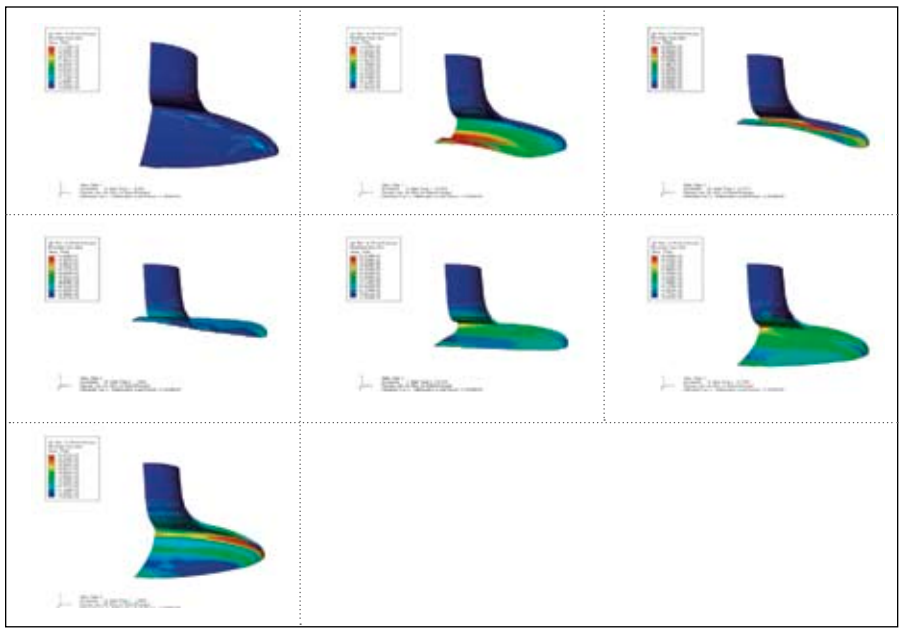


Image 5: This series of images shows an Abaqus FEA strain analysis of the "snap through" of a C-Pulse balloon membrane. Note how the area of maximum strain (red) moves from the short axis (upper right images) to the long axis of the oval balloon (final image at bottom) from start to end of the cycle.

Matrix ran its simulations as quarter, not full, models, using the assumption of symmetry to cut down on processing time and aid solution convergence (see Image 6). "There were some approximations with the quarter model since an aorta is not a straight pipe, but has some curvature," Campbell says. "However, for the purpose of optimizing the design, the lack of true quarter symmetry was thought to have a minimal effect on the ultimate design parameters. This approach also let us perform a large number of parametric runs in a reasonable amount of time."

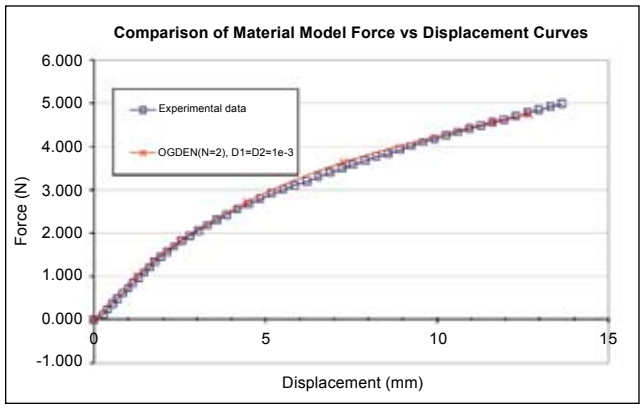


Image 7: Comparison of force deflection curves for hyperelastic material model of the C-Pulse balloon membrane (red is FEA prediction, blue is actual test results) shows how well the model performed in the test environment.

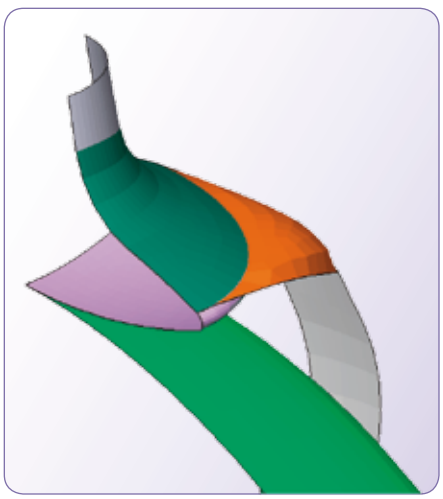


Image 6: FEA quarter model of balloon (lavender) within cuff (greys & orange) pressed up against aorta wall (green). The assumption of symmetry in the model allowed for decreased model size and shorter run times.

The ultimate goal of the FEA analysis was to arrive at a device shape which had the least variation of strain amplitude and the maximum mean compressive strain during an operational cycle. Says Campbell, "It was a project with interesting physics and the final model we came up with has performed very well in the test environment (see Image 7)."

FEA provides final design solution

The FEA models more than met Sunshine Heart's requirements. "We arrived at a design solution the first time through." says Miller. His group has subsequently proven that the solution holds true for different sizes, allowing for tailoring the device to individual patients.

And the durability of the C-Pulse design is being borne out by ongoing testing, Miller notes. "We have been running devices day and night literally for years now: the test machine requires regular maintenance because the C-Pulse keeps wearing the test unit out."



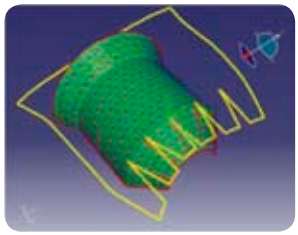
Scott Miller, M.E. is Manager of Mechanical Engineering at Sunshine Heart. U.S.-born and educated with an M.E. degree from Clarkson University, he is now an Australian citizen. He was one of the first employees of the company, which was founded in 2000.

For More Information
www.sunshineheart.com

Simulayt Provides Enhanced Composite Modeling and Simulation with Composites Modeler for Abaqus/CAE (CMA)

Composites Modeler for Abaqus/CAE (CMA) complements and extends the powerful new ply modeling features in Abaqus/CAE by providing proven fiber simulation capabilities and advanced model building—all seamlessly integrated within Abaqus/CAE. Furthermore, with CMA, composite models and ply layups can be shared between Abaqus and CATIA V5.

Composites Modeler for Abaqus/CAE provides draping simulation that computes continuously varying fiber orientations and ply thicknesses. This ply-level data is fed directly to Abaqus/CAE for detailed



structural analysis, ensuring simulations of unprecedented fidelity.

Building composite models with CMA also ensures that unmanufacturable plies cannot be specified—right at the beginning of the process—avoiding costly reengineering late in the development cycle. Along with providing the draped ply data to Abaqus for simulation, flat patterns of each ply are easily exported as 2D data files for manufacturing.

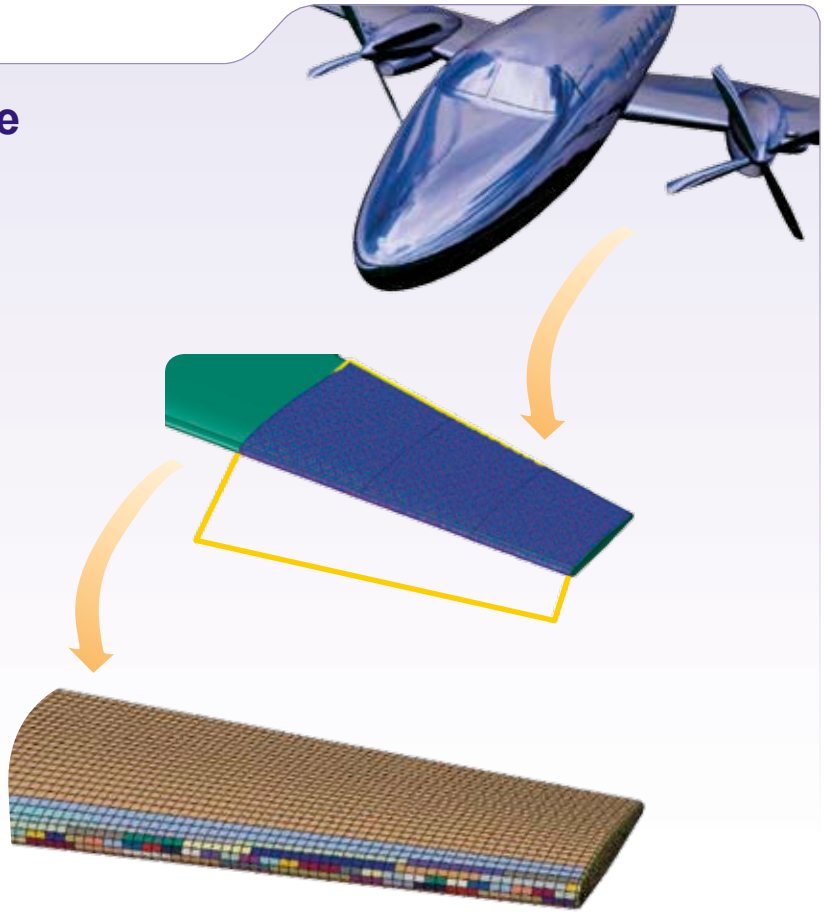
Composites Modeler for Abaqus/CAE is a partner product developed by UK-based Simulayt Limited. Simulayt, a leader in fiber simulation since 1992, also develops the Advanced Fiber Modeler for CATIA V5 (AFM).

Composites Modeler for Abaqus/CAE is now being directly sold through your local SIMULIA sales office.

CMA Application Example

Aerospace engineers can use CMA to understand the resultant orientations of composite plies after layup on an aircraft wing. When a ply layer is defined, CMA will calculate the draped thicknesses and orientations for that ply. The flat pattern of the material is also calculated and visually displayed to the user. CMA will then create new definitions for element properties based on the draped plies.

In the lower right-hand image, the elements of the FE model are color-coded by different section properties. The many different colors at the wing's leading edge highlight the fact that when a single piece of composite fabric is draped over the wing, the orientations of the ply change as it goes around the edge. This change in orientations is accurately captured by CMA and automatically assigned to the underlying FE model.



For More Information
Contact your local SIMULIA office or representative.

Western Michigan University Concrete Canoe Team Stays Afloat with Abaqus

Western Michigan University's student chapter of the American Society of Civil Engineering (ASCE) recently used Abaqus to enhance their competitiveness in the 2008 North Central Regional Conference of the National Concrete Canoe Competition. The competition is designed to provide civil engineering students an opportunity to gain hands-on project management experience and leadership skills by working with concrete mix designs and computer-aided engineering tools. It has challenged the knowledge, creativity, and stamina of more than 400 teams and 5000 students throughout its 20-year history.

The Western Michigan University team's entry, a concrete canoe dubbed *Meridian*, was modeled and analyzed in Abaqus under hydrostatic pressure load. In a departure from previous competitions, the students built finite element models of various canoe configurations and performed structural analysis to identify critical stress zones and optimize the hull thickness of their canoe.

After studying several configurations, the students established Meridian's length at 18.5 feet. This length was shorter compared to previous years' entries, and was selected to improve maneuverability and lower the overall weight. The length was calculated by allotting 3.5 feet of space per rower, with 2.25 feet of bow and stern to be left unused. The maximum width of the canoe was determined to be 28 inches, with a depth of 16 inches and a hull thickness of 0.875 inches.



Figure 3: *Meridian* (left) just after removing the forms and (right) during finishing

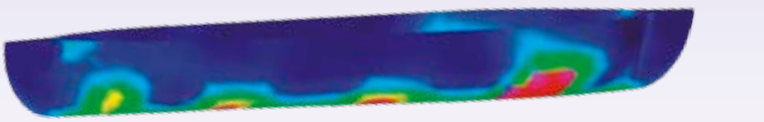


Figure 1: Stress contours



Figure 2: Displacement contours



Figure 4: *Meridian* in the race

Matt Czachowski developed the finite element model, which contained 809 shell elements, and then used Abaqus to run the analysis under hydrostatic pressure with the guidance of Dr. Upul Attanayake, Assistant Professor of Civil and Construction Engineering. The estimated total weight of the rowers and the canoe was 1,050 pounds, and the racing depth was 8 inches. The highest level of stress was located on the hull, where the paddlers' knees were placed (Figure 1). The largest of these was located in the bulge of the canoe—toward the stern—and was of a magnitude of 80 psi tensile stress. The magnitude of the deflection was not controlling with estimated modulus of light weight concrete of 300 ksi. Figure 2 illustrates the displacement contours.

After careful consideration of structural behavior and the level of stress under expected hydrostatic pressure loads, the students decided to make four structural ribs, located approximately 4 inches behind each paddler's knees, for structural reinforcement. The ribs were designed to dissipate tension in the composite concrete layers along the hull, and also to act as cantilevered supports that would resist deflection and bending in the hull (Figure 3). Finally, *Meridian* was built with a concrete mix that had an average density of 56.1 pcf, compressive strength of 700 psi, and tensile strength of 250 psi. The weight of the canoe was 252.5 pounds.

The Western Michigan University Concrete Canoe Team took third place in the women's endurance race, as well as fourth place in all the remaining events (Figure 4). They plan to continue leveraging Abaqus to further improve their canoe design for future competitions.

For More Information
www.wmich.edu/engineer/student-projects

New FEA Technology in Abaqus Student Edition 6.8

The latest academic edition of Abaqus provides engineering students with access to the same state-of-the-art technology included in the professional version of the product (limited to 1,000-node problems). Features in the new release help teach prospective engineers how to apply realistic simulation to industrial applications.

Abaqus Student Edition 6.8 features fully built-in and improved composites analysis capabilities that can assist aerospace engineering students in the study of behavior of composite crack propagation, delamination, and possible failure.

Automotive engineering students can benefit from industry-unique capabilities that allow the analysis of vehicle noise and vibration response due to tire rolling effects and viscoelastic material effects from tires, bushings, isolators, and laminated steel.

A low-cycle fatigue method in Abaqus Student Edition 6.8 can help prospective



electronics engineers assess the lifecycle of solder joints. This method is also useful to students studying automotive powertrain durability evaluation, or bone degradation in biomechanical applications.

Future medical device developers can learn to simulate soft tissue interaction of stents and orthopedic implants using a new anisotropic hyperelastic material model. This model can also be used for the study of materials such as reinforced rubber and wood.

A new Coupled Eulerian-Lagrangian (CEL) multiphysics capability allows prospective mechanical engineers to predict loads on earth-moving industrial equipment during soil excavation. The capability can also be used to predict the behavior of fluid-filled containers, hydroplaning tires, and bird strike on aircraft.

"Having Abaqus Student Edition has really improved the quality of my course. It has been excellent for the students to have Abaqus on their home computers. Using the Teaching and Student Edition versions together has been cost-effective for training undergraduate students in basic FEA concepts."

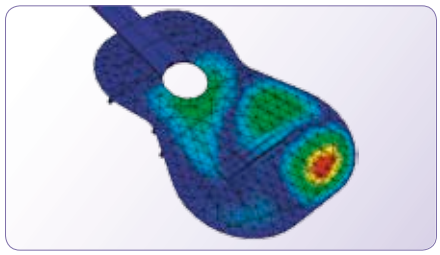
—Bill Klug, Assistant Professor of Mechanical and Aerospace Engineering, UCLA

For More Information
simulia.com/academics

Flexible Licensing for Abaqus Academic Editions

Experience with robust FEA software such as Abaqus provides today's engineering students with a strong, competitive edge at graduation. SIMULIA offers Abaqus academic editions that are specifically designed to fill the broad spectrum of requirements demanded by today's engineering educators. For over two decades, Abaqus FEA has been used at leading institutions around the world.

Abaqus Research Edition has the same functionality as commercial versions, enabling students in master's and doctoral programs to analyze and solve realistic engineering problems. Dhananjay Panchagade, of Auburn University, used Abaqus Research Edition to study the transient dynamics of printed circuit boards during drop impact. Dhananjay used ultra high-speed video to capture the deformation kinematics of the circuit board assembly and validated Abaqus predictions with experimental data.



Juan Fernandez used Abaqus Student Edition to study the vibration and acoustic effect of asymmetrical and symmetrical guitar designs.

Abaqus Teaching Editions complement the Research Edition by allowing students to run Abaqus/Standard, Abaqus/Explicit, and Abaqus/CAE in a classroom setting. A number of professors who are currently using Abaqus Teaching editions in their courses have created tutorials, which are an invaluable resource for demonstrating FEA concepts and helping students learn to use Abaqus programs. SIMULIA provides these tutorials—such as "Creating a Model for an Overhead Hoist with Abaqus/CAE," and "Large Deformation Analysis of a Beam-Plate in Bending"—on simulia.com to assist professors in developing their own course materials.

Abaqus Student Edition is a low-cost, personal finite element analysis tool for solving limited size problems (1,000 nodes) not requiring user subroutines. This edition is the ideal tool for students who wish to become more proficient with Abaqus. Juan F. Fernandez of Laboratoire ESE, Université Paris 11, used Abaqus Student Edition to evaluate natural vibration modes in simulated symmetrical and asymmetrical guitar soundboards. Juan's results indicated that asymmetric designs do improve sound production compared to symmetrical designs, as asymmetry increases the participation factor on the vertical axis of the soundboard.

The three Abaqus academic editions provide a flexible way for teachers, students, and researchers to collaborate by using common Realistic Simulation technology and methods.

For More Information
simulia.com/academics

Accelerating Simulation with High-performance Computing

High-performance computing has become surprisingly cost-effective and no longer requires extensive IT resources. In fact, a compute cluster provides an affordable high-performance computing system for running Abaqus FEA. The Abaqus solver technologies have been enhanced to include additional analysis features, reduced memory usage, and improved performance and scalability. A compute cluster typically has a fast switch connecting the computers, which enables parallel processing of single Abaqus jobs across multiple computers. You can cut your analysis runtimes dramatically by running on multiple cores.

Consider the following scenario: If you have an engineering group with four Abaqus users, you may be using 20 analysis tokens to allow each of your engineers to run a single CPU job on their workstations. Let's assume they are each running an analysis that takes eight hours to complete on their single CPU workstation. If all four engineers submit their jobs at 8:00 am, they will have to wait until the end of the day or the next morning to review their results, change the model, or make design recommendations. With a simple 16-core HPC solution, the eight-hour run could be done in less than one hour. This means that all four engineers could submit their jobs at 8:00 am and have results to review just after lunch. In fact, the first job submitted would actually be complete by 9:00 am. Changes to the model could be made and the modified job could be placed into the job queue by 9:30 am. With an HPC solution, your engineers can submit multiple analysis jobs and have all of their results in the same day.

Many of our technology partners specialize in setting up compute clusters. Additionally, SIMULIA regional offices can also provide services related to implementing a HPC system. Depending on your requirements, a compute cluster can be set up and operational in less then a week. Abaqus/CAE for pre- and postprocessing can operate on the individual desktop machines, while the actual analysis jobs can take advantage of parallel processing to achieve much faster runtimes.

With Abaqus 6.8, you are able to choose between running a cluster with Linux or with Windows HPC Server 2008. Windows HPC Server will appeal to IT managers who want to maintain a 100% Windows environment, while providing high performance computing resources to their Abaqus users.

Cluster Computing FAQ:

Q: What release of Abaqus is required?

A: To take advantage of the parallel scalability improvements, use Abaqus 6.7 or 6.8. For Windows HPC Server 2008, Abaqus 6.8-3 or newer is required.

Q: Do I need Infiniband?

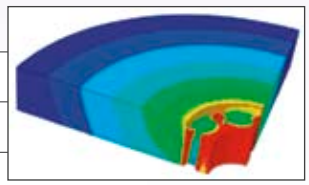
A: With a cluster of 4 to 6 nodes, you can experience good performance and scaling using a Gigabit Ethernet switch. For clusters with 8 or more nodes, Infiniband is recommended.

Q: How many Abaqus tokens are required?

A: The number of tokens required goes up with the number of cores used. The increase in token cost is typically less than the runtime speedup, making HPC a compelling proposition in terms of efficiency and effectiveness when taking into account all costs and benefits over time.

Q: What is a typical node in a compute cluster configuration?

A: 2P/Dual Core Nodes with 8 GB of RAM or 2P/Quad Core Nodes with 16 GB of RAM.



The graph at left shows the execution time and token hour savings for the S2A benchmark of a nonlinear static analysis of a flywheel with centrifugal loading in Abaqus/Standard.

For More Information
simulia.com/support/v68/v68_performance



SIMULIA Customer Conference – Call for Papers

May 18–21, 2009 — the Brewery — London, England

SIMULIA is pleased to announce the expansion of our traditional Abaqus Users' Conference (AUC) to include users of iSIGHT and FIPER. To support this expansion at our 2009 event at the Brewery in London, England, the AUC will formally become the SIMULIA Customer Conference (SCC).

The 2009 SCC will continue the 20-year tradition of providing a valuable forum for learning how experts in engineering and academia are applying the latest simulation technology and methods to accelerate and improve product development. The SCC will bring together an international community of realistic simulation users to share their knowledge and experience in advancing methods and technology for finite element analysis, multiphysics, process automation, design optimization, and simulation management.

Call for Papers

DATE	MATERIALS DUE
November 6, 2008	Final date for Abstracts and Permission Forms
January 23, 2009	Draft Manuscript
February 27, 2009	Final Manuscript and Registration for Conference
May 1, 2009	Final PowerPoint Presentation

We invite all of our worldwide customers to submit abstracts about their use of any of our solutions—including Abaqus FEA, Multiphysics, SIMULIA SLM, iSIGHT, and FIPER—for consideration for presenting at the inaugural SCC 2009.

Benefits of Presenting

- Enhance your professional credentials by being published in the Conference Proceedings, which is distributed internationally.
- Receive a $100 discount on your registration fee to the conference and a $100 discount on the Advanced Seminar fee.
- Gain recognition as an expert in your field in the simulation community.
- Establish beneficial industry contacts.

For More Information
simulia.com/events/call4papers

2008 RUM Schedule

Attend the upcoming Regional User's Meeting in your area. Learn about the latest enhancements to our products and the ongoing strategy of SIMULIA. For additional information on the Regional Users' Meeting in your area, visit your local office website.

Americas

Date	Location
November 6–7	San Diego, CA
November 13	Plymouth, MI
November 14	Baltimore, MD

Europe/Middle East/South Africa

Date	Location
November 4–6	Milano, Italy
November 6–7	Istanbul, Turkey
November 10–11	Linz, Austria
November 13–14	Antwerpen, Belgium
November 13–14	Madrid, Spain
November 20	Paris, France
November 20–21	Poznan, Poland
November 25	Herzelia, Israel

Asia Pacific

Date	Location
November 6	Beijing, China
November 10–11	Bangkok, Thailand

For More Information
simulia.com/events/rums

SIMULIA Helps Me Breathe.



Simulation for the Real World

People rely on quality care and innovative medical devices to maintain and enhance their well-being. Our customers in the medical industry use SIMULIA solutions to understand and improve everything from operating procedures and implants to hearing aids and inhalers. We partner with our customers to deploy realistic simulation methods and technology that helps them drive innovation and ensure device reliability—so everyone can breathe a little easier.

SIMULIA is the Dassault Systèmes Brand for Realistic Simulation. We provide the Abaqus product suite for Unified Finite Element Analysis, Multiphysics solutions for insight into challenging engineering problems, and SIMULIA SLM for managing simulation data, processes, and intellectual property.

Learn more at: **www.simulia.com**

www.3ds.com

SIMULIA

The 3DS logo, SIMULIA, and Abaqus are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries.
Other company, product, and service names may be trademarks or service marks of their respective owners.
Copyright Dassault Systèmes, 2008.