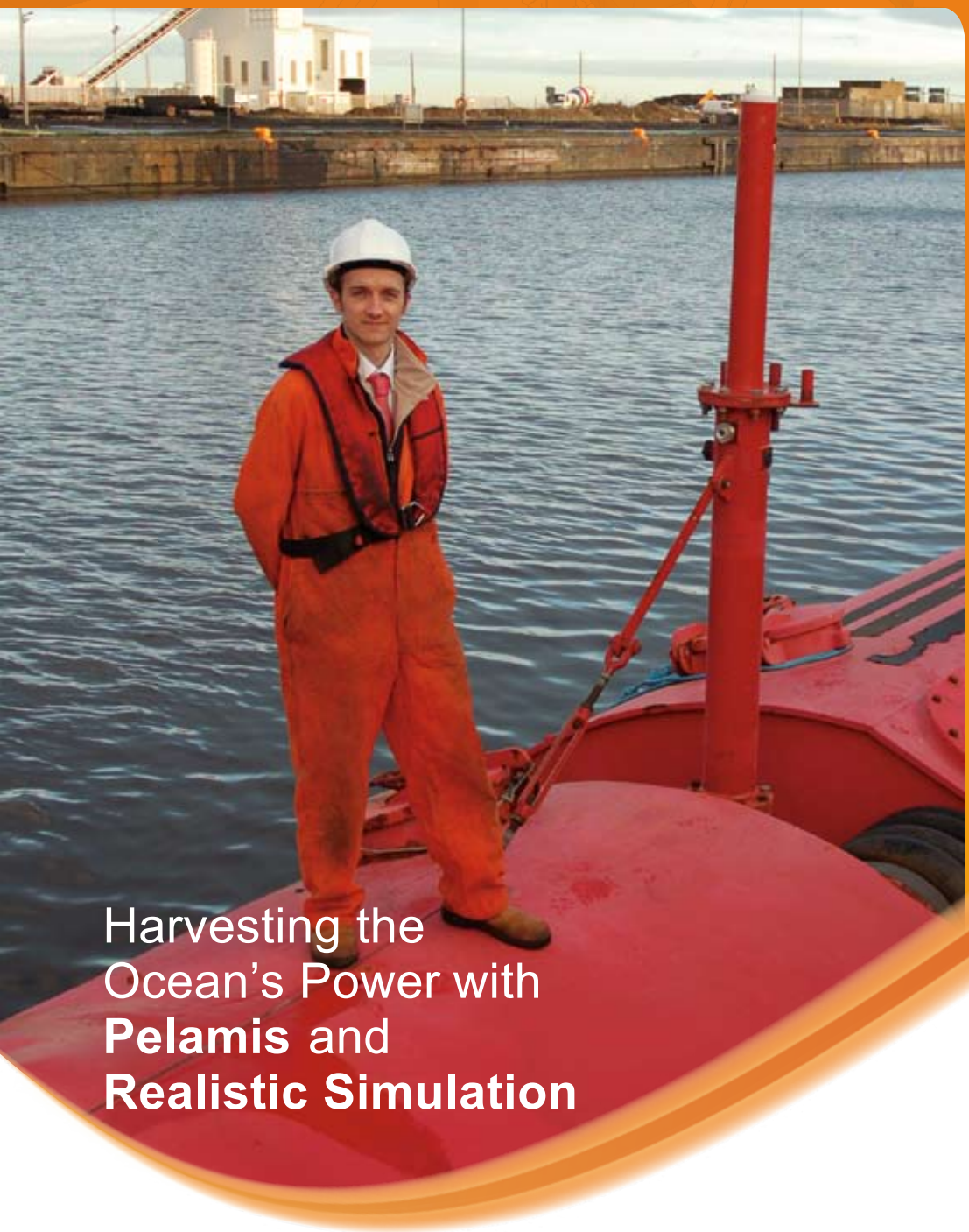
INSIGHTS

3
02 2008
Dassault Systèmes Realistic Simulation Magazine



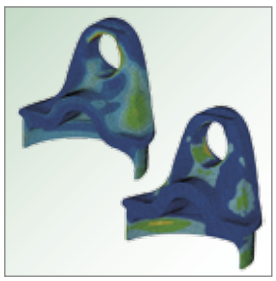
Harvesting the Ocean's Power with **Pelamis** and **Realistic Simulation**



New Multiphysics Capability
Coupled Eulerian-Lagrangian in Abaqus Version 6.7-EF



Building Better Bridges
Penn State University



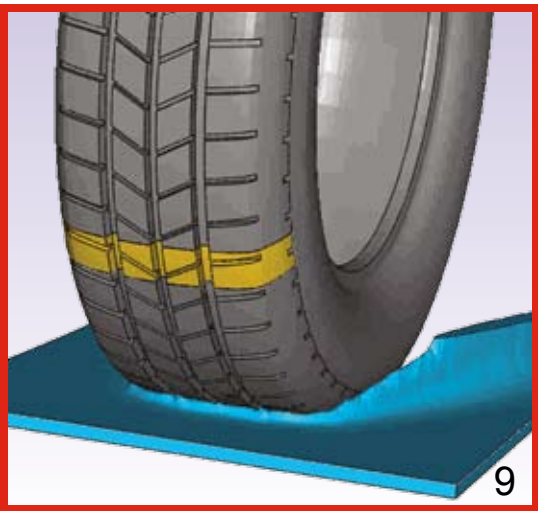
Fatigue Life Analysis with fe-safe™ 5.3

SIMULIA

INSIGHTS

January/February 2008







Inside This Issue

10 Cover Story

Harvesting the Ocean's Power with Pelamis and Realistic Simulation

On the cover: Jon Benzie, Sr. Engineer, Pelamis Wave Power

8-9 Product Update

- Abaqus Version 6.7-EF
- Coupled Eulerian-Lagrangian (CEL) Multiphysics in Abaqus

6 Customer Spotlight

Improving Bridge Performance with Finite Element Analysis Software - Penn State University

In Each Issue

3 Executive Message

Ken Short, Vice President, Strategy and Marketing

4 In The News

- Industry Press Coverage
- MicroSat Systems, Inc.
- Apollo Tyres Ltd.
- Society of Fire Protection Engineers Recognizes Kevin LaMalva

12 Product Update

- Simulation Lifecycle Management
- New Release—SIMULIA SLM

14 Customer Case Study

Building Blast Simulation and Progressive Collapse Analysis - University of Florida

16 Academic Update

- Mississippi State University Wins Challenge X Competition
- Latest FEA Technology Available in Abaqus Student Edition

17 Alliances Update

- Advanced Fatigue Life Analysis using fe-safe 5.3 and Abaqus
- Coupling AcuSolve with Abaqus to Analyze Subsea Pipelines

18 Events

- A New Year's Resolution: Make Training an Integral Part of Your Development Plan
- 2008 Abaqus Users' Conference

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
info@3ds.com

www.simulia.com

Editor:
Tim Webb

Associate Editor:
Julie Ring

Contributors:
Jon Benzie (Pelamis Wave Power), Fernando Carranza, Jeff Cipolla, Ted Krauthammer (Univ. of Florida), Paul Lalor, Daniel Linzell (Penn State Univ.), Parker Group, Subham Sett, Ken Short

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.

Evolving Trends in Realistic Simulation and Simulation Lifecycle Management



Ken Short
Vice President, Strategy and Marketing,
SIMULIA

In 2002, we embarked upon an ambitious strategy toward a unification of finite element analysis. At that time, we had a long way to go in providing competitive functionality in key areas that Abaqus was not particularly known for, such as automatic contact for explicit analysis, good scalability on distributed memory parallel computing platforms, large-scale linear dynamics, automotive occupant safety, and other capabilities, which meant that Abaqus could not be considered as a unified FEA solution.

Since then, we have established strategic roadmaps, worked hard, and invested a tremendous amount of development time and resources so that now—for many applications, especially in the aerospace and automotive industry—Abaqus is a real alternative to using multiple finite element programs.

There has been a definite increase in market acceptance of Unified FEA. Many of our customers are consolidating their software tools around Abaqus in order to reduce translation of data between applications, unify the approach to simulation, and study multiple performance attributes from essentially the same model. A unified approach also helps improve communication and collaboration between teams by incorporating approved simulation methods and sharing of simulation data and results.

Another trend I've noticed is the expanding use of nonlinear analysis technology and multiphysics simulation within the overall simulation user community. Engineers who have previously relied on a mostly linear approach to simulation are realizing that to get closer to simulating real-world behavior, they need more sophisticated and robust simulation technology like Abaqus. The migration of these types of users from their legacy software to the Abaqus user community is significant, as we have added dozens of new customers per month in 2007—most of whom were previously using other commercial FEA software products.

The use of Abaqus and other simulation technologies has proven to be an effective way for companies to drive innovation, evaluate product performance, ensure safety, and gain insight into real-world behavior more efficiently than testing physical prototypes. However, the intellectual property (IP) generated in this activity is often simply "lost," or at least not captured, and therefore cannot be reused. In fact, companies are spending millions on performing simulation but wasting much of the generated value. This has led to a growing industry acknowledgement of the need for Simulation Lifecycle Management solutions, and I expect this trend to accelerate in 2008 and beyond.

Companies are now beginning to understand that managing the data, methods, and design decisions that are an integral part of the simulation process will become a key competitive advantage. In other words, they are now ready to manage simulation within a defined lifecycle—starting from conceptual simulation methods for product concepts through to re-using simulation data, know-how and simulation-driven design decisions in subsequent product development cycles—or when an existing product requires additional simulations due to refurbishment, recycling, or disposal. Thus, the lifecycle of simulation IP exists in its own right, as well as being a key component of the overall product lifecycle.

In January this year, we announced the initial release of SIMULIA SLM, which is the first new product line to carry the SIMULIA brand name (see *INSIGHTS* article on p. 13). We, in the SIMULIA team, are very excited about this new product, since it is a great example of the synergy brought about by being a part of Dassault Systèmes. By combining the talents and technology of the development team that has brought you Abaqus over many years with the proven technology and platform of ENOVIA, we have been able to develop and bring to market rapidly the first real SLM solution that is suited to the needs of users of both Abaqus and other simulation software products.

I invite you to attend the 2008 Abaqus Users' Conference in May in Newport, Rhode Island. We have received a record number of abstracts from our customers, so we are expecting a varied and high-quality program of contributed papers. You will also learn about the latest enhancements to Abaqus, our new SLM product suite, and our vision for the future. I am continually motivated by the challenge of meeting your high expectations for our existing products as well as expanding the SIMULIA portfolio by bringing new products to market with the same commitment to quality, innovation, and customer support that you have grown accustomed to. I hope to see you in Newport, where you can let me know how we are doing and how we can improve.



Industry Press Coverage











Machine Design
October 11, 2007, pp. 109–115
Couplings for Multiphysics
Improved software technology and rising hardware capacity now allow the simulation of many real-world problems through multiphysics, reports SIMULIA Engineering Specialist Subham Sett in this bylined article. Using the example of a peristaltic pump, which has both industrial and medical uses, Sett takes the reader step-by-step through an Abaqus fluid-structure interaction (FSI) analysis using code-coupling (both direct and indirect) between FEA and CFD solvers. He also discusses the new Abaqus/Explicit coupled Eulerian-Langragian (CEL) method (see *INSIGHTS* article on p. 9), applicable to more highly dynamic FSI problems such as tire hydroplaning or automotive airbag inflation.

Aerospace Engineering
October 2007, pp. 24–25
Assessing the Strength of Z-pinned Composites
Creating Abaqus FEA models of Z-pinned composites to assess delamination during strength tests saves time and expense over physical testing, reports Kyle Indermuehle. Results from software simulations correlate strongly with physical tests performed by Adtech Systems Research in cooperation with a U.S. Air Force Research Lab program. Z-pinned composites are increasingly being used in aerospace for load-bearing parts. Delamination is one of the main failure modes of composites, and FEA testing helps fine-tune assessment of the margins of safety for such materials.

R&D
October 2007, pp. 16–17
Improving Bridge Performance with Finite Element Analysis
Higher-order analysis using Abaqus FEA software can help engineers ensure the reliability, maintenance, and economic viability of the bridges of the future, according to Pennsylvania State University Associate Professor of Civil and Environmental Engineering Daniel Linzell (see *INSIGHTS* article on pp. 6–7). He and his graduate students create FEA models that incorporate nonlinearities to account for realistic stresses and deformations that will influence the service life of a bridge. SIMULIA Senior Engineer Deepak Datye notes that Abaqus is also used to evaluate damaged or collapsed structures forensically.

Appliance
November 2007, p. 38
Real-World Design
Editor Lisa Bonnema interviewed SIMULIA Director of Technical Marketing Dale Berry for an in-depth article about how engineers can use the new Abaqus multiphysics capabilities to combine their existing FEA and CFD tools to solve problems in appliance product development. Applications include medical devices, ink-jet printers, and virtually any home appliance that regulates fluids by moving structures. SIMULIA Engineering Specialist Subham Sett created images of complex FSI interactions in a front-loaded washing machine specifically for this article.

Desktop Engineering
November 2007, pp. 14–15
Designing Better Running Shoes
Leading sports shoe manufacturer adidas achieved significant design cycle reduction for its new line of running shoes by coupling surface-based MCAD with Abaqus FEA. Mechanical engineer Tim Robinson reported from the company's German design facility that simulations sped up the entire development process from many weeks to just a few days. Data sources included adidas' confidential material libraries and biomechanical input from film of runners in motion. Strain-contour plots generated with Abaqus/CAE let engineers "see" inside an actual shoe to evaluate loading on a unique sliding plate technology in the heel that absorbs shock in three directions.

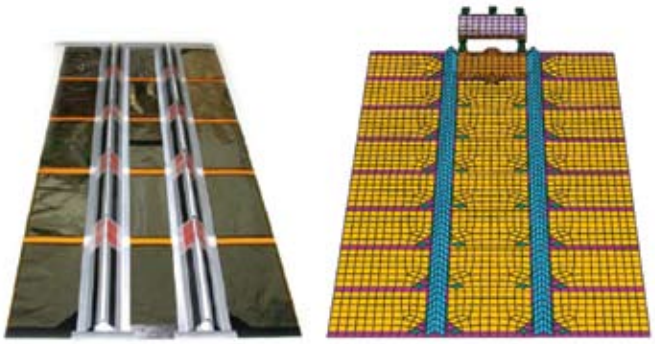
For More Information
simulia.com/news/media_coverage

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

Accelerating Realistic Performance Evaluation of Lightweight Satellites

MicroSat Systems, Inc., a fast-growing provider of high-performance satellites based in Colorado, USA, is accelerating the evaluation of product performance and reliability with Abaqus Unified Finite Element Analysis (FEA) software.

"With more commercial and government programs needing small satellites capable of carrying larger payloads, it is critical to our success to be able to deliver flexible satellite solutions that meet their requirements in a shorter amount of time," said Todd J. Mosher, Ph.D., director, advanced systems, MicroSat Systems, Inc. "We selected Abaqus FEA software due to its reputation for sophisticated nonlinear simulation including pre-loads, mechanisms, and thin-shell aerospace applications."



MicroSat Systems is using Abaqus Unified FEA software to analyze the structural and thermal response of their satellite systems, which consist of a modular bus structure; lightweight and foldable, thin-film, solar array systems; and miniaturized avionics. Realistic simulation solutions are enabling MicroSat Systems to develop a competitive product line of satellite buses that provide more payload, power, data processing, and pointing accuracy.

Apollo Tyres Meets Design Goals Efficiently with Abaqus

Tire development requires the evaluation of a complex combination of rubber, steel, and other layered materials. Leading Indian tire manufacturer Apollo Tyres Ltd. is using Abaqus Unified Finite Element Analysis (FEA) software to evaluate alternative concepts, guide design decisions, and refine product performance.

According to P.K. Mohamed, chief of technology and R&D at Apollo Tyres, Abaqus software is helping the company achieve design and performance goals efficiently. "Realistic simulation reduces design cycle time and cost, while enabling our engineers to evaluate design properties and optimize product performance," stated Mohamed. "We selected Abaqus because it is a world-class design analysis product with specific functionalities for tire analysis."



Apollo Tyres develops a wide range of passenger and commercial vehicle tires including the Aspire (pictured here).

Apollo Tyres has implemented Abaqus Unified FEA as an integral tool in the development of all categories of their radial tires. The software provides a unified simulation environment to accelerate the evaluation of tire durability, tread wear, vehicle handling, noise and vibration, and rolling resistance. Optimizing tire performance for these attributes plays a crucial role in improving vehicle safety, reliability, passenger comfort, and fuel economy.

Society of Fire Protection Engineers Recognizes Kevin LaMalva

Established in 1980, the Educational and Scientific Foundation of the Society of Fire Protection Engineers (SFPE) fosters scientific research and education relative to improving the safety of life and property from the hazards and damages of fire. In 2007, the Foundation honored Kevin LaMalva, a former student at Worcester Polytechnic Institute, with the Student Scholar Award at their annual conference in Las Vegas, NV for his failure analysis of World Trade Center 5 (WTC 5). Kevin's award-winning research was conducted using Abaqus to analyze the internal structural collapse that occurred in WTC 5 due to fire exposure alone on September 11, 2001.

Based upon forensic evidence, the research hypothesized that the steel column-tree assembly failed during the heating phase of the fire. A sequentially-coupled thermal stress model was created with Abaqus that is capable of predicting the structural behaviors that occurred and led to the internal collapse of WTC 5. The failure predicted with the Abaqus model confirmed the hypothesis and is very similar to a collected structural specimen from WTC 5 that underwent failure.



Kevin LaMalva (right) receives the 2007 SFPE Student Scholar Award from Doug Brandes of the SFPE.

Today, Kevin is an Engineer at Simpson Gumpertz & Heger Inc. (SGH) with a focus in structural fire protection engineering. Kevin has submitted his research for publication at the 2008 Abaqus Users' Conference. He and his colleagues at SGH also recently contributed details of the research for a new Abaqus Tech Brief, entitled *Failure Analysis of World Trade Center 5,* which is available for download at the SIMULIA website.

For More Information
simulia.com/techbriefs



View from beneath a new bridge under construction, showing pre-stressed concrete beams and deck at the abutments. The weight of the bridge itself, as well as traffic, wind, water, temperature, corrosion, and time, all contribute stresses and deformations that influence the performance and service life of the structure.

Improving **Bridge Performance** with Finite Element Analysis Software

Historically, most large bridges are "overdesigned" with substantial margins of safety built in to compensate for unknown forces that could affect their integrity over time. For the reliability, maintenance, and economic viability of the bridges of the future, better performance from the ground up is critical.



Associate Professor Daniel Linzell and his research group in the department of Civil and Environmental Engineering at Penn State University employ advanced Finite Element Analysis (FEA) to create computer models for studying the structural behavior of bridges. Using this technology, Linzell and his graduate students are able to focus in on potential trouble spots in individual bridges, helping civil engineers anticipate problems and make adjustments before construction begins. The simulation results can also be used to make decisions about maintenance requirements.

Modeling Real-World Stresses

"Civil engineering has until very recently relied on linearly elastic, small deflection FEA methods used in software tools as the backbone of bridge analysis. However, this method is, in many instances, an approximation and doesn't capture the full range of real-world nonlinear responses in these increasingly larger and more complicated structures," says Linzell. "Higher-order methods found in advanced FEA software are becoming more commonplace in the industry because they provide the capability to incorporate nonlinearities to account for realistic stresses and deformations that will influence the performance and service life of a bridge."

Aspects that can be incorporated into an advanced FEA model in addition to material and geometric nonlinearities to assess the structural integrity of a bridge include the response of concrete or steel to the weight of the bridge itself, as well as to traffic, wind, water, temperature fluctuation, corrosion, and even time (both concrete and steel "creep," resulting in long-term deformation).

Economic Factors

Economics is another driving force behind the need for more sophisticated analysis tools, Linzell says. "Extreme overdesigning has become too expensive. There's now a strong push to minimize material costs and simplify design to reduce labor."

Lighter, stronger materials are being developed: steel that is available now has yield stresses of 100 ksi (100,000 psi), almost three times what it was just 10–15 years ago. But while stronger steel allows builders to use smaller sections to support

the same bridge loads, the new materials may also be more flexible. "You need higher-order tools to better predict such nonlinear geometric deformation," Linzell points out.

Helping the Bridge Building Industry Communicate

Linzell's group is concerned about a long tradition of separation between designers and constructors in the bridge industry. "Designers need to understand that a bridge doesn't arrive in one piece and get dropped into place; on-site construction decisions can influence bridge life and functionality. You have to think about how it's going to be built, because that in turn influences how it's going to perform in the end," he says.

"What we're trying to do is come up with guidelines or tools that will bring designers and constructors closer together. Those are the intentions with what we're doing with our FEA models."

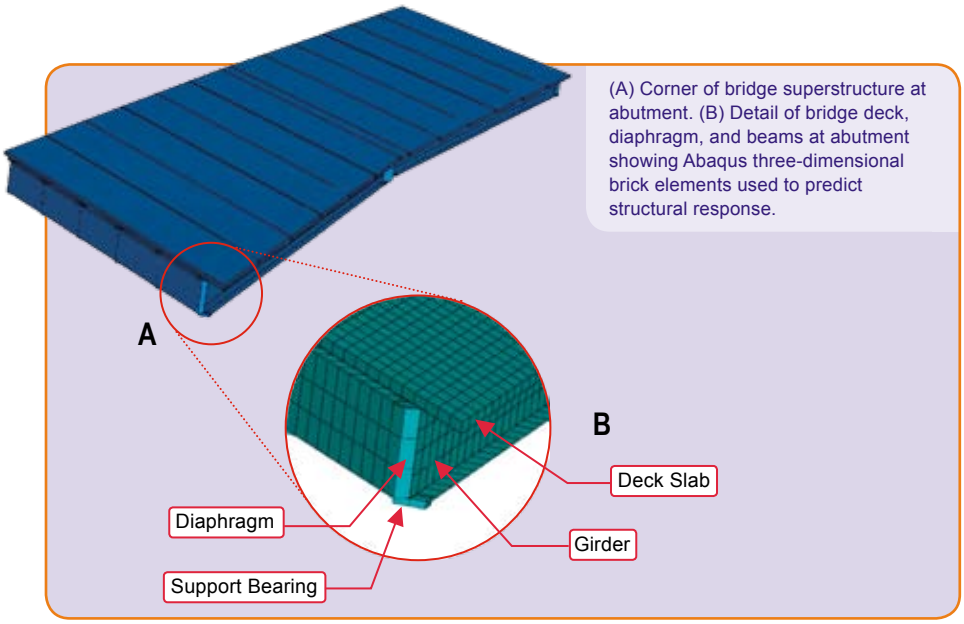
How to Build a Virtual Bridge

Linzell's group uses Abaqus/Standard to create these virtual bridges. First, they build a numerical model based on an existing design. (They can work with designs that are still on paper or in CAD (computer-aided design) format, or they can use as-built measurements taken from a structure already under construction.)

They next select elements (these are the tiny geometric shapes, mathematically representing physical units, that are linked by nodes to form a numerical model) and then material models. "You choose elements depending on available material (constitutive) models and geometry, then select what's best for the materials being used, such as concrete or steel."

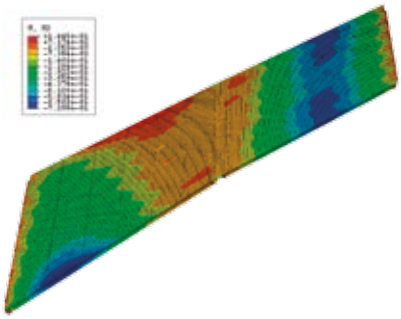
Next they set up the boundary conditions for the model. "We use Abaqus a lot in this stage," he says. "We select how the bridge is going to be restrained, whether we are going to utilize a contact condition or a discrete restraint, and how friction will be represented, for example."

Finally Linzell's group applies various loads to the parts of the bridge, such as the bridge deck to represent vehicle loads or



(A) Corner of bridge superstructure at abutment. (B) Detail of bridge deck, diaphragm, and beams at abutment showing Abaqus three-dimensional brick elements used to predict structural response.

onto the beam faces to represent wind loads. (Stresses can be determined either at nodes or in the elements themselves.) "This is a fairly prescribed process, but it depends on what you are looking at—such as traffic loads, or the weight of the structure itself," he says. The Abaqus creep module is used for time-dependent factors. "Creep is a big issue with concrete, and similar time-dependent effects influence steel behavior as well. Thermal loads are important, too: We've taken data from bridges where there was a 50–60 degree temperature change during construction that certainly affected structure behavior."



Abaqus 3D finite element analysis (FEA) model of vertical displacement contour of a bridge deck, representing how the entire bridge moves under its own weight.

Depending on where loads prove to be excessive, the bridge model—and, ultimately, the performance of the actual bridge—can be modified. The process can be repeated until the optimum configuration for the bridge is reached. "There's a lot that goes into modeling how a massive, highly-indeterminate structure like a bridge is going to respond," Linzell notes.

"Abaqus helps us get our bridge models as accurate as possible."

Real-World Applications

The software has other application potential in the field of bridge analysis besides designing and testing new structures, according to SIMULIA senior engineer Deepak Datye. "Abaqus can be used to evaluate the residual life of a damaged structure that is still standing but may be cracked. And it can also be used for forensic purposes, to help pinpoint the reason for a collapse."

Linzell sees nonlinear FEA playing an increasingly important role in building better bridges for the future. He is part of a group of researchers, practitioners, and Department of Transportation (DOT) engineers who are collecting questionnaires from fellow bridge-building professionals related to their current use of numerical tools.

"We are hoping to come up with unified FEA guidelines for bridges because our industry really doesn't have a unified publication yet," he points out. "Other disciplines like aerospace engineering, and to some extent mechanical engineering, already do, so we're trying to initiate that process."

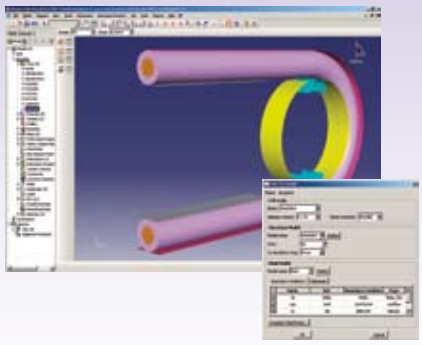
For More Information

simulia.com/solutions/architecture
www.engr.psu.edu/ce

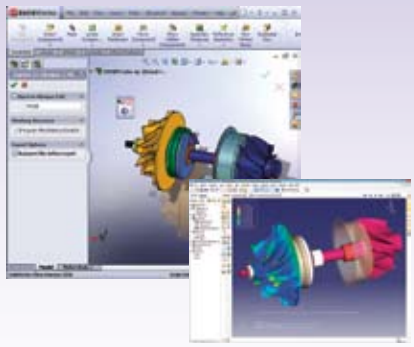
Abaqus Version 6.7 Extended Functionality (EF) Release

New Native Multiphysics Capabilities and Enhanced Features

The latest release of Abaqus FEA software builds on SIMULIA's promise of delivering a broader set of realistic simulation solutions within an open, yet unified modeling and simulation environment. New and enhanced capabilities in Abaqus Version 6.7-EF include, but are not limited to: native Coupled Eulerian-Lagrangian (CEL) technology for multiphysics fluid-structure interaction (FSI) (see *INSIGHTS* article on p. 9), an associative geometry interface to SolidWorks, Abaqus/CAE FSI Interface, collaborative modeling capabilities, tools for managing material libraries, and improvements in the Distributed Memory Parallel solver and modeling of hyperelasticity.



The Abaqus/CAE FSI Interface is used to simplify the process of defining coupled fluid-structure interaction models with third-party computational fluid dynamics (CFD) software directly in Abaqus/CAE.



The SolidWorks Associative Interface enables users to share components or an entire assembly between SolidWorks and Abaqus/CAE for advanced structural analysis. Modifications to the SolidWorks model can be quickly updated in Abaqus.

Enhanced Features

For customers who use SolidWorks as their 3D design tool and want to leverage Abaqus for advanced structural analysis, the new SolidWorks Associative Interface provides geometry transfer and maintains the relationship between SolidWorks and Abaqus models. Once a SolidWorks model has been transferred to Abaqus/CAE, the user can continue to make design modifications in SolidWorks and propagate these modifications to the Abaqus/CAE model with a single mouse click. Any of the features created in Abaqus/CAE, such as partitions, loads, boundary conditions, sets, and surfaces, are regenerated each time the user imports the modified model from SolidWorks to Abaqus/CAE. "This new capability provides significant gains in efficiency and quality by enabling SolidWorks users to make immediate design performance decisions based on advanced analysis results from Abaqus," stated Rainer Gawlick, vice president of worldwide marketing, SolidWorks.

The new Abaqus/CAE FSI Interface simplifies the process of setting up coupled fluid-structure interaction models with third-party computational fluid dynamics (CFD) software, such as STAR-CD or AcuSolve. It enables Abaqus users to define the structural and fluid sub-domains to be used in a coupled FSI analysis directly in the Abaqus/CAE environment. The fluid sub-domain setup is standardized, regardless of the CFD code being used. The rest of the FSI workflow includes these steps:

- Define the interactions and properties
- Run the coupled analysis
- Postprocess the combined solution using Abaqus/Viewer

The FSI interface enables Abaqus/CAE users to manage, store, and re-use FSI model data efficiently. Initially introduced in Abaqus Version 6.7 as an Abaqus Answer, the Abaqus/CAE FSI Interface is now available with Abaqus Version 6.7-EF as a SIMULIA-supplied plug-in.

The Abaqus Version 6.7-EF release also features collaborative modeling in Abaqus/CAE, which allows the import of models and objects from other Abaqus databases into the current analysis session, facilitating concurrent modeling between multiple engineers.

The Material Library in Abaqus/CAE 6.7-EF enables users to build local and enterprise-wide libraries of CAE materials, which facilitates data reuse and sharing of material information between multiple Abaqus/CAE users. In addition, third-party plug-ins provide access to a broad range of materials associated with industrial and military/aerospace databases, such as MMPDS-03. A plug-in developed by Granta Design (www.grantadesign.com) enables Abaqus/CAE users to directly access corporate materials data managed through the GRANTA MI™ materials information system, as well as Granta's broad range of reference data. A Matereality (www.matereality.com) plug-in, which is available for download from the Abaqus Process Automation Portal (www.simulia.com/PAPortal), enables users to access properties from personal, corporate, and public material databases residing within the Matereality Global System and incorporate them into their Abaqus/CAE Material Libraries.

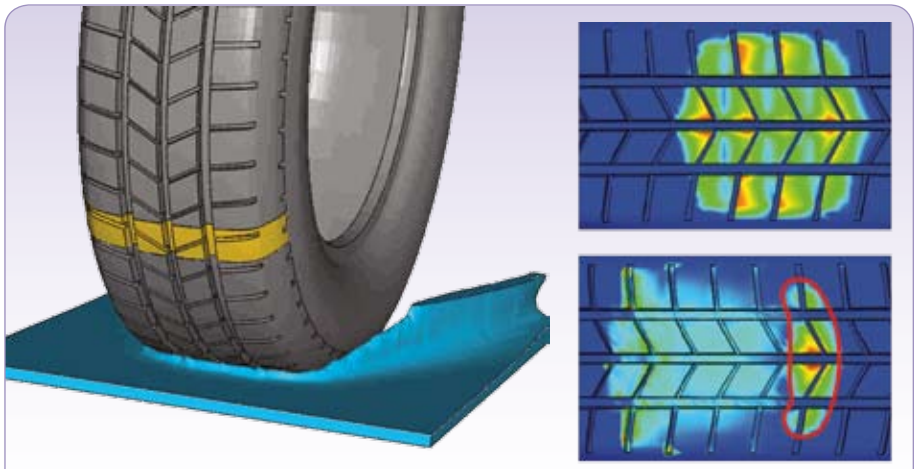
The new release provides improvements for automatic contact detection, geometry- and mesh-based searches, and contact output. Enhancements to the Distributed Memory Parallel solver support parallel element operations with steady-state transport and improve performance and scalability for analyses with a large number of processors. The permanent set with hyperelasticity capability provides higher fidelity modeling for applications involving filled elastomers and thermoplastics.

Abaqus Version 6.7-EF demonstrates SIMULIA's commitment to providing usability, technology and performance enhancements that improve efficiency and productivity in evaluating realistic product behavior.

For More Information
simulia.com/products/abaqus_fea

Coupled Eulerian-Lagrangian (CEL) Multiphysics Available in Abaqus Version 6.7-EF

The newest multiphysics capability in Abaqus is the Coupled Eulerian-Lagrangian (CEL) approach in Abaqus/Explicit. CEL, which is available with Abaqus Version 6.7-EF, streamlines the process of evaluating industrial applications where the hydrodynamic effects of fluids on structures is critical in influencing product design, quality, and safety. This approach, using a multi-material finite element formulation, allows the structure to be modeled in a Lagrangian fashion and the fluid in an Eulerian fashion with the fluid-structure interface and interactions (FSI) being tracked by the general contact feature in Abaqus/Explicit. CEL is particularly well-suited to handle the class of FSI problems where the structural response is dynamic, highly nonlinear, and may include damage or failure of the structure during the process.



Coupled Eulerian-Lagrangian (CEL) analysis in Abaqus/Explicit enables the modeling of complex fluid-structure interaction analysis, such as tire hydroplaning.

Tread-road contact patch before (top) and after (bottom) hitting a puddle of water in a hydroplaning analysis.

Tire hydroplaning is an example of how CEL can provide efficient and accurate modeling capabilities. The speed at which a tire hydroplanes on a wedge of water is a function of the vehicle velocity, water depth, vehicle load, tire pressure, and—most importantly—the tread pattern and depth. When the tire is unable to remove sufficient water from its path, water lifts the tire completely off the road, causing the possibility for the driver to lose control of the vehicle. Using the CEL methodology, the tire is modeled in a Lagrangian framework and the water in an Eulerian framework. This approach allows engineers to study the effect of various design parameters on the onset of hydroplaning completely within the Abaqus environment.

Another CEL application is the simulation of a water bottle drop. Plastic bottles used for water coolers must be able to withstand certain severe loadings—specifically, drops from various heights and angles. However, the repeated sterilizations that the bottle must endure slowly reduce the stiffness and strength properties of the polymeric material. To predict the working life of the bottle, the effect of the degraded material properties on the burst strength must be understood. The fluid behavior is easily simulated with equation of state capabilities. Rupture of the bottle can be captured with strain-based dynamic failure models that allow for element deletion.

With the ability to accurately simulate impact loading of water bottles, the need for physical testing is reduced. A greater understanding of how a water bottle performs as its material properties change allows for a better prediction of the bottle's working life. Taken together, these benefits translate into lower production costs and higher quality bottles.

Additional examples of CEL analysis application include water impact problems such as slamming of boat hulls, wave loading on offshore structures, deployment of airbags for occupant safety, fuel tank sloshing, and high-velocity impact and penetration.

To learn if the new CEL technology in Abaqus Version 6.7-EF is the appropriate solution for your application, please contact your regional SIMULIA office.



CEL analysis allows drop testing simulation of containers filled with fluids to study the durability and working life of the container under severe loading conditions directly in Abaqus.

Model courtesy of Bayer MaterialScience, LLC.

For More Information

simulia.com/products/extd_multiphysics

Harvesting the Ocean's Power with Pelamis and Realistic Simulation



Energy generated from ocean waves could provide electricity on a similar scale to existing nuclear or hydroelectricity sectors, according to the World Energy Council, which estimates a large global market potential for wave energy of 2,000TWh/year. Now Pelamis Wave Power Ltd (PWP) is rising to the challenge with a novel Wave Energy Converter (WEC) machine it is designing with the help of Abaqus FEA.



The Pelamis WEC is a semi-submerged, articulated structure composed of cylindrical sections linked by hinged joints. The wave-induced motion of these joints is resisted by hydraulic rams, which pump high-pressure fluid through hydraulic motors via smoothing accumulators, driving electrical generators to produce energy.

Utilities and energy companies can access electricity from power projects consisting of arrays of interlinked Pelamis machines known as "wave farms." A wave farm of 40 WECs, covering a square kilometer of ocean surface, is capable of generating electric power for 20,000 homes.

The Challenge of "Farming" Ocean Power

Of course, the ocean is no peaceful farmland: Pelamis machines must be able to absorb power from waves, keep functioning no matter what the weather, and even be able to withstand unlikely, but nevertheless possible, impacts from boats or other floating objects.

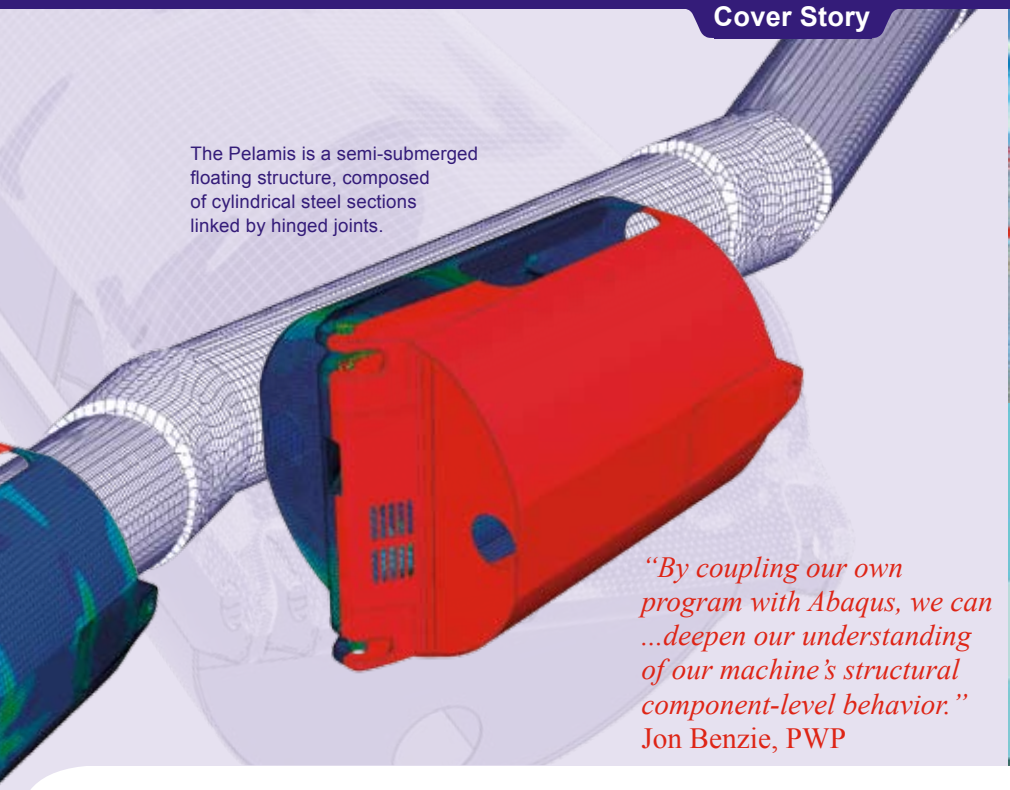
Regulatory permitting for a wave farm also involves environmental impact and site studies. "Our machines are one of the most environmentally benign forms of electricity generation available," says PWP Business Development Director Max Carcas. "Everything is sealed so no greases or fluids are in direct contact with the seawater, and the fluids we do use are biodegradable and non-toxic."

To design such an adaptable, rugged, and clean-running machine, Pelamis turned to Abaqus from SIMULIA for complex, nonlinear, finite element analyses (FEA) of its product's structure, materials, and performance.

Survivability is Key

"The main design driver for the Pelamis WEC concept is survivability," said Jon Benzie, Senior Engineer at PWP. "Our WECs absorb power in small waves through hydrostatic forces; i.e., buoyancy versus weight or hydrostatic pressure. However, extreme loads in waves arise from hydrodynamic forces such as inertia, drag, and slamming. For fine-tuning the response of our machines to their environment, numerical modeling with Abaqus is a vital part of our development program, as it allows rapid design evaluations and optimizations to be made."

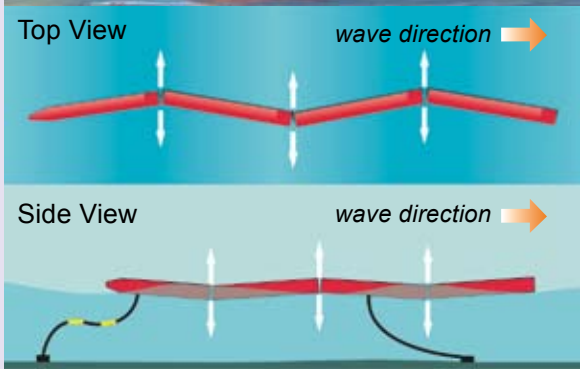
To develop a Pelamis machine, PWP begins with an initial concept based on either



The Pelamis is a semi-submerged floating structure, composed of cylindrical steel sections linked by hinged joints.

"By coupling our own program with Abaqus, we can ...deepen our understanding of our machine's structural component-level behavior."
Jon Benzie, PWP





information from prototype testing or from existing designs to which performance improvements for certain climatic conditions need to be added. The initial design is then analyzed both computationally and with scaled prototypes during tank tests where large waves are simulated and nonlinear behavior is observed. Once the global machine is defined, PWP engineers design the key machine components. This is where most of the FEA analysis is done, incorporating input related to hydraulic systems, electrical layouts, and production assembly requirements.

Since the lifespan of a machine can reach 20 years, PWP performs a considerable number of FEA design iterations on components that are put to the test with respect to fatigue performance and stress analyses. Once the global design is complete, PWP creates the detailed ones from which the machines are assembled and linked into wave farms by PWP's offshore installation team.

Novel Design Requires Unique Analysis

When designing the linked machines, Pelamis engineers created a novel joint configuration to induce a tuneable, cross-coupled resonant response to waves, which greatly increases power capture in small seas. Control of the joints allows this response to be "turned up" in small seas and "turned down" to limit loads and motion in what are known as "survival conditions" —i.e., rough seas—allowing the machine to swing head-on and "dive" into oncoming waves.

"This response adaptability means that a Pelamis machine is very different from a traditional offshore structure such as a platform, which experiences a few very large load cycles (from storms) that cause a great amount of fatigue damage," says Benzie. "To model our WEC's unusual response characteristics, we chose a different style of fatigue analysis, called a rainflow program, to simulate and evaluate the behavior of the machine during a variety of sea states and machine configurations."

PWP tested its rainflow program using several different Abaqus FEA models, from a global shell model of an entire unit, to a sub-model based on one fabrication detail (a main bearing plate and surrounding structure). Results of stress testing indicated that the submodeling method worked well for solution accuracy when compared against the global one.

PWP also employed the software's extensive material modeling capabilities to understand the behavior of different materials for machine design in order to produce the most efficient, cost-effective solutions.

Extending FEA Capabilities with Abaqus

For detailed nonlinear analysis, PWP turned to SIMULIA because "we needed to extend our finite element capabilities and Abaqus was, by far, the best solution available," says Benzie. "It allows us to analyze scenarios of nonlinear behavior that we cannot test for, such as the unlikely event of a ship driving into a farm of machines."

The high level of technical support provided by SIMULIA was another deciding factor in PWP's choice of the Dassault Systèmes brand. "Abaqus has become the global finite element package at PWP, which we now use for practically all our analysis needs," says Benzie "We use it for initial concept analysis, general design work, detailed design work, and what-if scenarios."

"We are also looking to expand the types of analyses we can do with Abaqus FEA by tying it into our dedicated in-house analysis program to directly model machine behavior in the waves. By coupling our own program with Abaqus, we can tailor our stress and fatigue analyses to deepen our understanding of our machine's structural component-level behavior."

For More Information
simulia.com/solutions/power
www.pelamiswave.com

Critical Elements for Managing Simulation Data and Processes

Paul Lalor, Product Manager, SIMULIA

It's a given in our industry that simulation technology and methods have resulted in faster and more cost-effective delivery of innovative and reliable products than in the historical build-and-break process. However, even those companies gaining significant benefits from simulation will admit that they often fail to retain their processes or manage simulation results efficiently enough so that others in their organization can reuse the knowledge gained in an effective way.

A 2007 survey from industry research firm CPDA reported that only 6.1 percent of companies are using any form of a simulation data management solution, which is typically designed to handle large, complex, and highly-specialized simulation data files. Thirty-five percent use legacy databases or data management applications, and the largest percentage of respondents, 42.7 percent, keep their simulation data on local or departmental drives. This makes it difficult—if not impossible—for other decision-making stakeholders to access the information or even be aware of its existence, leading to the possibility of repeating the same simulation, or overlooking an important performance metric. These issues have created a growing industry consensus that the data, processes, and tools associated with simulation must be brought under control and managed.



During the past few years we have recognized this trend, and when we became a part of Dassault Systèmes, we were provided the opportunity to leverage proven Product Lifecycle Management (PLM) technology and combine it with our years of simulation expertise to develop a new product suite for Simulation Lifecycle Management (SLM). Our vision for this solution is to provide robust technology and methods that enable our customers to bring order to their simulation processes and achieve a new level of efficiency in shortening development cycles, reducing waste, and improving product quality while fostering a culture of collaboration and innovation.

Engineering organizations that wish to incorporate SLM into their processes face several tasks, which include evaluating their current processes, integrating various design and simulation applications, developing standardized simulation methods, and incorporating these processes and tools into a managed and collaborative environment.

We believe that a robust Simulation Lifecycle Management solution must deliver a core set of capabilities:

Collaboration

Product designs involve trade-offs between multiple performance disciplines such as strength, weight, vibration, and durability. SIMULIA SLM supports cross-functional collaboration among all key stakeholders so that innovative products that satisfy wide-ranging performance requirements can be developed quickly and efficiently.

Simulation Data Management

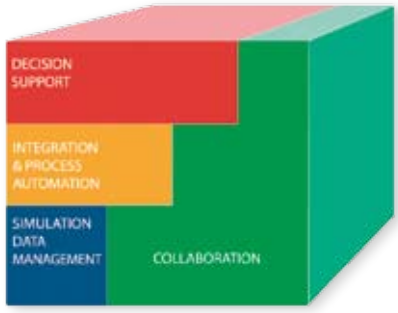
The SIMULIA SLM solution enables simulation-related data to be collected, secured, managed, and associated with related product data in a central repository. It maintains the relationship between engineering targets and key simulation results and provides a searchable environment for valuable related data. It also provides the ability to trace the history of individual simulation processes including parameters, assumptions, and results that influence key design decisions.

Integration and Process Automation

Complete product performance analysis is facilitated by a broad spectrum of best-in-class and proprietary simulation applications. SIMULIA SLM provides the capability to connect these various tools in an open, yet controlled manner. In addition, as organizations mature the management of their simulation data and processes, SIMULIA SLM provides the resources to capture, automate, and deploy approved simulation workflows to a wider group of non-expert simulation users.

Decision Support

Simulations are used to predict the performance attributes of design candidates and their suitability toward meeting engineering targets and marketing requirements. SIMULIA SLM will enable decision support mechanisms that provide cross-functional insight and guide requirements-driven design decisions.



The ongoing industry challenges of shorter product lifecycles, higher material costs, and stricter government regulations ensure that realistic simulation will play a growing role in the product design and manufacturing process of the future. Those companies performing realistic simulation on a regular basis find themselves at a crossroads. They can either continue to let simulation processes and results to be disconnected and unmanaged, or they can explore the emerging Simulation Lifecycle Management solutions. We believe that organizations that embrace this new paradigm will be able to fully leverage their simulation expertise as the valuable corporate asset that it has become.

For More Information
simulia.com/products/slm

SIMULIA SLM—New Product Suite for Managing Simulation Data, Processes, and Applications

The early availability release of SIMULIA SLM provides significant functionality out of the box so that users can gain tangible benefits efficiently and affordably. The new software delivers capabilities to manage all data associated with simulations, 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.



The top image shows the 2D web-based interface for SIMULIA SLM. By leveraging PLM functionality from ENOVIA, SIMULIA SLM enables the management of data, process, and tools associated with all types of simulation. The lower image demonstrates how SIMULIA SLM takes advantage of ENOVIA - 3DLive to search, view, and navigate simulation information in a 3D collaborative environment.

"The release of SIMULIA SLM marks a major milestone for SIMULIA as we expand our product portfolio beyond the Abaqus product line," said Mark Goldstein, CEO, SIMULIA. "By leveraging PLM technology from ENOVIA and simulation expertise from SIMULIA, we have been able to rapidly develop what we believe is an industry-leading solution. SIMULIA SLM will enable our customers to secure their simulation intellectual property and transform it into a valuable and controlled corporate asset."

A Collaborative Environment for Simulation

SIMULIA SLM offers two types of user interfaces. One option is a standard web-based thin client browser application, such as Microsoft Internet Explorer, which provides users with a full SLM authoring environment in which simulation-related data can be created, structured, edited, and viewed and in which the resulting simulations can be executed. The other user interface option is ENOVIA - 3DLive, which provides read-only, 3D, visual navigation of all SIMULIA SLM entities.

The software provides a variety of Simulation Entities and Operations. The top-level entity is known as a "Simulation," which collects or references all of the data and controls all of the activities associated with building and executing a product or process simulation. Simulation data can be logically grouped and collected or referenced beneath a Simulation in folder-like entities called Simulation Categories. There are six default Simulation Categories—Product, Specifications, Internal Data, Validated Data, Context, and Results. User privileges can be assigned for accessing, creating, editing, or altering any entity. The History operation provides a fully traceable and auditable record of all transactions related to a Simulation entity.



In a typical manufacturing simulation, such as this blow molding of a bottle, SIMULIA SLM integrates and executes simulation applications, captures process workflows, manages simulation data, and enables the review of simulation results in a distributed, collaborative environment.

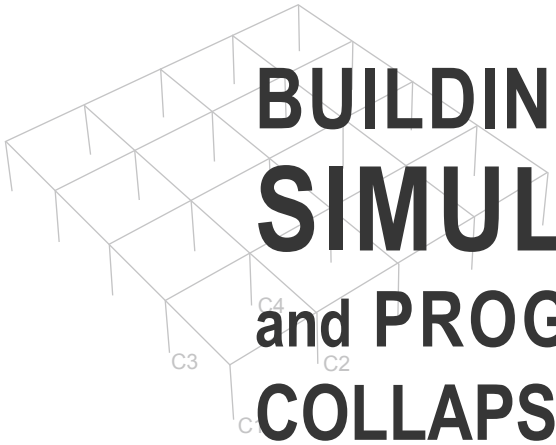
Most simulation processes consist of one or more Simulation Activities. Often, these activities are enabled by a simulation application such as Abaqus/CAE for pre-processing or Abaqus/Standard for solving. Any number of Simulation Activities can be configured within a Simulation. These activities collect or reference their own data in Categories and can integrate other software applications into SIMULIA SLM via Connectors. The Connector framework allows a diverse set of applications to be deployed and executed from within the SIMULIA SLM environment. Connectors are currently available for Abaqus, CATIA, and select third-party simulation applications such as Nastran, HyperMesh, AcuSolve, and STAR-CD.

A Simulation or Simulation Activity can be launched and executed from the SIMULIA SLM Job Execution Framework. The framework allows the jobs to be run on a local host or be distributed to a compute cluster and optionally utilize Distributed Resource Management (DRM) software. Simulation Jobs record detailed information about all instances of the execution of a Simulation or Simulation Activity, and pre-configured rules can be established to govern the specification of import and export data prior to executing a Simulation or Simulation Activity. Document management principles are used to manage versioned and/or non-versioned data files within Simulation entities.

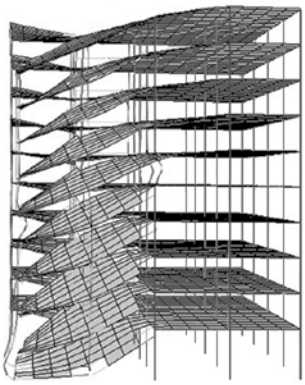
Any SIMULIA SLM entity such as those described above can be assigned a Lifecycle. Promotion or demotion of the entity through its lifecycle states can invoke a review cycle or determine entity behaviors such as access control policies. The Impact Graph displays whether an entity is current or out of date, depending on changes to the status of upstream input and the resulting effect on the downstream output of the entity.

The early availability release of SIMULIA SLM is available today and being deployed at select customer sites. If you would like more information about SIMULIA SLM and its potential benefits for your organization, contact your regional office.

For More Information
simulia.com/products/simulia_slm



BUILDING BLAST SIMULATION and PROGRESSIVE COLLAPSE Analysis



Ted Krauthammer, University of Florida
and Jeff Cipolla, SIMULIA

FEA analysis of severe blast loading supports the design of survivable structures without necessarily requiring expensive physical simulations of a specific explosive or combustion event. It also suggests that established structural design guidelines for severe blast loading on steel structural members contain underestimates and may need revising. In the study described below, we gained a number of significant insights through modeling the progressive collapse of a steel frame structure.

Analysis of Steel Frame Connections Under Blast Loads

Current U.S. design guidelines for steel connections in structures subjected to blast loads are based on recommendations in Department of Defense Technical Manual (TM) 5-1300TM. The approach idealizes real structures and structural elements as "equivalent" lumped-mass single degree-of-freedom systems. In addition, the guidelines are for single-storied steel frames, not subjected to any significant dead loads.

We used Abaqus/Explicit Version 6.5 to assess the behavior of steel moment connections under dead loads. We then compared the maximum rotational capacities of the connections to values derived with the TM 5-1300 approach, using four different load cases for each connection type (see Table 1).

Reference maximum blast pressures were based upon the TM 5-1300 criteria, using standalone codes SHOCK and FRANG to compute equivalent shock pressures and gas pressures (Table 1) and assuming an 18.5 lb. TNT charge in the centre of the room. We applied blast pressures as spatially uniform surface loads on the sidewalls that transferred to the beams and the column of the connection and dead loads equivalent to those for a 10-story office building on the top flanges of the beams and axially on the top cross-section of the column. The numerical model was analyzed with and without dead loads to evaluate their influence on connection response.

An isotropic elasto-plastic model simulated the material property for each connection component. Yield and ultimate strengths were increased to account for strain rate effects using dynamic increase factors (DIF) as recommended in TM 5-1300. We adopted the shear failure model; Abaqus removed elements from the mesh as they failed. The finite element models (Figure 2) were created using predominantly 8-node continuum brick elements with reduced integration.

When blast pressures were applied to the floor and sidewalls, the predicted global rotations of the beams were close to the TM 5-1300 results for the frangible wall cases. However, the beams near reflecting walls rotated much more than the TM 5-1300 computation predicted, transferring

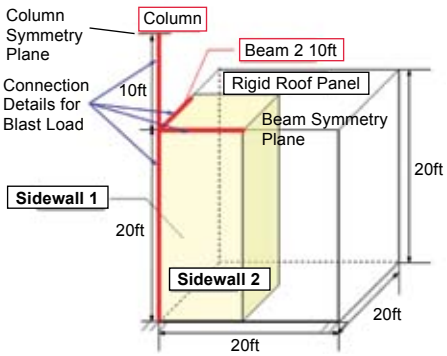


Figure 1: Geometrical Model used for Numerical Study

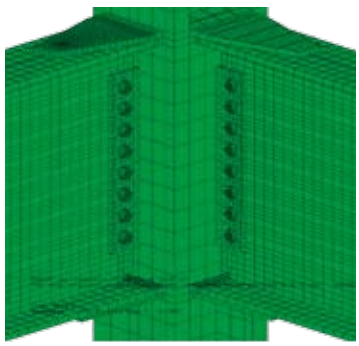


Figure 2: Representative Finite-element Model of Steel Connection

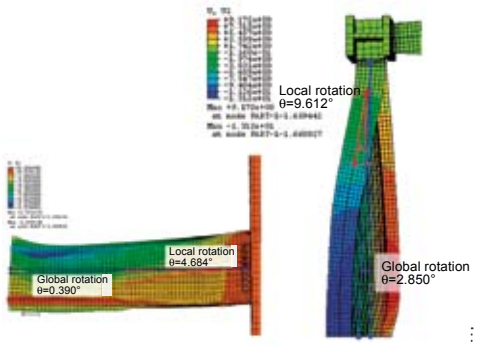


Figure 3: Global vs. Local Rotations Case 1, No DL, No DIF. Horizontal (left) and Vertical (right)

greater impulse and energy to the beam and column (Figure 3). All local rotations for the different cases exceeded the limit of 2 degrees specified in TM 5-1300.

The beams twisted severely horizontally and clearly exceeded the TM 5-1300 limit criteria. These findings indicate that extensive damage in the connections comes from the blast radiating in three dimensions as well as the vertically applied pressure. Deformation data for beams and columns in the various cases indicate that dead loads and DIFs enhanced structural strength, but the beam cross-sections twisted additionally due to dead loads. The column rotations indicate that the columns did not significantly affect the connection damage.

According to the stress and strain results, components in all connections yielded for all the cases.

These analyses show the value of investigating structural connections using high-resolution finite element analysis. For example, a steel moment connection judged safe based on TM 5-1300 criteria failed in the finite element simulations. Moreover, the TM 5-1300 criteria may need revision to reflect findings based on more complex behavior.

Progressive Collapse of Steel Frame Structures

In progressive collapse, local damage leads to large-scale structural failure—an intrinsically transient nonlinear phenomenon that is difficult to model, understand, or design against without finite element analysis.

We studied ten-story 3D moment frames with rigid and semi-rigid connections for their sensitivity to material, buckling, and connection failures of specific columns. The nonlinear moment-rotation relationship of the 10-story frame was obtained through extensive preliminary 3D finite-element simulation of steel connections.

Six initial failure cases with rigid and semi-rigid connections were used to analyze the frames for progressive collapse of five stories. Frame columns were based on a simple LRFD design procedure manual. Both ideal (rigid plus hinge) and semi-rigid connections were adopted for the progressive collapse analyses.

In the simulation with ideal connections, only Case 6, where three columns were removed, caused total collapse of the building. Case 6 with semi-rigid connections also collapsed, but differently, as shown in Figure 5. The first failure was initiated at a connection. As additional connections failed, the floors above the removed columns fell, causing columns to buckle in the 6th floor. These column buckling cases initiated horizontal failure propagation in the 6th floor, and the whole floor failed. After that, the columns in the first floor buckled because the floors collapsed, leading to the total collapse of the building.

Even though the ideal and semi-rigid connection cases both caused total collapse for Case 6, nonlinear finite element results showed very different qualitative behavior. The collapse of the semi-rigid connection case was caused by a cascade of local failures, such as connection failures and

Table 1: Loading Data

Sidewall Case		Member	Shock Pressure		Gas Pressure	
			Peak Pressure, Psi (MPa)	*Time, msec*	*Peak Pressure, Psi (MPa)*	*Time, msec*
1	Two failed	Side walls	154.7 (1.07)	1.81	28.1 (0.19)	29.95
		Floor	29.5 (0.2)	7.19	28.1 (0.19)	31.08
2-1	Sidewall 1 failed, sidewall 2 reflects	Side walls	154.7 (1.07)	2.23	28.1 (0.19)	43.5
2-2	Sidewall 2 failed, sidewall 1 reflects	Floor	29.5 (0.2)	9.06	28.1 (0.19)	44.52
3	Two sidewalls reflect	Side walls	154.7 (1.07)	2.65	28.1 (0.19)	5864.63
		Floor	29.5 (0.2)	11.17	28.1 (0.19)	5866.79

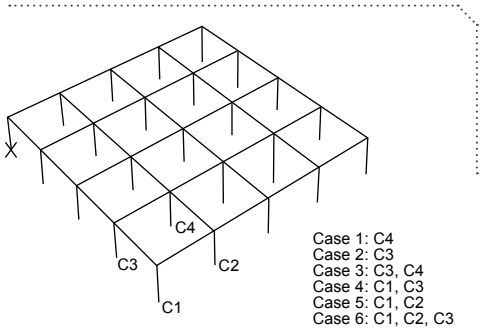


Figure 4: Initial Column Failure Cases

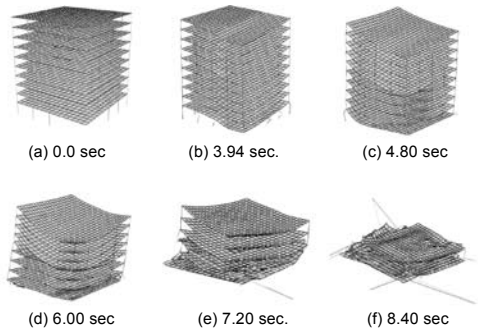


Figure 5: Case with Semi-rigid Connections

columns buckling. However, the collapse of the ideal connection case was caused by column buckling in the first floor. The analyses also showed that once failure propagation initiated (i.e., horizontal column buckling), it would not stop until it caused total, or almost total, collapse. Horizontal column buckling propagation appears to be the most critical factor to control.

Conclusions

Our analyses suggest that connection behavior under blast loading varies significantly from standard design criteria, which may not be conservative enough and may require refinement and revision in light of nonlinear transient effects, such as progressive collapse. Finite-element analysis of progressive collapse due to blast effects also revealed sensitivity of failure mode to connections.

The complete technical paper, "Building Blast Simulations," was presented at the NAFEMS 2007 World Congress. It can also be downloaded from the SIMULIA website.

For More Information

simulia.com/solutions/architecture
www-testing.ce.ufl.edu

Mississippi State University Re-Designs "Green" SUV to Win Challenge X Competition

At the third annual national Challenge X competition, Mississippi State University's team placed first among sixteen other universities. The competition, sponsored by General Motors and the U.S. Department of Energy and organized by Argonne National Laboratory, is a multi-year contest that requires the teams to re-design an SUV to minimize energy consumption, emissions, and greenhouse gases while maintaining or exceeding the vehicle's utility and performance.

Abaqus and CATIA software played a key role in component design, optimization, and packaging. The diesel engine and transmission were positioned in the SUV's engine bay and the engine mounts were both modeled and machined with the help of CATIA. The team used Abaqus to perform multiple finite element analysis studies to optimize the performance of the Chevrolet Equinox's rear suspension cradle.

"Dassault Systèmes software is what helped separate our team from the other schools," said Neil Littell, PLM Coordinator for the Center for Advanced Vehicular Systems (CAVS) at MSU. "CATIA was instrumental in redesigning the components to fit our needs while Abaqus simulated realistic stresses on the components."



MSU edged out the other teams in the competition with its diesel-electric, parallel hybrid vehicle. The front wheels are powered by a 1.9L GM direct injection turbo diesel engine fueled by B20 biodiesel, and the rear wheels are powered by an electric motor. Compared to the original design, the team was able to increase fuel economy by 48% and reduce its well-to-wheels greenhouse gases by 60%.

"Programs such as this offer students the opportunity to work with real-world processes and tools, better preparing them for success in the automotive industry," said Xavier Fouger, head of Dassault Systèmes Learning. "We are proud that our CATIA and Abaqus products played a critical role in helping the Mississippi State team win the competition."

For More Information
www.challengex.org
www.cavs.msstate.edu

Latest FEA Technology Available in Abaqus Student Edition

By using the current academic versions of Abaqus, engineering students are able to use the same state-of-the art technology included in the professional version of Abaqus. New features in this release mirror those included in Abaqus Version 6.7 and provide improved architecture for high-performance linear dynamics, advanced simulation capabilities for nonlinear materials and composites, and a highly-customizable user interface for accelerated model building and results visualization. The enhanced HTML documentation also provides a thorough, searchable resource for students to find detailed information.

"For our introductory finite element method (FEM) course, Abaqus Student Edition is an excellent companion to the Abaqus Teaching Edition," stated Professor Richard Regueiro, University of Colorado at Boulder. "Students can do their homework problems on their personal computers with the student version and then use their initial model setup to solve larger problems in the lab using the Abaqus Teaching Edition. This upward compatibility helps them be prepared to use the Abaqus Research Edition for their graduate work."



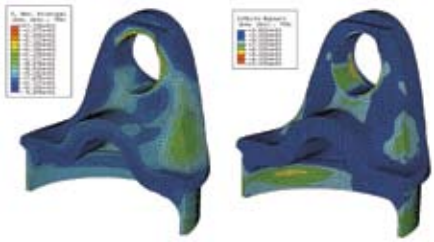
Professor Regueiro uses Abaqus Student Edition in an innovative introductory FEM course at the Center for Advanced Engineering and Technology Education (CAETE) at the University of Colorado at Boulder (http://caete.colorado.edu/). The class is taught via a web-based service, allowing Professor Regueiro to demonstrate Abaqus on a large-screen monitor while remote students follow the tutorial on their laptops.

For More Information
simulia.com/academics/student
caete.colorado.edu

Advanced Fatigue Life Analysis using Abaqus and fe-safe

Manufacturers are increasingly driven to improve profitability, increase market share, and reduce the environmental impact of their products by improving the reliability and durability of lighter weight designs and high performance materials. While many companies are well-versed in using finite element analysis to calculate stresses and thermal loads, fatigue studies are often performed using spreadsheet analysis of a few manually selected stress points or by performing time-consuming physical tests.

Safe Technology Ltd is a SIMULIA Software Alliance Partner that develops fe-safe, a highly effective tool for leveraging Abaqus results to accurately predict durability under complex load conditions. The software automatically identifies fatigue hotspots based on Abaqus results and provides accurate life predictions for complex multiaxial load histories. Three dimensional contour plots can be displayed for fatigue life and for allowable stress factors for a specified design life. fe-safe also provides a direct interface for reading and writing results data to the Abaqus output database (.odb) file, which enables Abaqus users to examine fe-safe fatigue results within Abaqus/Viewer.



Images courtesy of Dana Corp, Automotive Systems Group, USA.

In an example of an end-yoke component, the principal stress approach (left image) identifies the hotspot on the outer right side of the part. However, the log-life results from fe-safe (right image) indicate that the area with lowest predicted life is the inner cylindrical tube area at the lower left side of the component. The analysis results were corroborated by lab tests.

fe-safe 5.3 offers enhanced features for fatigue analysis of steady-state modal response solutions and modal superposition transient response solutions. It also provides support for temperature-dependent material properties. The new version has improved the speed of accessing sequential and non sequential data and caching of model data.

fe-safe 5.3-03 supports Abaqus Version on Microsoft Windows and Linux platforms. When used in combination with Abaqus 6.7, fe-safe is capable of reducing time and costs while providing accurate fatigue-life evaluation of components. As the authorized distributor for Safe Technology Ltd, SIMULIA provides experienced and knowledgeable support staff. For more information on leveraging fe-safe, contact your regional office or visit the SIMULIA website.



SIMULIA's technical and marketing team is trained on the latest fe-safe release. Participants from SIMULIA regional offices and headquarters are shown above with Ian Mercer, Stephanie Wood, and John Draper of Safe Technology Ltd.

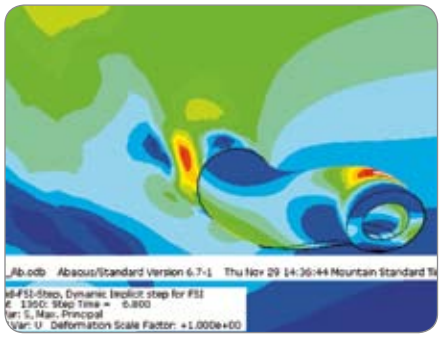
For More Information
simulia.com/products/fesafe
www.safetechnology.com/fe-safe.html

Coupling AcuSolve with Abaqus to Analyze Subsea Pipelines

Transport of oil and gas from ocean floor wells to land-based production facilities requires the use of subsea pipelines that can vary in length from tens of kilometers to over 1,000 kilometers. Along the undersea route, the pipeline can traverse a complex landscape of jagged peaks and valleys. Strong subsea cross-currents can cause a pipeline to go into a self-excited oscillation and potentially cause fatigue or slam the pipeline against the seabed floor. The resulting damage can be very expensive to repair and may result in environmental contamination.

The potential for self-excitation is caused by a fluid flow phenomenon called vortex-induced vibration (VIV). Richard Gosling, of Andrew Palmer and Associates, regularly faces the challenge of solving this type of FSI problem. "In exposed pipeline spans, VIV can cause significant fatigue problems," states Gosling. "The direct coupling of AcuSolve and Abaqus enables our engineers to use a 3D environment to analyze fluid-structure interaction (FSI) and predict the effect of vortex-induced vibration on the structural integrity and fatigue life of pipelines. This capability has transformed our ability to assess the effectiveness of mitigation measures, such as the use of helical strakes, to avoid fatigue problems in exposed subsea pipeline spans."

The recent direct coupling between AcuSolve, a general-purpose, finite element-based computational fluid dynamics solver from ACUSIM Software, and Abaqus FEA software provides a unique solution for solving complex FSI problems.



Co-simulation results of an FSI analysis performed by coupling AcuSolve with Abaqus. A flow field and deformed pipeline are shown at an elapsed simulation time of 6.8 seconds. The ocean current flow is from right to left, and the stress on the pipe is visualized along with the flow velocity magnitude.

For More Information
simulia.com/products/direct_coupling
www.acusim.com
www.andrewpalmer.com

A New Year's Resolution: Make Training an Integral Part of Your Development Plan

Fernando Carranza, Manager, Education Products and Engineering Specialists, SIMULIA

Training is often overlooked, and yet we all know that continued education is essential to our personal development as well as the efficient and effective use of software technology. New features and capabilities in simulation products, such as Abaqus, are now being made available every six months. For you and your team to remain knowledgeable and efficient and provide added value to your organization, it is important to include budget and time for training in your annual plan.

We understand that cost and time are two major constraints for you and your teams. To assist in making training affordable and convenient, SIMULIA regional offices, representatives, and affiliates offer more than 30 unique course titles at over 30 regional locations. In the first half of 2008, the combined worldwide course schedule includes more than 250 classes focused on Abaqus applications. In addition, this year we have added a course on the use of SIMULIA SLM, our new product suite for managing simulation data, processes, and applications.

The worldwide training schedules are on our public website and allow you to quickly find and register for courses based on topic or location. Our class selection offers you the opportunity to choose a course that is appropriate to your skill level or specific to your application and that will be convenient for you in terms of location, budget, and time. Many of our courses are followed by one-day, hands-on workshops. These workshops offer you the opportunity to put theory into practice on your own models under the supervision of Abaqus experts.

We also offer on-site training options that are cost-effective when you have three or more people who need to be trained simultaneously. On-site training helps you get more from your software investment, while requiring less time to get your entire team educated. In addition, through our professional services, we are able to customize a class or a series of classes to your specific workflows or applications, which can be of significant benefit in meeting your project goals and objectives.

The mission of our training services is to provide you with the highest-quality education to help you be more productive in using our simulation software. By investing your budget and time in the appropriate training classes, I am confident that you will boost your analysis skills, increase your productivity, and gain faster returns on your software investment.



A partial list of available training courses:

Introductory Courses

- Introduction to Abaqus
- Introduction to Abaqus/CAE
- Introduction to Abaqus/Standard and Abaqus/Explicit
- Introduction to Abaqus Scripting
- Introduction to Abaqus for CATIA V5
- Simulation Lifecycle Management

Advanced Courses

- Abaqus/CAE: Geometry Import and Meshing
- Abaqus/Explicit: Advanced Topics
- Abaqus for Offshore Analysis
- Adaptive Remeshing with Abaqus/Standard
- Analysis of Composite Materials
- Analysis of Geotechnical Problems
- Automotive Powertrain Assembly Analysis
- Buckling, Postbuckling, and Collapse Analysis
- Contact in Abaqus/Standard
- Coupled Eulerian-Lagrangian Analysis with Abaqus/Explicit
- Crashworthiness Analysis
- DDAM Analysis with Abaqus
- Flexible Multibody Systems
- FSI Simulation with Abaqus
- Heat Transfer and Thermal-Stress Analysis
- Linear Dynamics with Abaqus
- Metal Inelasticity in Abaqus
- Modeling Fracture and Failure
- Modeling Rubber and Viscoelasticity
- Structural-Acoustic Analysis Using Abaqus
- Tire Modeling Using Abaqus

For More Information
simulia.com/services/training_schedule

2008 Abaqus Users' Conference is on the Horizon

May 19–22, 2008 at the Newport Marriott, Newport, Rhode Island



The 2008 AUC marks the 30th anniversary of the commercialization of Abaqus software, so it is fitting that we return to Newport, Rhode Island, the host city of many of our past conferences. First released in 1978, Abaqus software has undergone major developments to become the industry's technology-leading unified FEA solution.

During the Advanced Seminars, General Lectures, and one-on-one discussions, our professional staff will share details with you about the latest enhancements to Abaqus, the expansion of our multiphysics capabilities, and the capabilities of our new Simulation Lifecycle Management solution.

Customer Presentations

Our customers' commitment to presenting their strategies and applications is the main reason for the ongoing success of the AUC. This year's abstracts and manuscripts, from the Automotive, Aerospace, Electronics, and Oil & Gas industries as well as others, have been reviewed and selected to uphold the AUC's reputation for outstanding quality. You will discover new analysis techniques and make important industry contacts by interacting with international experts in industry and academia.

Advanced Seminars

Prior to the start of the conference, we offer the option to register for Advanced Seminars, which provide instruction on the theory and application of the latest simulation capabilities. **Seminars include:**

- Practical Use of Multiphysics Simulation for Industrial Applications
- Solve Challenging Contact Problems with Abaqus
- Managing Simulation Data, Processes, and Applications with SIMULIA SLM
- Multiscale Computations: From Theory to Practice (conducted by Professor Jacob Fish, Director, Multiscale Science and Engineering Center, Rensselaer Polytechnic Institute)



Networking Opportunities

Your conference registration includes breakfasts, breaks, lunches, and evening social events to provide you with time to network with our partners, your peers, and the SIMULIA staff. On Wednesday evening, the AUC Annual Banquet will be held at Belle Mer, nestled between Newport Harbor and Naragansett Bay. You'll enjoy a contemporary, yet elegant dinner complemented by expansive bay views and lively entertainment.

Special Interest Groups

This year we will hold Special Interest Group sessions. Our technical and product specialists will outline specific industry and application workflows, then open the floor for a roundtable discussion. By participating in these breakout sessions, you will have the opportunity to share your requirements, ideas, and experience among peers with similar interests.

Alliance Partner Pavilion

A significant part of the conference is our Alliance Partner Pavilion. You will have the opportunity to explore complementary solutions that will help you improve productivity and streamline your overall engineering process.



Conference Proceedings



A valuable benefit of your attendance at the AUC is the Conference Proceedings. You will receive a high-quality, bound proceedings book and companion CD-ROM containing the complete conference papers prepared by the authors and presenters at the conference.

Current Partner Sponsors include:

- ACUSIM Software, Inc.
- AVL List GmbH
- Beta CAE Systems SA
- CD-adapco
- CEI
- Elysium Inc.
- e-Xstream engineering
- Fraunhofer Institute SCAI
- Granta Design Limited
- Hewlett-Packard Company
- IBM
- Intel Corporation
- NW Numerics
- Platform Computing
- Quest Reliability
- Red Cedar Technology
- Safe Technology

For More Information
simulia.com/auc2008

SIMULIA Helps Keep My Family Safe.



Simulation for the Real World

We depend on our automobiles to be reliable, economical, and safe. Our customers use SIMULIA solutions to understand and improve the safety performance of new vehicles. We partner with our customers to develop realistic simulation methods and technology, which helps them get closer to real world behavior, drive innovation, and make safer cars.

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

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.