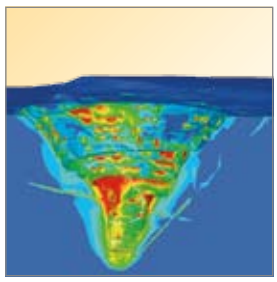
INSIGHTS

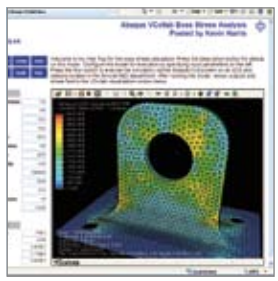
6
2 2009
Dassault Systèmes Realistic Simulation Magazine



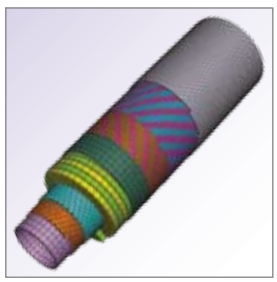
TÜV Evaluates Safety of Nuclear Power Plants



Beck Arndt Engineering
Accelerates Mine Safety Evaluation



Isight and Fiper 3.5
Newest Products from SIMULIA

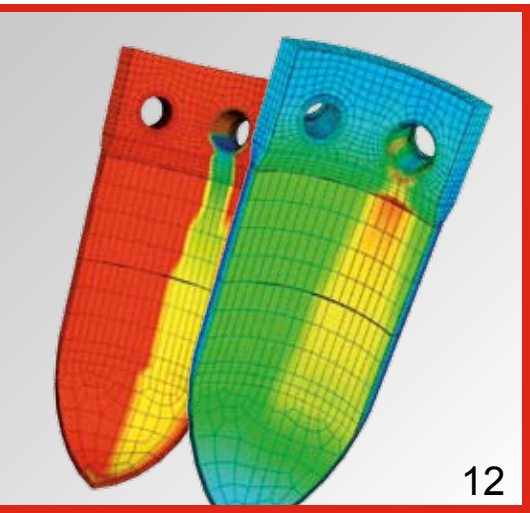


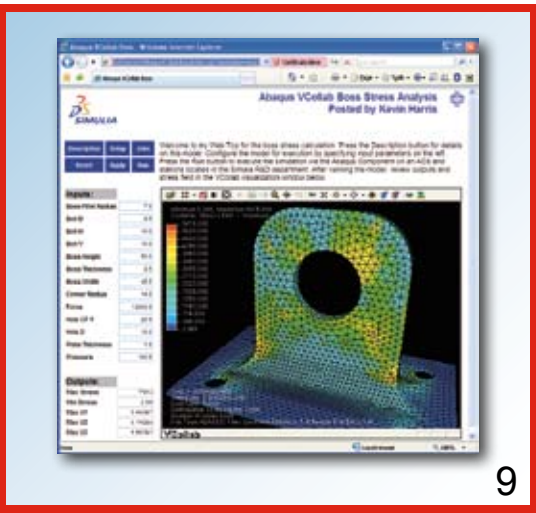
DeepFlex
Composite Pipes for Offshore Energy Applications

SIMULIA

INSIGHTS

January/February 2009







Inside This Issue

12 Cover Story

TÜV Uses Realistic Simulation to Assist Nuclear Power Plant Certification

On the cover: (L to R) Wolfgang Hienstorfer, Thomas Hermann, Sabine Böhm, and Marc-Steffen Sedlaczek of TÜV SÜD ET

8-9 Product Update

- Abaqus 6.8-EF
- Isight and Fiper 3.5

6 Customer Spotlight

Beck Arndt Uses Realistic Simulation to Accelerate Safety Evaluation of Mine Designs

In Each Issue

3 Executive Message

Bruce Engelmann, CTO, SIMULIA

4 In The News

- Industry Press Coverage
- The American Bureau of Shipping Evaluates Offshore Platforms with Abaqus FEA
- Alenia Aeronautica Selects Fiper to Support Enterprise Simulation Framework
- R Systems Achieves Significant Speed-Up for Abaqus FEA Using Flexible Cluster Configuration

10 Customer Case Study

Human Tissue Modeling at Frankfurt University Targets Patient Comfort and Health

14 SIMULIA Energy Strategy

Mahesh Kailasam, Energy Industry Lead, SIMULIA

16 Customer Case Study

DeepFlex Uses Abaqus to Customize Pipeline for Offshore Applications

19 Alliances

- Evaluating Valve Stem Seal Performance with FlowVision and Abaqus
- SIMULIA Hosts Sixth Annual Partner Summit

20 Academics

- Purdue Grad Students Study Computational Fracture Mechanics
- Georgia Tech Students Use Abaqus in AHS Helicopter Design Competition

22 Services

Customer Satisfaction Is Our Top Priority

23 Events

2009 SIMULIA Customer 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
simulia.info@3ds.com

www.simulia.com

Editor:
Tim Webb

Associate Editor:
Julie Ring

Contributors:
Stephan Arndt (Beck Arndt Engineering), Shankar Bhat (DeepFlex), Sabine Böhm (TÜV SÜD ET), Mike Bryant (DeepFlex), Pierre Burgers, Bruce Engelmann, Thomas Hermann (TÜV SÜD ET), Wolfgang Hienstorfer (TÜV SÜD ET), Paul Jacob (MMI), Mahesh Kailasam, Paul Lalor, Tomasz Luniewski (Capvidia NV), David Palmer, Parker Group, Alexander Robledo (Georgia Institute of Technology), Marc-Steffen Sedlaczek (TÜV SÜD ET), Thomas Siegmund (Purdue University), Gerhard Silber (Frankfurt University), Christophe Then (Frankfurt University), Alex van der Velden, Jim Vandermillen

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, 2009.

Our Dedication to Simulation Technology Innovation



Bruce Engelmann
Chief Technical Officer,
SIMULIA

It has been over two years since I last wrote the executive message for *INSIGHTS* magazine. In the fall of 2006, I wrote about three important factors in advancing realistic simulation technology: a multidisciplinary approach to advanced physics modeling, a strategy for exploiting improvements in high-performance computing (HPC), and a passion for advancing technology that makes a positive impact on society.

It is with a sense of satisfaction, as we enter 2009, that I can report on significant achievements in these areas demonstrating our dedication to simulation technology innovation. Our focus on advanced physics modeling is not only continuing, but accelerating. Abaqus 6.8-EF provides new and enhanced capabilities for modeling and analyzing general contact, spot welds, fasteners, foam materials, composites, and fluid-structure interaction.

With the addition of Isight and Fiper to our product portfolio, customers can automate the process of multidisciplinary design exploration while leveraging distributed computing resources and technology for Design of Experiments, optimization, and Monte Carlo studies (see *INSIGHTS* p. 9). Many of these automation and decision support capabilities will become an integral part of our new SLM product suite for Simulation Lifecycle Management.

With regard to high-performance computing, at the end of 2006, we were pleased with the excellent performance of Abaqus running on 32 and 64 cores. It may seem odd that this achievement was reached after 28 years of developing Abaqus FEA technology! The long development cycle to reach that milestone was due to many factors, including the maturation of parallel algorithms, access to affordable computing resources, and a lack of general industry requirements for distributed computing solutions. So, while HPC was slow in coming, it is definitely here to stay. Our customers are rapidly adding more fidelity and size to their models and regularly taking advantage of 32- and 64-core computing systems.

We are placing significant R&D effort in the HPC area. We have surpassed the 256-core mark and are testing models on 512+ cores with promising results. Driven by advances in powertrain simulation, this represents a nearly 10x increase in computing power in less than three years. Our customers can now apply these HPC advances to other simulation domains such as geophysics, oil and gas exploration, mining (see *INSIGHTS* pp. 6-7), and hydropower.

The future for advancing realistic simulation technology is bright. With Abaqus 6.9 and beyond, we are focused on achieving our long-term goal of making the modeling of fracture and failure as common as including the effects of Mises plasticity. In the near future, you will have access to new technology that will enable the simulation of stationary and propagating cracks in 3D models.

As we enter 2009, our customers are more passionate than ever about sharing their experiences with our software. Their success in employing realistic simulation to drive innovation is evident—not only in every issue of *INSIGHTS* magazine, but also in the products that we all use every day. We are also fortunate to have received yet another record number of abstracts for the upcoming SIMULIA Customer Conference (see *INSIGHTS* p. 23). I encourage you to make plans now to attend this valuable conference. You will be able to make worthwhile connections, expand your simulation knowledge, and find out more about our current and future endeavors.

By engaging with our professionals within SIMULIA, you will be able to share your requirements for realistic simulation and ensure that we continue our strong focus on delivering market-leading technology that meets your needs well into the future.

See you in London.



Industry Press Coverage













Railway Strategies
August/September 2008, online
Bridging the Gap
This U.K. publication aimed at senior management in the railway infrastructure industry featured Pennsylvania State University Professor Daniel Linzell's work on improving bridge performance with Abaqus software. Linzell's research group uses FEA to accurately depict the stresses and deformations that affect the performance and service life of a bridge over time. Such results can also help with maintenance, and even forensics in the event of a structural failure.

Medical Design Technology
September 2008, pp. 24-27
The Beat Goes On
Matrix Applied Computing used Abaqus FEA to help Sunshine Heart, Inc. develop a successful design for a novel heart pump that works inside the body but outside the bloodstream. The software was used to model and refine the critical parts of the system, a cuff that encircles the aorta and a balloon that inflates and deflates to compress that blood vessel in time with the heartbeat. The analysis produced an optimal device shape that provided the least variation of strain combined with the maximum amount of compression. The success of this FEA-guided medical product development project was later affirmed when Sunshine Heart received the go-ahead from the FDA to begin human trials in the U.S.

Industry Week
September 3, 2008, online newsletter
Simulation Replaces Physical Prototyping and Testing
SIMULIA product manager Paul Lalor authored this article on how to maximize the business advantages of Simulation Lifecycle Management (SLM). Historically, the isolated nature of simulation in an enterprise has resulted in tremendous inefficiencies; SLM promotes collaboration, data management, integration and process automation, and decision support. This helps companies optimize product performance, reduce material use, and detect and correct errors more efficiently than current methodologies.

Designfax
September 9, 2008, online
When did sports equipment get so smart?
"Intelligent" shoes, balls, and turf that adapt to use by people share a common element of innovative "smart" design enabled by realistic simulation. This online article details how the Abaqus Unified FEA product suite is used by Loughborough University Sports Technology Group (soccer balls), adidas (running shoes), and TenCate (artificial turf) to help design, build, create, test, and fine-tune their products before manufacturing.

Power Engineering International
November 2008, pp. 38-39, 41
Model Behavior: Finite Element Analysis Has All the Answers
SIMULIA's Dale Berry, Mahesh Kailasam, and Jack Cofer teamed up for this in-depth byline about FEA and optimization software applications in the power engineering industry. Advanced Abaqus capabilities—developed through decades of work with automotive, aerospace, and oil and gas customers—now serve the diverse engineering needs of turbomachinery, nuclear plants, wind, wave, and solar power. The combination of Abaqus FEA and Isight for design optimization accelerates product development, while SLM offers data and workflow management and secures intellectual property.

Energy Profile
Issue One, 2008, pp. 2-6
Design on Energy
Three examples of Dassault Systèmes software applications in the energy industry were given in-depth treatment in this extensive U.K.-published article. In nuclear fusion research, CATIA V5 and ENOVIA SmarTeam, supplied and supported by Applied PLM Solutions Limited, are being employed by the world-leading Culham Science Center in Oxfordshire to create and maintain vast amounts of engineering data. In oil exploration, JP Kenny's use of Abaqus FEA is reducing simulation times and improving the accuracy and efficiency of pipeline design and route mapping. Pelamis Wave Power also uses Abaqus FEA for initial concept development, design work, and detailed functional analysis to make their wave energy converters efficient, cost-effective, and environmentally sound.

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.

The American Bureau of Shipping Evaluates Offshore Platforms with Abaqus FEA

The American Bureau of Shipping (ABS), the world's leading offshore classification society, has selected Abaqus FEA to assist in the evaluation of the structural strength of offshore drilling and production units.

Dedicated to promoting the security of life, property, and the marine environment, ABS is at the forefront of research and technological innovations in setting standards for the international marine and offshore industry. To further advance its activities, ABS has selected Abaqus Unified FEA to evaluate the operational performance of offshore jack-up structures.

"Full-scale testing of offshore jack-up rigs is cost and time prohibitive," stated Jer-Fang Wu, head of the ABS Singapore Offshore Technology Center. "In the energy sector, there is a continuing trend towards larger, more complex projects that demand shorter, more intense design cycles. The realistic simulation capabilities of Abaqus provide a reliable and economical way to evaluate structural standards of engineering designs."

"The accuracy of structural analysis in order to engineer the safe operation of offshore units is imperative," states Ken Short, VP Strategy and Marketing, SIMULIA. "The selection of Abaqus by ABS validates our approach to the development and quality assurance of mission-critical tools for the certification of these structures dedicated to economical and environmentally safe energy exploration."

ABS selected Abaqus for its powerful nonlinear analysis solution technology, its robust contact formulations, and its unique simulation capabilities. Utilizing the soil and concrete material models available in Abaqus, ABS can more easily evaluate the behavior of jack-up rig designs operating in harsh, offshore environments.

Alenia Aeronautica Selects Fiper to Support Enterprise Simulation Framework

Major commercial and defense aeronautics supplier Alenia Aeronautica, "a Finmeccanica Company," has chosen Fiper as a key component of their enterprise-wide simulation process integration and collaboration framework to be developed under the Alenia Networked Enterprise Transformation (AleNET) initiative, which has been created to accelerate Alenia's product development and innovation.

Within the scope of the Virtual and Physical Prototype Simulation stream of the AleNET project, Fiper will be used and integrated in the Alenia VPPS platform to capture and manage simulation workflows used across the multi-disciplinary design domain. SIMULIA will work with Alenia engineers and third-party partners, including Exemplar s.r.l., to implement the project.

"We selected Fiper for its capability to efficiently capture simulation workflows and its open component architecture that allows us to integrate a variety of in-house and commercial analysis systems," said Vittorio Selmin, AleNET VPPS leader, Alenia Aeronautica. "Fiper enhances our VPPS vision to leverage the broader scope of process integration within the framework of Simulation Lifecycle Management and enterprise-wide PLM."

"Companies such as Alenia are demanding open architecture solutions that allow them to make simulation an integral, decision-driving practice within their product development process," said Ken Short, VP Strategy and Marketing, SIMULIA. "The selection of Fiper by Alenia confirms our strategy of providing robust Simulation Lifecycle Management solutions that help companies improve efficiency while reducing the time and cost associated with bringing high-quality products to market."

R Systems Achieves Significant Speed-Up for Abaqus FEA Using Flexible Cluster Configuration

R Systems, a technology-leading IT solutions provider, in cooperation with SIMULIA and Dana Holding Corporation, has completed a simulation benchmark using Abaqus FEA software and Windows HPC Server 2008.

"With Windows HPC Server 2008, Microsoft has made huge advancements in performance and scaling that give users of realistic simulation solutions more IT flexibility," stated Brian Kucic, VP of Business Development for R Systems. "Using the analysis model provided by Dana and the same 32-node compute cluster, we evaluated the performance of Abaqus FEA software on both Linux and Windows. Switching between the two operating systems was straightforward and the performance of Windows HPC Server 2008 was highly competitive."

Using 128 cores of a 256-core Intel Harpertown cluster with Quad Data Rate Infiniband, R Systems performed the benchmark study to evaluate distributed memory performance of Abaqus FEA on Windows. The analysis, which normally takes a little over two days to complete using eight cores, ran to completion in just under 3.5 hours.

"We were extremely pleased by the outcome of the R Systems study," stated Frank Popielas, Manager Advanced Engineering, Sealing Products Group, Dana Holding. "The combination of flexible computing clusters and the power of parallel processing enabled by SIMULIA and Microsoft will result in significant time and money savings for Dana and the manufacturing industry as a whole."

Realistic Simulation Accelerates **Safety Evaluation** of Mine Designs

Global mining company achieves significant productivity gains with 3D mine models developed with Abaqus finite element analysis software



In a step change beyond traditional processes, Abaqus finite element analysis (FEA) software is being used to enhance mine design and engineering simulation at a number of major mines around the world. In North and South America, Africa, and Australia, some of the world's biggest mining companies are applying FEA technology to evaluate safety and improve design planning, implementation, and operations.

Beck Arndt Engineering (BAE), a Sydney-based international consultancy, is a pioneer in the commercial development of engineering solutions for the mining industry. The consultancy has worked closely with engineers at SIMULIA Australia to expand the use of Abaqus FEA for mining applications.

Among the early adoptors of mine-ready FEA technology is the world's largest miner, BHP Billiton. With BAE's help, BHP has already applied this technology to evaluate mines in Canada and Australia. At the BHP Billiton Nickel West Perserverance Deeps Project in Western Australia, Abaqus FEA software is now being used to help engineer the safety and productivity of planned deep-mining operations.

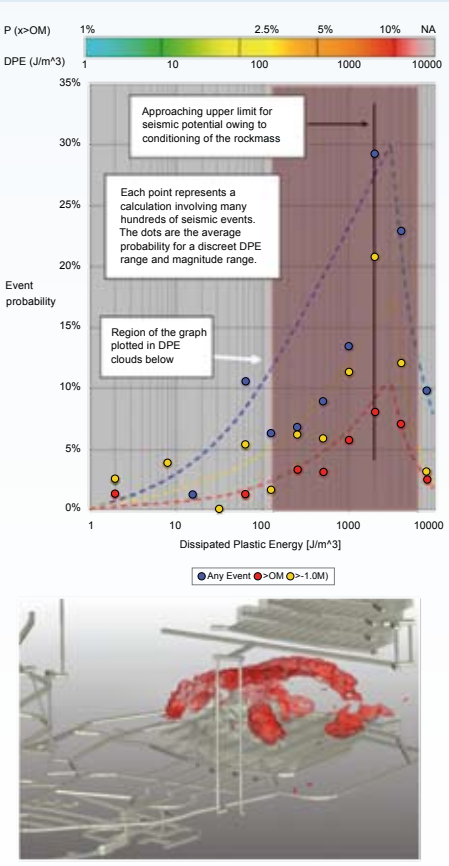


Figure 1: Simulated seismogenic zone above a developing deep mine cave, shown by calibrated Dissipated Plastic Energy.

Using measurements of site deformation and seismicity, Abaqus FEA models have been calibrated and, in a single day, used to simulate a full, three-dimensional, inelastic analysis of a mine's life cycle.

In recent years, similar applications at Debswana's Jwaneng Mine in Botswana, the Newcrest Mining Ridgeway Deeps Project in New South Wales, Australia, and Rio Tinto's Argyle Diamond mine in Western Australia have established Abaqus FEA as the leading technology for multi-scale, simulation-aided mine engineering.

Dr. Joop Nagtegaal, a pioneer in FEA and a Dassault Systèmes Corporate Fellow (retired), says that Abaqus FEA is unique in its capabilities to enable mining engineers to investigate design innovations from the drawing board to full production. "In the design stage, Abaqus models, which include rockmass volumes spanning several kilometers around the ore body and down to excavations just a few metres across, are used to compare and optimize engineering options," he said. "Then, as the mine goes into production, large volumes of data from the field are incorporated with the analysis models to allow them to be calibrated to a precision not previously available to the mining industry."

Seismic-event forecasting has become increasingly important at several sites where mining-induced seismicity is a concern. Dr. Stephan Arndt, principal engineer at the BAE Perth office, said the vast amount of analysis required to create solutions in today's competitive mining markets requires new technologies and methods.

One innovation has been the development of the Dissipated Plastic Energy (DPE) analysis method. DPE analysis has been used to develop controls for potential problems, as well as to better understand how rock masses are damaged (Figure 1).

As the size and complexity of mining problems being studied increase, engineers are facing the need to leverage high-performance computing solutions.

"The size of the models we now use in mining is unprecedented," said Dr. Arndt. "Distributed Memory Parallel (DMP) processing, using 32 CPUs with Abaqus FEA software, gives us the capacity to compare a number of different scenarios for mine-scale model simulations in a very short time. The level of detail achieved in these models allows us to calibrate deformation and rockmass damage, seismogenic potential, and ground support performance (Figures 2 – 3). Abaqus has an important role to play in mining and our analysis methods are setting new standards in this industry."

Another application of nonlinear modeling is the design of ground support. Similar to applications in tunneling and civil engineering, mine excavations are subject to high deformation (Figure 4). Not so typical are the strains and loads involved. In some mining cases, tunnels must survive in very weak rock a very short distance from massive underground excavations at great depth.

"Acceptance of FEA technology in mining is similar to the automotive industry experience, in which Abaqus has been accepted as a part of the vehicle body design process," said Dr. Nagtegaal. "Auto makers have learned that performing crash simulations of their designs with FEA software is much less costly than real-life barrier smashes, and provides a better platform for developing 'what if' scenarios.

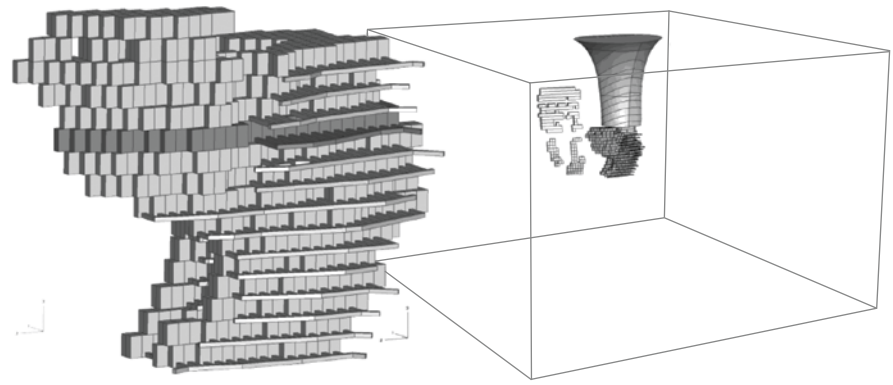


Figure 2: Geometry of finite element model for sub-level caving simulations at Perseverance Nickel Mine.

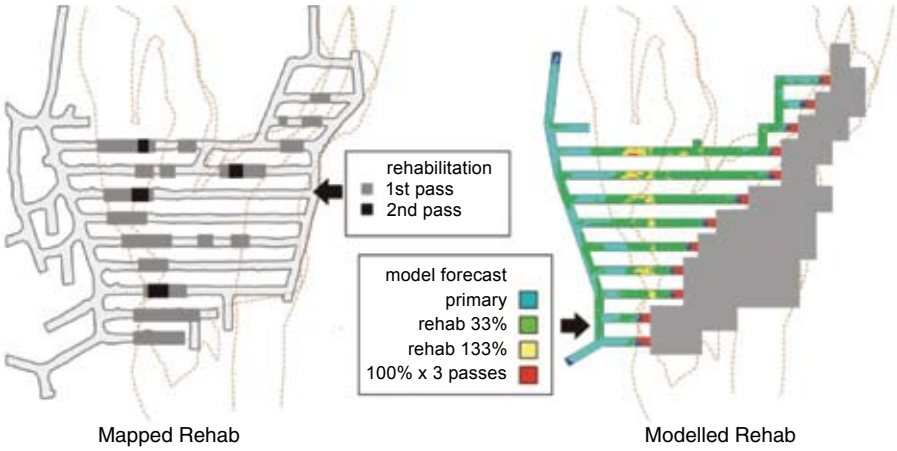


Figure 3: Calibration of ground support performance.

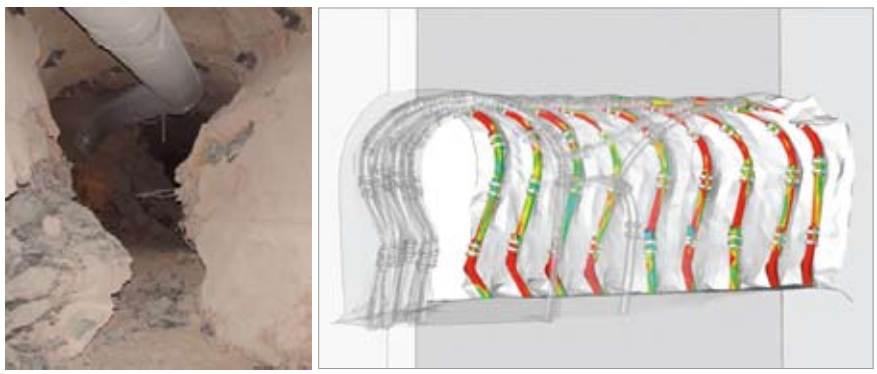


Figure 4: Simulation of extreme deformation in an intersection of weak rock using Abaqus.

Today, SIMULIA is integrating Abaqus as a tool for simulation-aided mine engineering in much the same way, and with similar achievements in cost savings and improved safety."

"To ensure the safety of people and to achieve productivity objectives at these challenging sites with unique geological characteristics, mining engineers need to think outside the box," said Dr. Arndt.

"This technology enables quick, cost-efficient analyses, which in turn facilitate the logical decision-making process necessary for the future development of mines in safe, environmentally sound and more economical ways."

For More Information

www.beckarndt.com.au
simulia.com/solutions/energy

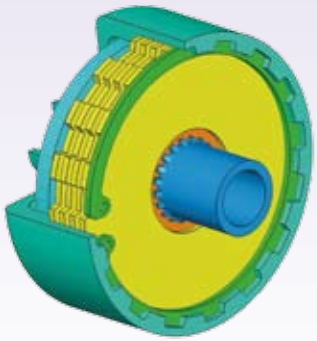
Abaqus 6.8 Extended Functionality Release

New General Contact and Modeling Capabilities

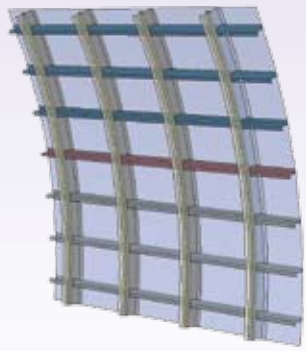
The Abaqus 6.8 Extended Functionality (EF) release enables engineers, designers, researchers, and scientists to lower costs and reduce cycle times through the realistic simulation of products, materials, and processes, including stress, impact, crush, fluid-structure interaction, thermal dynamics, and more.

Abaqus 6.8-EF includes new and improved capabilities in general contact, the modeling of spot welds, fasteners, and elastomeric foams, and computational performance. It is focused on delivering technology to solve specific engineering challenges across all industries including automotive, aerospace, electronics, energy, packaged goods, and medical devices.

"The latest release of Abaqus demonstrates SIMULIA's commitment to delivering innovative realistic simulation technology for our customers in a wide range of industries," stated Steve Crowley, director of product management, SIMULIA. "The new and enhanced features in Abaqus 6.8-EF will enable our customers to deepen their understanding of product behavior and accelerate the development of innovative products."



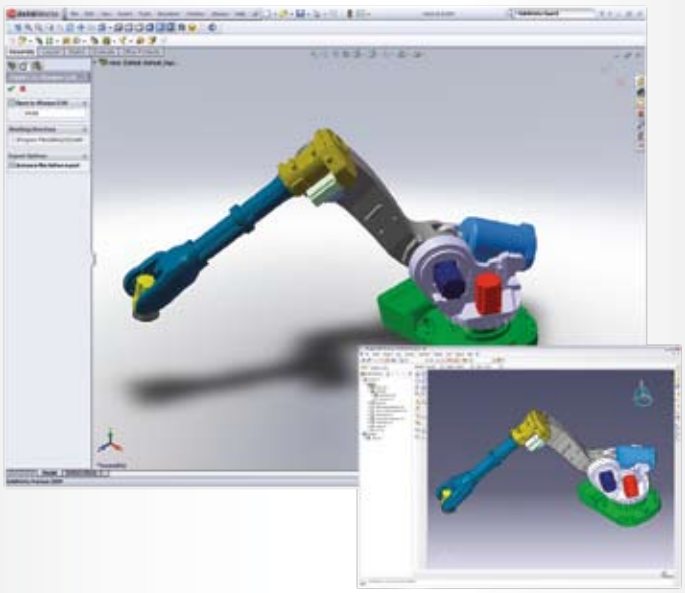
The new general contact capability in Abaqus/Standard 6.8-EF greatly simplifies contact definition for complex models with many interacting parts. Engineers can use this capability to understand the realistic behavior of products such as this automobile hydraulic clutch assembly.



A new fastener modeling capability in Abaqus/CAE 6.8-EF accelerates the modeling of point-to-point connections such as spot welds and rivets in applications such as this aircraft fuselage skin-stringer panel.

Key enhancements in Abaqus 6.8-EF:

- The new general contact implementation in Abaqus/Standard offers a simplified and highly automated method for defining contact interactions. This is useful for a diverse range of industry applications including automotive transmissions and brake assemblies, medical devices and surgical equipment, and the behavior and manufacturing of packaged goods.
- New Abaqus/CAE modeling techniques for spot welds and fasteners allow users to create attachment points that follow a model edge or conform to a regular pattern, which is useful for simulating welded components.
- A low-density foam model in Abaqus/Explicit enables automotive engineers to simulate energy-absorbing materials for crash applications. This allows users to model highly compressible elastomeric foams that are widely used in automobile passive safety systems. The capability can also be used in the design of foams commonly used in packaging of electronic devices.
- A selective subcycling feature in Abaqus/Explicit improves model performance when finely meshed components are included in an assembly. This feature enables engineers in the automotive and electronics industries to assess damage and failure using detailed 3D representations of components such as suspension control arms and ball grid arrays.
- An enhanced SolidWorks Associative Interface provides geometry transfer and maintains the relationship between SolidWorks and Abaqus models. Updates include improved performance and robustness for large assemblies and support for SolidWorks 2009.



The enhanced SolidWorks Associative Interface in Abaqus/CAE 6.8-EF provides geometry transfer and maintains the relationship between SolidWorks and Abaqus.

For More Information

simulia.com/products/abaqus_fea

SIMULIA's First New Release of Isight, Add-on Components, and Fiper for Accelerating Design Exploration

SIMULIA is pleased to announce its first new release of Isight, Add-on Components, and Fiper (3.5) since closing the acquisition of Engineous Software. These market-leading tools expand the SIMULIA portfolio of realistic simulation solutions and enable customers to combine multiple cross-disciplinary models and applications together in a simulation process flow, automate their execution across distributed compute resources, explore the resulting design space, and identify the optimal design parameters subject to required constraints.

Isight 3.5

Isight 3.5 (formerly named iSIGHT-FD) is a desktop product for creating simulation process flows, consisting of a variety of applications, in order to automate the exploration of design alternatives and identification of optimal performance parameters. Isight provides a suite of visual and flexible tools to set up simulation process flows and interconnect the computer software required to execute simulation-based design processes, including commercial CAD/CAE software, internally developed applications, and Microsoft Excel spreadsheets.

The rapid integration of simulation applications in a process flow, Isight's ability to manipulate and map parametric data between process steps, and the automation of the process execution greatly accelerate the evaluation of product design alternatives.

Additionally, by leveraging advanced techniques such as optimization, DFSS (Design for Six Sigma), approximations, Monte Carlo, and Design of Experiments (DOE), engineers are able to perform probabilistic studies and thoroughly explore the design space. Advanced, interactive postprocessing tools, such as the Visual Design Driver, allow engineers to see the design space from multiple points of view. Design trade-offs, sensitivity studies, and the relationships between parameters and results are easily understood and assessed, providing guidance to users to make the best possible design decisions.

Add-on Components

Isight comes equipped with a standard library of components, which form the building blocks of Isight process flows. A component is a container with its own interface for integrating and running a particular simulation application directly from within Isight.

SIMULIA also offers Add-on Components, an extension to the standard Isight library of components, that provide interfaces to Abaqus FEA software as well as other major third-party simulation applications and a range of design exploration/optimization algorithms. The Isight component architecture also supports the integration of customer-proprietary applications. This open integration technology is generic in order to work with a wide range of internally developed scripts, applications, and databases.

The Add-on Components offer customers great flexibility and benefit, including:

- Easy integration of your existing simulation applications in Isight
- Timely updates of high-quality Add-on Components through a release process that is independent from the release of the core Isight software
- Reduced simulation process costs

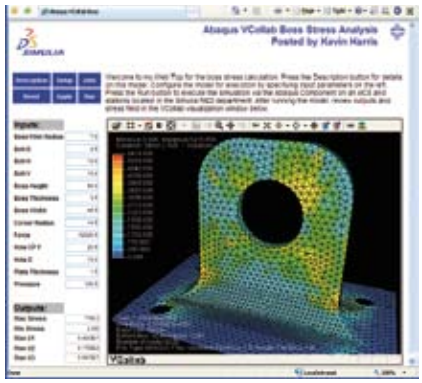
Fiper 3.5

Fiper, an add-on product to Isight, enables a group of engineers to share Isight workflows, distribute and parallelize their execution across available compute resources, and share results. The Fiper add-on can be accessed directly from Isight or from a customizable Web user interface.

Using Fiper, engineering groups are able to execute complex, multidisciplinary design processes in the most cost-effective manner to quickly deliver more competitive and profitable products to the market.

Fiper streamlines engineering design processes by:

- Seamlessly integrating with your IT infrastructure
- Leveraging your existing hardware resources as a powerful computing environment to more effectively and efficiently run complex models
- Providing a distributed product development infrastructure that allows organizations to access, execute, and reuse design tools and processes, including a Web-enabled front end



Fiper enables the execution of simulation workflows from a Web browser.

New & Improved Features of Isight and Fiper 3.5

- Users can now run more complex and larger models due to 64-bit native support.
- A new search capability has been added to Isight to assist in finding any component, parameter, or file in a simulation process flow.
- The improved Visual Design Driver enables users to view contour plots with superimposed constraint violations.
- The enhanced integration of Platform Computing's LSF with Fiper improves the reliability of distributed resource management of simulation jobs.
- Fast-running components can now use the Fiper DRM while resource-intensive work items use the Platform LSF DRM, improving simulation performance.

For More Information
simulia.com/products/isight

Human Tissue Modeling Targets Patient Comfort and Health

Researchers at Frankfurt University use Abaqus FEA for in-depth study of stress and strain on bodies at rest

Pressure sores are a costly challenge to the healthcare industry (an estimated $4 billion a year in the U.K., according to one study) and the problem may be growing globally due to aging populations.



The medical care industry and bed mattress manufacturers are highly aware of the problem and diligently looking for solutions. Researchers into body mechanics are finding that the answer goes deeper than the latest "miracle foam."

"Current techniques for pressure mapping of mattresses don't adequately evaluate the underlying supporting foam materials—or how the human body interacts with them." says Gerhard Silber, professor of Materials Science at the Center of Biomedical Engineering (CBME) at Frankfurt University.

Prof. Silber and a group of researchers at the university have taken on the challenge from the inside out, using Abaqus finite element analysis (FEA) technology, in conjunction with magnetic resonance imaging (MRI), to study the dynamics between cushion materials and human skin, fat, muscle, and bone. Their findings bring significant insight into the causes of bedsores. Their work also holds implications for biomechanical design optimization beyond mattresses to wheelchair cushions, operating room table covers, airplane seats, saddles, and even sports shoes and helmets.

The original motivation for Silber's work was a request from a mattress manufacturer looking for a foam cushion that would prevent bedsores. Also called pressure sores or ulcers, these can appear anywhere on the human body but are most often found on a person's buttocks, where up to 40 percent of body weight is concentrated when lying down.

To provide realistic simulation of the mechanics of body/bed interaction, Silber's group turned to Abaqus finite element analysis software. "With Abaqus FEA we can create computer models that let us look inside both the mattress material and human tissues to evaluate internal stresses and strains," Silber says. "This is extremely important because most pressure sores develop from deep within tissue outwards to the skin."

Modeling the human body

Abaqus FEA is proving to be an extremely useful tool for understanding human tissue response as the software provides complex material models, contact, multiphysics (for fluid-structure interaction), and high-performance parallel processing, among other capabilities.

But collecting the data needed to build, and then validate, a human FEA model requires a different methodology from what an automaker or cell-phone design engineer might use. In the world of product development, graphs showing close agreement between FEA simulations and prototype tests are commonplace because the verifying data can be derived from real-world physical testing of inanimate objects. But in the case of human tissue modeling, confirming FEA stress/strain predictions with direct measurements from deep within a living body is not physically possible.

Modeling mattress foam was a fairly straightforward process for Silber's group. The engineers obtained the data they needed for FEA through laboratory testing using a device that would load, hold, and then unload different kinds of foam samples while directly recording force and indentation displacement. This procedure ensured a distinct separation of the elastic from the inelastic material properties of the foam.

But in order to "see" the hard-to-reach human tissues they were modeling, the researchers used magnetic resonance imaging technology to provide the data they needed. First, human test subjects were MRI-scanned to obtain an undeformed tissue configuration of the buttock region. Next, loading was applied during an MRI scan. Working in an inverse fashion from the MRI images, the researchers were able to derive metrics that could be used as constraints in an optimization process to reveal the distinct mechanical properties of different tissue types.

"We needed to find the appropriate material parameters for in-vivo fat and muscle tissue that would reflect the test conditions of tissue indentation," says Silber's research associate, Ph.D. candidate Christophe Then. "So we parameterized the material constants and simulated the models iteratively until the force-displacement and simulation output coincided." Then the researchers were able to accurately describe skin/fat and muscle tissue parameters, build their FEA models describing body-support interactions, and simulate the effects of various support materials/designs on the different tissue types.

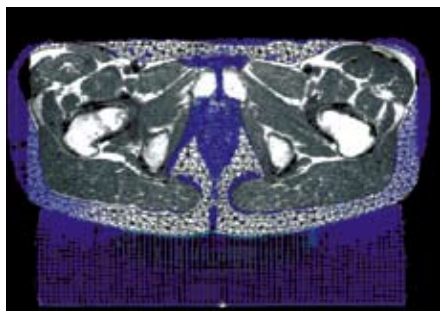


Figure 1: Abaqus FEA overlay of MRI image was used to validate the body/foam interaction model. Shown is an FEA-mesh of a cross-section of a human pelvis MRI (upper grey and white area) resting on a foam cushion (square purple meshed portion at bottom). Using imaging techniques in this way is key for verifying the accuracy of biomechanical modeling of living subjects.

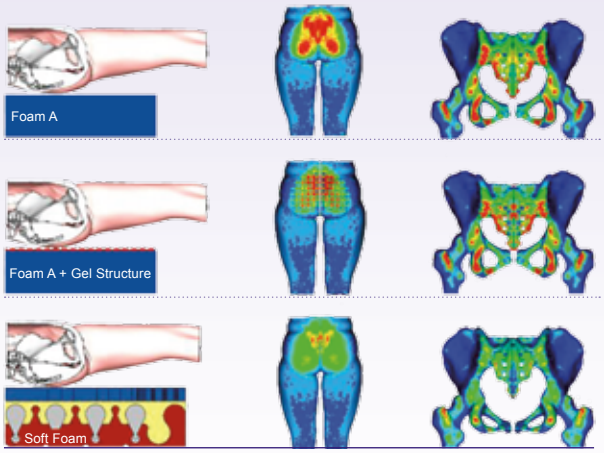


Figure 2: Body-Optimized-Simulations-Systems (BOSS) models of human pelvis and thigh resting on three different types of foam mattress (left column); corresponding Abaqus FEA images showing resulting interface stress on lower torso (center column); and FEA imagery of pelvis bones (right column) showing areas where strain is greatest. The mattress design/material configuration at lower left produces the least amount of loading.

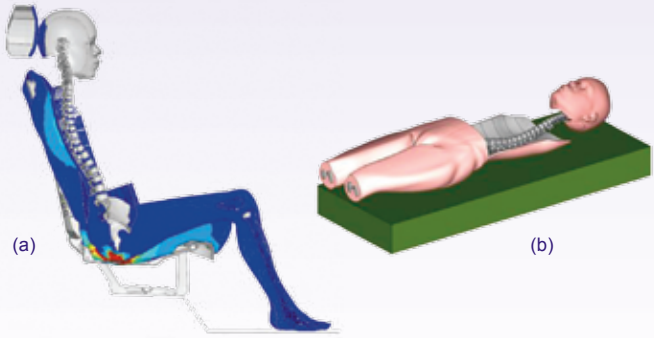


Figure 3: Abaqus FEA analysis of seated (a) human figure and recumbent (b) BOSS MODEL are used by the Center for Biomedical Engineering at Frankfurt University as part of an ongoing program to develop a research methodology that can be applied to products interacting with any part of the human body.

MRI helps validate FEA results

To validate their FEA models of body/foam interaction, the researchers again turned to MRI (Figure 1). "By superimposing a simulation result over the corresponding MRI image—both of them at the same deformed state—we were able to compare the boundaries of the human tissue and the outer surface of the foam we were testing," says Silber. "Using imaging techniques in this way is essential for biomechanical modeling; it provides key information for validation."

Prof. Silber's results clearly supported clinical observations of where bedsores arise. The Abaqus FEA results showed highest stress/strain concentration near the bones of the lower back and pelvis—the ischial tuberosity, the posterior superior iliac spine, and the sacral and tail bones—exactly below where visible bedsores are clinically observed to occur most frequently on the skin surface.

Even more important than the location of the sores was their origin within the body. "FEA showed areas of greatest stress and strain at the deep interface between muscle and bone, not in the surface skin/foam support interface," says Then. The researchers theorize that this is due to the normal "irregularities" of the human skeletal structure. "Tissue movement is restricted at the relatively small, prominent surface of a bone," explains Then. "As loading causes the tissue to displace 'around' a bone prominence, stresses and, even more significant, strains increase particularly in the immediate neighborhood of that prominence." These results are also consistent with surgical findings that show cone-shaped necroses, with the base located near the bone surface, in the majority of cases of severe deep tissue pressure sores.

"Clearly, healthcare products require better design to effectively reduce or eliminate bedsores and improve the quality of life for patients," points out Silber. "Our research is providing data that can be a foundation for that kind of design." With continued funding from foam manufacturers and healthcare companies, the team has expanded the initial scope of their work to model many different mattress configurations and materials to analyze and compare their impact on human tissue models (Figure 2). They are also studying the effects of biological variability of mechanical human soft tissue characteristics—taking into account gender, age, and physical condition—on tissue displacement under loading.

FEA enables biomechanical product development head to toe

Following the success of their work on gluteal tissue/support modeling, the team is exploring other areas in which the FEA/MRI combination can benefit the development of products for human use. "These tools can be applied to biomechanically optimize many new products for minimal stress and strain inside living tissue," says Prof. Silber. "We can now approach comfort-related questions by considering discomfort to be related to pathologically high tissue stresses and strains over a prolonged period of time."

The researchers are now extending their scope beyond the human gluteal area to larger "BOSS (Body-Optimized-Simulations-Systems) Models" in seated and recumbent postures, with the addition of leg and spine FEA (Figure 3). "Our BOSS Models let us explore such areas as mattress/heel impact and car seat vibration," says Then. "The kind of research methodology we have developed could be applied to products interacting with any part of the body such as feet and running shoes, or heads and helmets."

"Abaqus FEA with its visualization options has allowed us to get a feeling for very complex processes which one could not imagine otherwise," says Silber. "With this knowledge we can achieve a better understanding of what is actually occurring in the human body and develop new ideas that serve both comfort and health."

For More Information

www.cbme-hessen.de
prof.dr.silber@t-online.de
simulia.com/solutions/life_sciences

Realistic Simulation Assists Nuclear Power Plant Certification

From the onset of the civilian nuclear era, there has been a strong awareness of the importance of safety within the nuclear energy industry. Experts have devoted much time and effort to ensuring the integrity of reactor cores and facility containment.



Wolfgang Hienstorfer, TÜV SÜD ET

Global cooperation on nuclear safety issues is widespread. The U.N.'s International Atomic Energy Agency (IAEA) has established mandatory benchmarks for nuclear plant siting, design, construction, operation, resourcing, assessment, and verification of safety, quality assurance, and emergency preparedness. All countries with operating nuclear power facilities are expected to bring their plants up to the latest IAEA standards.

Aging nuclear facilities

An integral part of reactor safety assurance is the mitigation of facility aging. Designed for 30- to 40-year operating lives, the systems, structures, and components of nuclear plants can change with time and use. Components can wear out, corrode, or degrade; instrument and control systems may become obsolete as technologies evolve. Complicating the issue, the properties of critical materials may change through heat and neutron irradiation.

Identifying and correcting longevity issues can extend the operating license of a plant by several decades, which is why upgrading older facilities is a major focus of nuclear regulatory bodies and plant operators. In addition, new facilities are held to the highest standards of quality to ensure a lifetime of safe operation.

"The structural integrity and operational management of nuclear facilities must be secured far into the future—whatever the type or age of the plant," says Wolfgang Hienstorfer, head of the department of structural analysis at TÜV SÜD ET, a leading global technical service corporation in Filderstadt, Germany.

Hienstorfer's team independently tests, inspects, and certifies nuclear facilities for licensing by the German government. He is also chairman of the advisory group on nuclear facility aging management to Germany's Nuclear Safety Standards Commission, and a technical consultant to the IAEA on nuclear facility aging. Many of his recommendations developed during his work at TÜV have been incorporated into existing international standards.

"On behalf of the regulatory bodies, we encourage the power utilities to follow the latest relevant research findings whether they are maintaining an older plant or designing and building a new one," says Hienstorfer.

FEA assists safety evaluation

To assist in the evaluation of nuclear plant integrity, Hienstorfer's group employs Abaqus FEA software. "Abaqus is a very useful and powerful tool for many aspects of our work," says Hienstorfer. "The processes of sensitive industrial facilities are very complex, and FEA helps us evaluate the safety margins in a more sophisticated way."

TÜV uses Abaqus to analyze stress loads over a wide range of scenarios such as rapid temperature and/or pressure changes, airplane impact, earthquakes, and radiation embrittlement. The software is used to analyze everything from key mechanical components—including pumps, piping systems, vessels, supports, and tanks—to fuel assemblies, building structures, and lifting devices.

Strict standards for nuclear reactors

An ongoing focus of regulators is the reactor pressure vessel (RPV), the steel "heart" of the power plant that houses the nuclear fuel rods (Figure 1). A nuclear power plant using fission to produce steam that drives electric generators is subject to temperature and pressure stresses similar to those at any kind of steam facility. But the possibility of pressurized thermal shock (PTS) affecting a radiation-embrittled RPV is unique to the nuclear industry: bombardment from neutrons can, over time, alter the molecular makeup of the metal from which an RPV is built, making the vessel more prone to structural damage



Figure 1: A nuclear reactor pressure vessel that houses the fuel rods. Exterior view of the nozzles (with red caps) through which hot and cold water circulate into and out of the vessel. (Photo courtesy of Westinghouse)

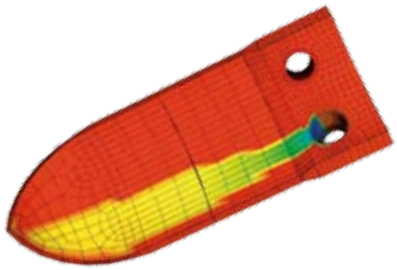


Figure 2: Cutaway view of reactor pressure vessel (RPV) at the start of a pressurized thermal shock (PTS) simulation by TÜV, using Abaqus FEA. The vessel, which normally operates at 300º C (indicated in red), is shown as cooler water (30º C) begins pouring in through the nozzle on the top right. (Image courtesy of TÜV)

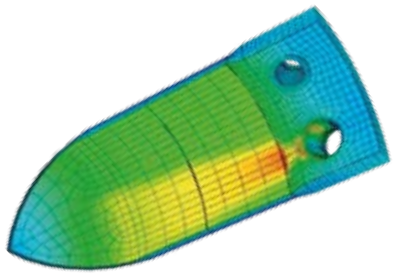


Figure 3: The same reactor vessel in pressurized thermal shock (PTS) simulation shows the stress distribution on the inner wall (from red to blues and greens). TÜV uses Abaqus FEA to evaluate the ability of RPVs to withstand such an event. (Image courtesy of TÜV)

under stress. In a classic loss-of-coolant (LOC) scenario, a broken pipe in the primary system deprives the reactor core of vital coolant, and the hot vessel (300º C) is then subjected to extreme PTS as colder water (at 30º C) is rapidly piped into the vessel to cool the core and shut the reactor down.

IAEA standards require that RPVs have a proven ability to withstand this kind of event in order to receive certification for operation. "You have to document the damage tolerance of the systems, structures, and components of a plant to pass inspection," says Hienstorfer. "FEA is integral to that analysis. FEA can be used for virtual testing to provide guidance for new designs in the early stages of product development, as well as for performance assessment of existing components under simulated stress conditions."

A typical FEA analysis of an RPV takes into account temperature transients, internal pressure fields, and radiation embrittlement behavior of the vessel during a simulated LOC event. The simulations examine stresses at vessel walls and entry points of the hot and cold water nozzles feeding into the RPV.

Modeling an RPV with Abaqus

To create their FEA models, TÜV engineers first obtained component condition data for the vessel and nozzles from nondestructive x-ray and/or ultrasound testing. Every vessel is plant-specific—in the case described here, the material was ferrite steel coated with austenitic cladding to protect the load-carrying ferrite layer from corrosion. Embrittlement of the metal over time was represented by end-of-life calculations based on existing data from irradiated material.

Next, Abaqus/CAE was used to build and mesh computer models of the vessel and the four water pipe nozzles that fed into it. Using larger, linear hexahedral elements reduced computation time for solving the global model (Figure 2), while smaller, quadratic hexahedral elements were used in the submodels (Figure 4) for more accurate depiction of stresses at the edges of nozzles.

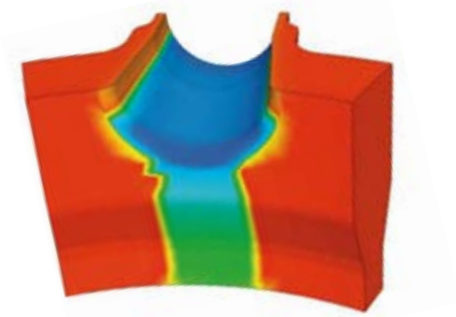


Figure 4: Abaqus FEA half-model of an RPV nozzle opening (shown as holes in Figures 2 and 3) through which cold water is quickly introduced to shut down the reactor, resulting in pressurized thermal shock (PTS). (Image courtesy of TÜV)

Simulating pressurized thermal shock

The TÜV team then used Abaqus/Standard for linear elastic simulation of the rapid cooling of the vessel, calculating the effects of a large increase in tensile stresses on the inner vessel wall. This increase is the result of two phenomena. First, the thermal conductivity of the two materials is different, so each reacts differently to the rapid temperature change. Second, the emergency injection of colder water creates a temperature plume that produces stress buildup at its leading edge (Figure 3).

The effect of the high pressures under which the system would operate was also incorporated into the models; an elastic/plastic Abaqus simulation predicted where the greatest surface and/or volumetric stresses would occur in the system. The simulations were run beyond the required tolerance levels to the point at which cracking would occur. Such data is useful for fracture mechanics analyses, and can be used in the future by inspectors, says Hienstorfer.

FEA facilitates regulatory compliance

The RPV in this example passed TÜV's simulation testing, indicating that its walls and nozzles would withstand the extreme conditions of an LOC event over a 40-year lifespan. "The Abaqus FEA calculations helped evaluate compliance of the vessel to regulatory safety requirements," says Hienstorfer.

Successful design, development, and maintenance of nuclear power facilities are challenges that must be managed from both an organizational and an engineering viewpoint, says Hienstorfer. He sees FEA as playing an integral role in both operational evaluation and ongoing monitoring of nuclear facilities to help comply with regulations designed to ensure the world's growing energy needs can be met safely.

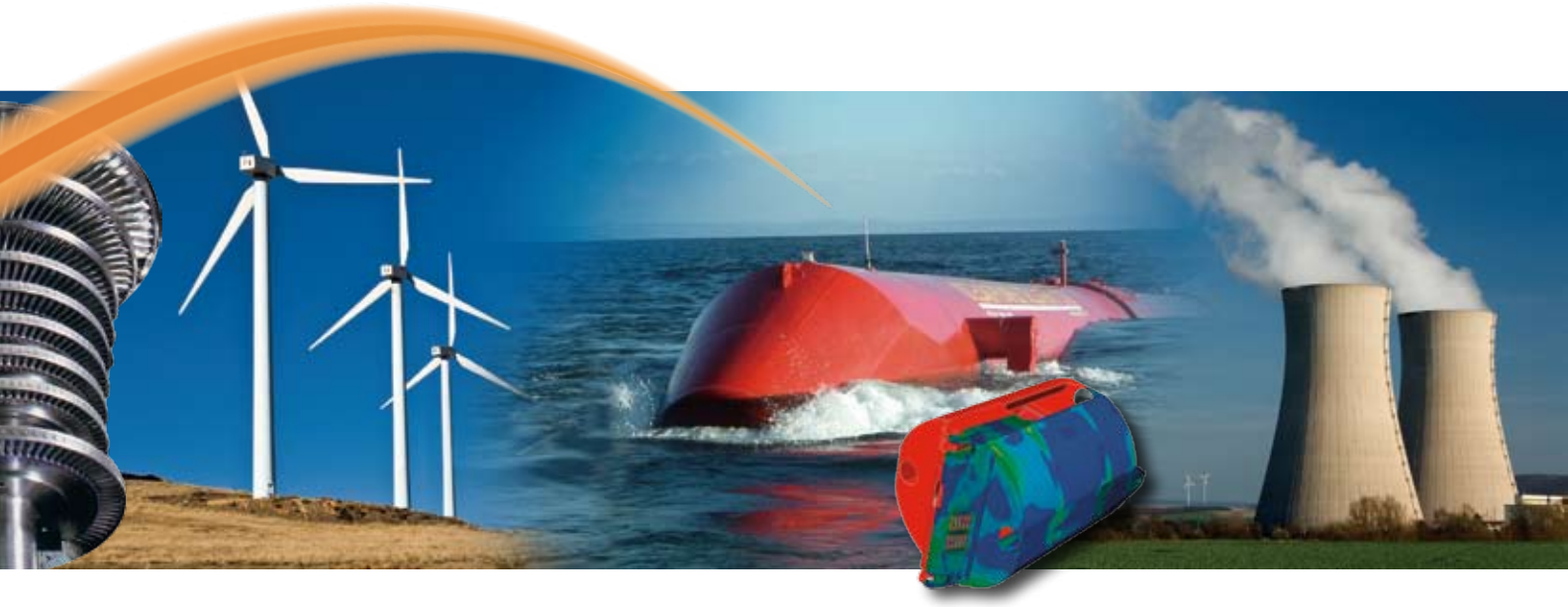
"We depend on FEA for computer modeling and virtual testing of reactor pipelines, vessels, and materials under extremes of stress and time," he says. "It definitely provides guidance to engineers building safety and longevity into their nuclear power plant designs."

For More Information
wolfgang.hienstorfer@tuev-sued.de
simulia.com/solutions/energy

SIMULIA Product Strategy for Energy Innovation

Solutions for Realistic Simulation, Design Optimization, and Simulation Lifecycle Management

Mahesh Kailasam, Energy Industry Lead, SIMULIA Technical Marketing



Energy sources are becoming increasingly diverse, and require a wide range of engineering solutions to meet industry challenges—such as extracting oil from deeper offshore locations; designing safer, longer-lasting nuclear plants; and making solar, wind, wave, and other alternative energy sources more economical.

These energy development challenges are being driven by a combination of events, including an increase in environmental awareness, the drive of various nations to be "energy independent," fluctuations in the price of oil and gas, and the rapid increase in worldwide energy usage. Every segment of this industry is faced with the demand to develop more cost-effective, reliable, and sustainable technologies to meet current and future energy demands. Energy companies are aggressively seeking to apply new and innovative engineering solutions to meet regional and world demand for energy.

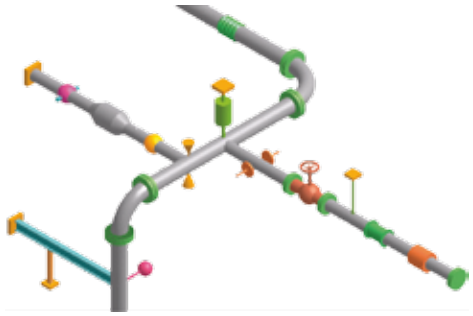
SIMULIA's realistic simulation solutions are playing a critical role in helping the industry meet these challenges. Our robust design simulation tools are helping oil exploration companies tap into deepwater energy resources. We are enabling alternative energy systems to be developed economically through fast, affordable virtual testing technology. Our solutions are also extending the use of traditional energy sources by enabling evaluation of stress, fracture, and failure of existing components under severe operating scenarios that cannot be tested in real life.

Expanding realistic simulation capabilities

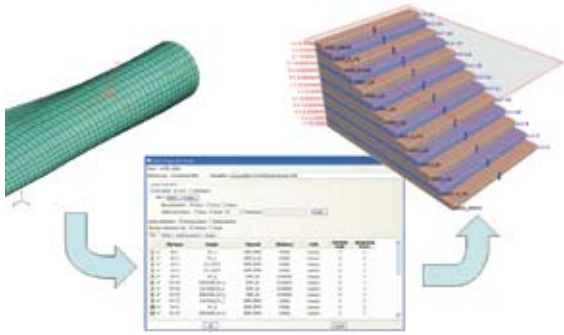
Our products, such as Abaqus FEA and Isight, are used extensively throughout the energy industry for a very broad range of applications, including oil and gas geomechanics, offshore platform analysis, gas and steam turbine design optimization, nuclear energy safeguards evaluation, wind turbine blade and tower design, concentrated photovoltaic systems for solar energy, and wave energy converter development.

Abaqus FEA is well-suited to energy applications due to capabilities such as advanced material models, general contact, implicit and explicit dynamics, multiphysics simulation (such as fluid-structure interaction, coupled pore pressure-stress, and coupled thermal-stress), composites modeling and analysis, flexible multibody dynamics, and high-performance parallel solvers.

Isight is an established industry tool for creating simulation process flows (consisting of applications such as CAD, FEA, and CFD) and automating the exploration of design alternatives to identify optimal performance parameters. Fiper is an add-on product to Isight that enables users to share process flows, distribute and parallelize their execution across compute resources, and share simulation results (see *INSIGHTS* p. 9). We have also



The number and locations of hangers in a complex power plant piping system can be optimized to meet stringent earthquake requirements while minimizing cost by using Isight with Abaqus or third-party products.



Abaqus can be used to specify and visualize composite layups with varying material properties, thickness, and orientations, capabilities that are needed for the study of wind turbine blades. (Image courtesy of Energy Research Unit – Rutherford Appleton Laboratory)

released Isight for Abaqus, which allows Abaqus users to leverage the power of Isight for design exploration and optimization.

Industry applications

The nuclear industry has used Abaqus FEA for decades because it provides accurate solutions and sophisticated capabilities, such as fracture analysis and material models for plasticity/creep analysis of metal and concrete, which meet the demanding quality standards for plant design, construction, and maintenance. It is used throughout the entire lifecycle of a plant, including evaluation of reactors, piping, and turbines; safety assessments of accident scenarios, earthquakes, or impact events; evaluation of storage options for spent nuclear fuel; and for safe decommissioning.

Wind energy engineers use Abaqus for simulating wind turbine systems and structures. Applications include analyzing wind turbine blades, towers, foundations, bearings, drivetrains, and braking systems. Many of the applications in this industry are similar to those in other industries—the evaluation of offshore wind foundations draws upon many features used by the oil and gas industry, including capabilities for soil-structure interaction and fluid-structure interaction. Blades are being made of new, lightweight composite materials that can be analyzed using extensive Abaqus modeling and simulation capabilities that have been developed for the aerospace industry. These capabilities include the definition of layups and the visualization of results, such as stresses, within individual plies. Abaqus provides a wide range of element types (such as solids, shells, and continuum shells), material models, and failure analysis techniques (such as VCCT, the Virtual Crack Closure Technique, and cohesive elements) to provide comprehensive composites simulation capabilities that enable engineers to analyze the strength and durability of blades under various operating conditions.

Isight has a strong history of use in the turbomachinery industry and provides significant capabilities that are beneficial to the development of new wind power systems. Its simulation process automation and design optimization capabilities can be applied in the analysis of turbines to perform sensitivity studies, identify optimum design parameters, and quickly meet engineering targets.

The need for SLM

To achieve confidence in simulation results, engineers must apply and reuse standard analysis methods. Additionally, with the increasing complexity of simulation models, growing use of optimization techniques, and affordability of high-performance computing, engineers are creating larger amounts of simulation-related data. The new Simulation Lifecycle Management (SLM) tools from SIMULIA enable individuals, workgroups, and large enterprises to manage simulation processes, applications, data, and results. SLM provides unique online collaboration capabilities that allow distributed engineering teams to share simulation methods, models, and results in order to make better-informed design decisions. These capabilities offer significant benefits to the energy industry as a whole, but have particular importance to the nuclear energy field, where long-term traceability of simulation results and their impact on design decisions for plant maintenance and operation is critical.

Customer-focused strategy

As our technology capabilities and product portfolio grow, it is critical that our solutions meet the needs of the energy industry. We are closely engaged with our customers to understand their processes and simulation requirements. The goal of our technical marketing team is to drive appropriate customer-requested enhancements into our products, develop strong customer alliances, and continue to expand our product portfolio as necessary to be the realistic simulation leader in the energy segment.

Innovative, cost-effective development of traditional and emerging energy sources requires the use of state-of-the art design and simulation solutions such as Abaqus, Isight, and SLM. SIMULIA's solutions are enabling engineers to evaluate real-world behavior of a diverse array of energy-generating equipment and make rapid—and accurate—performance-based design decisions to help meet energy needs today and in the future.

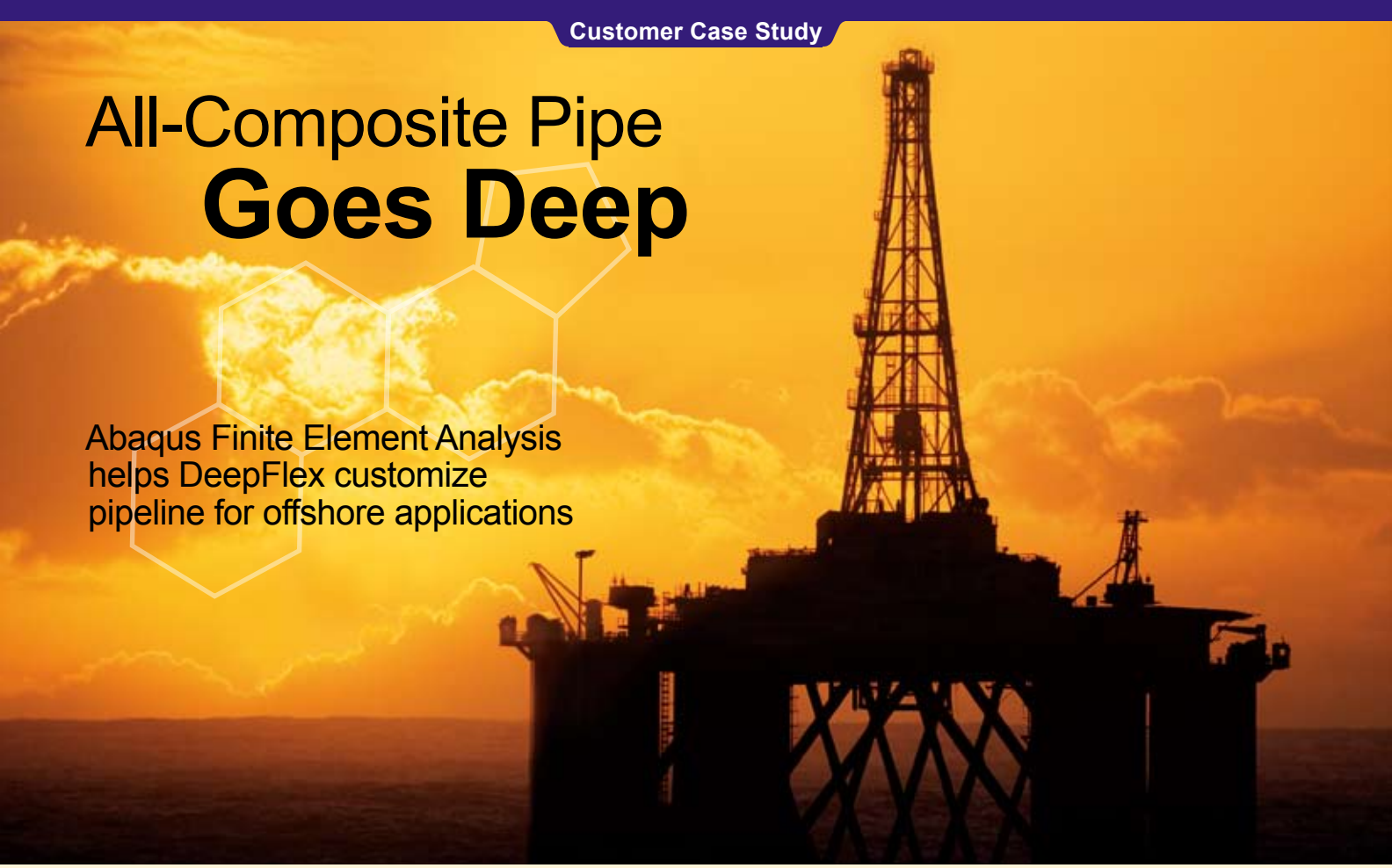


Mahesh Kailasam – *Energy Industry Lead*, SIMULIA

Mahesh is responsible for developing and directing SIMULIA strategy for the Energy Industry. He has over 10 years of experience in engineering simulation, achieved through various roles in SIMULIA Customer Services, Development, Product Management, and Strategy. He has a PhD from the University of Pennsylvania and a B.Tech from the Indian Institute of Technology, Madras (Chennai).

For More Information
simulia.com/solutions/energy

All-Composite Pipe Goes Deep



Abaqus Finite Element Analysis helps DeepFlex customize pipeline for offshore applications

Deepwater production is a challenging reality for many oil and gas companies. Limits on existing petroleum resources require the search for new fields to be conducted farther offshore and in deeper water than ever before. But operating in a harsh ocean environment, and thousands of feet below sea level, puts demands on pipelines that are much greater than those onshore or in shallower water. Traditional steel pipe can have performance limitations under such conditions.

Enter the next generation: all-composite flexible fiber reinforced pipe (FFRP), a lightweight, nonmetallic, unbonded pipe developed specifically for use in subsea and deepwater floating system applications. The need for FFRP becomes more critical as the industry moves out to 3,000-meter water depths. Constructed from extruded polymeric layers reinforced with laminated glass-fiber tape stacks, FFRP is the patented brainchild of Bruce McConkey and Mike Bryant, and has been successfully commercialized by DeepFlex Inc. It is in use in the Gulf of Mexico, with ongoing projects in West Africa and Far East Asia. “Due to its unique performance characteristics, FFRP has the potential to enable new development scenarios in deep and ultra-deepwater fields around the globe,” says Bryant, Chief Technical Officer at DeepFlex.

New material, new design challenges

Earlier generations of fiberglass-reinforced plastic bonded pipe systems have been in use for over 40 years in onshore oilfields and some shallow water applications. But DeepFlex faced the challenge of designing and producing a completely new all-composite type of pipe that could withstand the greater external hydrostatic pressures, higher internal wellhead pressures, and temperature extremes that accompany deepwater work.

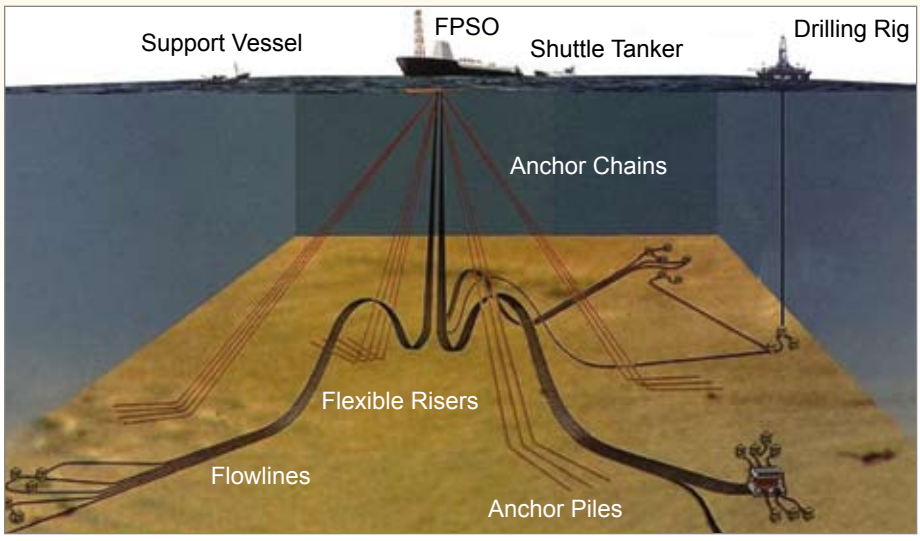
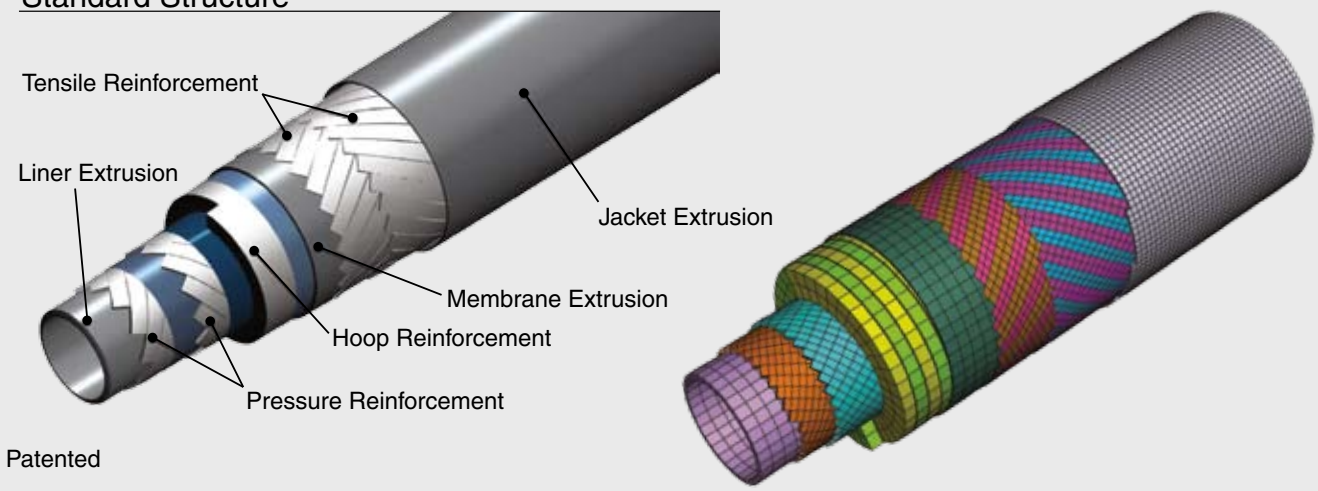


Diagram showing intended applications of DeepFlex pipe in deepwater installations. All-composite flexible fiber-reinforced pipe can be used for dynamic risers, subsea flowlines and pipelines, subsea jumpers, and surface jumpers on hybrid risers or on platform decks.



On the left is a typical structure of a standard, all-composite Flexible Fiber Reinforced Pipe (FFRP) for deepwater petroleum product recovery, detailing the multiple layers of extrusions and reinforcement that give DeepFlex pipe its strength and flexibility. On the right, Abaqus FEA model of DeepFlex pipe showing meshed representation of the layers of extrusions and reinforcement.

Metallic reinforcement wrap can strengthen composite pipe, but the highly corrosive nature of seawater limits its lifespan. Another reason to avoid metal is that composite materials are inert in the "sour gas" (hydrogen sulfide) environment of many deepwater natural gas reservoirs. When creating DeepFlex's all-composite product, "the feasibility of achieving the necessary collapse resistance without metallic reinforcement was a focus of our early developmental effort," says DeepFlex Director for Applications Engineering Shankar Bhat, Sc.D.

Tough, flexible, lightweight

To maximize the strength of its composite-only pipe, DeepFlex created overlapping layers of composite reinforcement, using multi-start stacks of specially made pre-cured unidirectional glass fiber composite tapes. The pipe is continuous, and is made in long lengths limited only by storage capacity. Performance is impressive: tests of the 2-inch pipe, for example, have demonstrated its ability to survive the pressure found in the Marianas Trench, the deepest spot in any ocean of the world. A 4-inch pipe has been tested to a collapse pressure of 10,000 psi—over 6000 meters (22,482 feet) of seawater equivalency. "Our pipe is designed to take a tremendous compressive load with a generous safety factor," says Bhat.

Pipes are offered at various internal pressure design ratings up to 10,000 psi working pressure. The FAT (Factory Acceptance Test) is carried out at 1.5 times the working pressure and burst ratings are a minimum of 2.5. No existing codes cover this new product directly, but "our goal is to meet or exceed API (American Petroleum Institute) 17 requirements when they are applicable," says Bhat.

While the plies within each FFRP stack are bonded together by epoxy resin, each stack remains unbonded from the others, ensuring true flexibility under extreme conditions and increasing fatigue resistance in dynamic applications. Unbonded construction also allows the pipe to be produced and installed in continuous long lengths in the size range of interest to offshore oil and gas operators. In addition, the composite materials act as effective insulators, keeping product flowing through pipes at colder deepwater temperatures. The all-composite makeup results in pipe that is lighter than traditional steel or other types of flexible pipe—allowing significant reduction of loads on host facilities in deep water.

FEA provides insight

The unique way that FFRP is constructed permits tailoring to the variables of the particular environment in which it will be used: a cross-section lay-up allows each layer to be custom-designed to meet specific requirements for burst, collapse, axial extension, bending, and torsion. For meeting such exacting specifications, "we needed further insight into the performance of each layer of composite to optimize pipe cross section configuration," said Bhat. To gain that insight, DeepFlex worked with structural mechanics consultants at MMI Engineering, Inc. (MMI), who applied Abaqus FEA software for realistic simulation computer modeling of FFRP.

As prototype testing began generating data during the design and development stages, DeepFlex supplied design information and pipe cross-section data to MMI for use in the numerical model creation and testing. "We were looking for a complex model able to handle the internal interactions of the materials in a more complete way," says Bhat.

"DeepFlex has a proprietary method of sizing the pipes," says Paul Jacob, Associate, MMI. "They would come to us with their pipe makeup for, say, 10,000 feet of water and 5000 psi burst pressure, and give us a cross-section and the properties we needed for our analysis. DeepFlex has an extensive prototype test program that provides overall results for product performance, but they wanted to build on this and gain

(Story continued on page 18)

an understanding of how the various components in the pipe behave under loading. This is where a tool such as Abaqus FEA can provide the needed insight into product performance."

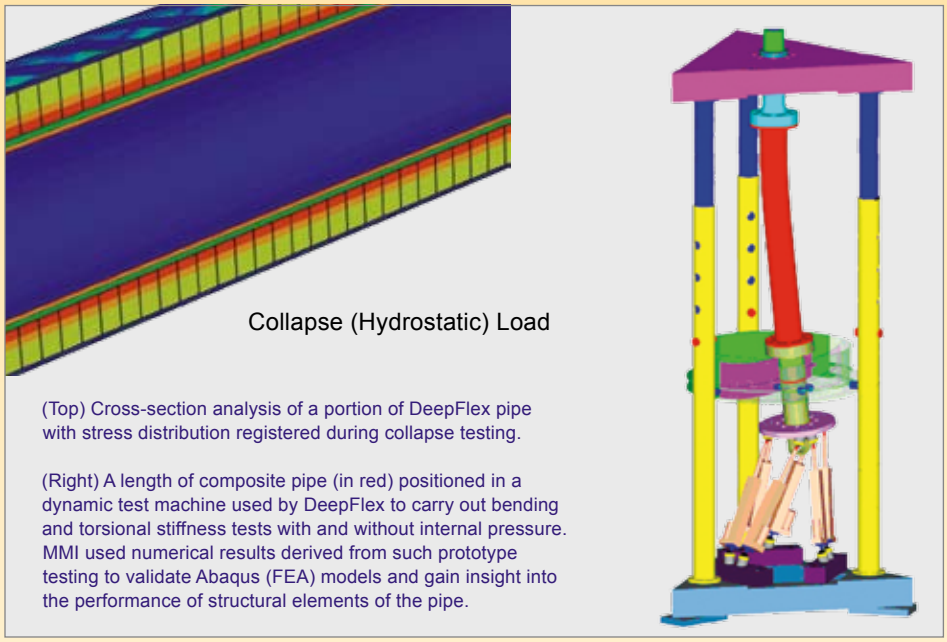
MMI used the preprocessing capabilities of Abaqus/CAE to create meshed FEA models of the pipe that could be analyzed for performance characteristics. The Abaqus analysis products were then used to conduct the simulations. "We used a combination of Abaqus/Standard and Abaqus/Explicit in this project," says Jacob. "Abaqus/Explicit was used to verify the interaction between components as it is easier to shake out numerical problems with contact. Once we had confidence in the contact interaction, we used Abaqus/Standard to complete our main set of performance analysis runs."

Modeling composites at the right level of detail

"To model the composite components of the pipe, instead of creating the individual plies, we built up orthotropic solids of each composite section," says Jacob. "We could have used Abaqus to model all the individual layers, but we did not need that level of detail at this point in our studies. Greater detail could be included at a later stage of product development if required."

MMI began their numerical analysis by performing sensitivity studies with 2D models to determine where to focus on the interactions between composite layers within a pipe structure under various loading conditions. From these studies, MMI created 3D models with each composite component modeled explicitly with contact (such as friction between reinforcement stacks) where required. Boundary conditions and loads were then applied and benchmark tests were performed to confirm that the model behavior was realistic.

The FEA model included nearly one million degrees of freedom and the analysis was run overnight on a single processor 64-bit Intel Xeon processor machine with the Red Hat Linux 64 operating system. "We used the FEA results as a starting point for establishing an understanding of the failure limits of particular pipe specifications, simulating burst and collapse tests," says Jacob. "The analysis helped us understand the mechanisms and responses of the structure under loading."



(Top) Cross-section analysis of a portion of DeepFlex pipe with stress distribution registered during collapse testing.

(Right) A length of composite pipe (in red) positioned in a dynamic test machine used by DeepFlex to carry out bending and torsional stiffness tests with and without internal pressure. MMI used numerical results derived from such prototype testing to validate Abaqus (FEA) models and gain insight into the performance of structural elements of the pipe.

Efficient modeling promotes efficient design

"We were looking to find out what the failure modes would be, how they would progress through the structure, for internal pressure, external hydrostatic collapse loads bending, torsion, and axial loads," says Jacob. "This is where the DeepFlex all-composite pipe has its advantage, because you can design it efficiently: tailoring individual components in the cross-section to meet the demands of the different layers in loading conditions such as burst or collapse. With a steel pipe, there is one material and thickness; you don't have that flexibility."

MMI developed a method for assessing failures between the individual layers, using the "Model Change" command in Abaqus to alter the states between them and applying loads to the model structure gradually until components began to fail. "This approach allowed us to develop global characteristics for load extension, and bending, that took into account the effects of burst and collapse pressures," says Jacob. "Analytically, that was the high point for me, as we were able to begin to understand the failure mechanism and load redistribution in the remaining components."

MMI provided their FEA analysis data back to DeepFlex for use as part of their design process going forward. "MMI's work was an important first step in our gaining a more complete understanding of the structural mechanics of pipe cross-sections," says Bhat. "Going forward we will continue to use FEA to deepen our understanding, which will enable further customization of the high-performance composite materials that make our pipe so uniquely suited to deepwater operations."

About DeepFlex, Inc.
Headquartered in Houston, with offices in the United States, Brazil and the United Kingdom, DeepFlex, Inc. designs, manufactures and installs premium composite flexible pipe used in the subsea oil and gas production environment. Established in 2004, DeepFlex works in the world's major offshore producing regions to meet the needs of oil and gas companies of all sizes.

About MMI Engineering, Inc.
MMI provides engineering consulting services to global clients in the oil and gas, energy, utilities, security, government, industrial and commercial markets. MMI uses state-of-the-art engineering, science and technology in combination with practical design, construction and project management experience to meet their client's unique needs.

For More Information
www.deepflex.com
www.mmiengineering.com
simulia.com/solutions/energy

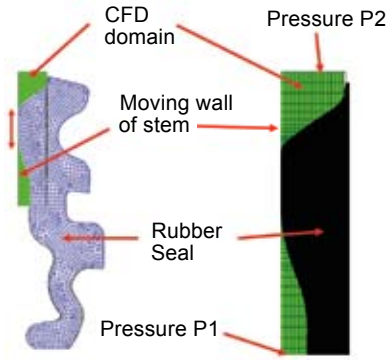
Evaluating Valve Stem Seal Performance with FlowVision and Abaqus

Ensuring that oil does not pass by valve stem seals in more than the allowed quantities is a critical design element for ensuring the quality and reliability of vehicle engines. This function of the valve stem seal is called "oil metering." Design engineers must evaluate oil flow against hydrodynamic pressure while taking into account the lubrication of the valve stem. If there is too much oil, it will cause unwanted emissions and deposits on the parts; if there is too little oil, it will cause excessive wear of the valve seat, face, and guide.

Engineers in the Sealing Products Group at Dana Holding Corporation used FlowVision-HPC from Capvidia combined with Abaqus FEA software from SIMULIA to simulate oil leakage flow via the valve stem seal at different engine operation conditions and investigate changes in characteristics due to the aging process. The unique capabilities of FlowVision-HPC, combined with Abaqus FEA, enable engineers to solve heavily coupled fluid-structure interaction problems.

In this example, the oil flow in the valve stem seal was calculated and the oil metering was represented as a function of time, engine speed, and rubber aging. The coupled simulation leveraged the Sub-Grid Geometry Resolution (SGGR) method used in FlowVision for grid generation. The SGGR method provided a natural link between the CFD grid and FEA mesh. The FlowVision Clearance Model was then used to simulate the flow in the thin channel between the stem and the seal. Step pressures were transferred to Abaqus, which calculated the resulting deformation of the seal. The coupled fluid-structure interaction analysis enabled Dana engineers to identify design modifications to improve sealing performance.

"The coupled Abaqus-FlowVision simulation mimics exactly the physics and experimental setup, giving us faster results at significantly lower cost," stated Frank Popielas, Manager of Advanced Engineering for Dana's Sealing Products Group. "We can simulate the use of new materials and design configurations



Fluid-Structure Interaction occurs in the zone of large seal deformation contacting the oscillating valve stem through a thin oil film. On each FSI iteration, step pressure calculated in FlowVision is transferred as a new load case to Abaqus, which calculates resulting deformation of the rubber seal.

to evaluate their characteristics and overall performance. Comparative studies such as ranking can be performed to get better insight of the performance in different operation conditions and design variants."

For More Information
www.capvidia.com/cfd-simulation

SIMULIA Hosts Sixth Annual Partner Summit

More than 60 business and technical representatives from 40 companies spanning the SIMULIA partner ecosystem gathered at our world headquarters in Providence, R.I. in October 2008 for the sixth annual SIMULIA Partner Summit. This growing event continues to attract many recurring participants and new members of our ecosystem each year, which is a testament to the business value our alliances programs bring to our partners and our customers. The Partner Summit provides participants with a venue for strengthening relationships with SIMULIA, sharing experiences with other partners, and getting an intimate view of our brand strategy and product initiatives.

At this year's meeting, we were especially pleased to be able to host and welcome former Engineous Software partners into the SIMULIA ecosystem. As outlined for attendees during presentations, the Isight and Fiper products and technologies that we acquired from Engineous will provide our



Representatives from 40 companies attended the 2008 SIMULIA Partner Summit. Pictured, left to right: Stefan Dietz, Intec GmbH; Ken Won, SGI; Craig Collier, Collier Research Group; Malcolm Panthaki, Comet Solutions Inc.

partners with exciting opportunities to create new solutions integrated with Isight and our open platform for Simulation Lifecycle Management. "Our partners are enthusiastic about having the opportunity to expand their solutions around new SIMULIA products," stated Tom Battisti, Director of Alliances. "Our mutual customers will most certainly enjoy the benefits of new partner-authored, tightly integrated solutions with Isight and SIMULIA SLM."

"The SIMULIA Partner Summit clearly demonstrates SIMULIA's commitment to openness and working with a wide variety of partners," commented Prasad Mandava, CEO of Visual Collaboration Technologies. "As a former Engineous partner we are extremely pleased to discover the opportunity to grow our relationship even further with SIMULIA."

The SIMULIA Partner Ecosystem

SIMULIA has an extensive and mature partner ecosystem that is fueled by strong relationships with more than 125 independent software, technology, and research and development partners. Through these alliances, SIMULIA supports the development of best-in-class solutions that increase customer productivity.

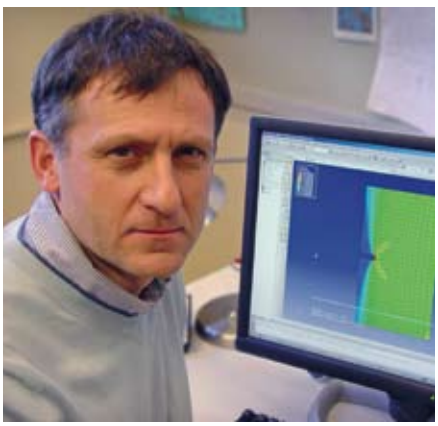
For More Information
simulia.com/alliances/alliances

Purdue Grad Students Study Computational Fracture Mechanics

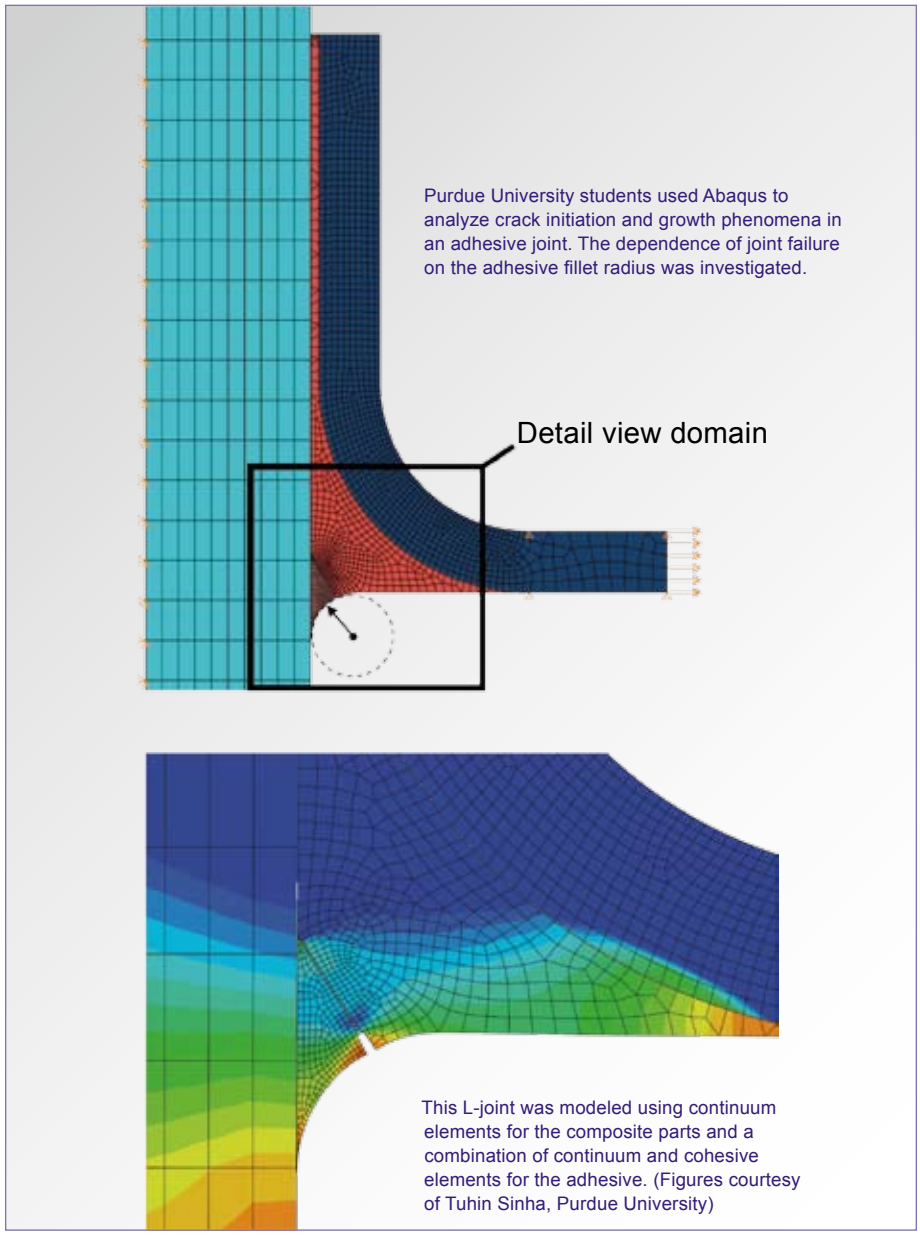
During the Spring 2008 semester, Professor Thomas Siegmund of Purdue University's School of Mechanical Engineering introduced a new generation of engineers to computational methods for material failure. The 17 students in Professor Siegmund's graduate-level "Computational Fracture Mechanics" class came from several engineering sub-disciplines, including mechanical, aeronautics and astronautics, and civil engineering. From continuum mechanics concepts to advanced constitutive equations, Professor Siegmund's students spent the semester learning how to develop and apply computational methods to predict failure in composite engineering structures.

In addition to the usual lectures associated with a graduate-level course, Professor Siegmund provided his students with extensive hands-on classroom experience using Abaqus FEA. His students used Abaqus Student Edition for much of the initial computational work, and then later experimented with more complex models using the Abaqus Research Edition. Professor Siegmund introduced his students to the use and coding of user-defined subroutines for Abaqus, line-by-line coding of input files, and modeling with Abaqus/CAE.

"Abaqus Student Edition is an outstanding tool," stated Professor Siegmund. "I was able to introduce students to the software quite readily, and after a short time the students were able to develop their own models. It was also extremely effective in developing homework assignments."



Thomas Siegmund, Professor of Mechanical Engineering, Purdue University



Purdue University students used Abaqus to analyze crack initiation and growth phenomena in an adhesive joint. The dependence of joint failure on the adhesive fillet radius was investigated.

This L-joint was modeled using continuum elements for the composite parts and a combination of continuum and cohesive elements for the adhesive. (Figures courtesy of Tuhin Sinha, Purdue University)

Professor Siegmund's course included an independent semester research project. Each student was assigned the review of a recently published paper on the use of adhesive bonding, and was then required to use Abaqus to create a finite element model of a related adhesive joint, leveraging the software's cohesive zone modeling capabilities to analyze crack initiation and crack growth phenomena. "Though fracture mechanics concepts are on the more advanced spectrum of applications, Abaqus was a tremendous tool. Students used it to extract stress intensity factors, compute crack growth directions, compute J-integrals, and investigate path dependence," continued Professor Siegmund. "Most interestingly, the students were quickly able to develop crack growth models with the use of the cohesive zone elements. Many of the students are using these concepts in their graduate work."

To accompany the lectures, Professor Siegmund made his course materials available to students as electronic documents, enabling students to keep them as a reference when using Abaqus in their future careers.

For More Information

engineering.purdue.edu/tsgroup
simulia.com/academics

Georgia Tech Students Use Abaqus in AHS Helicopter Design Competition

A team of 12 graduate students from the Georgia Institute of Technology's Daniel Guggenheim School of Aerospace Engineering used Abaqus while designing their entry in the American Helicopter Society (AHS) International's 2008 Annual Student Design Competition. The competition, which challenges students to design a vertical lift aircraft that meets specified requirements, provides a practical exercise for engineering students at accredited colleges and universities while promoting student interest in vertical flight technology.



The Georgia Tech students used Abaqus for CATIA V5 to perform a static fuselage analysis on the Athena, a short-range, medium-speed, five-seat "SMART-COPTER."

The Georgia Tech team's goal was to design a new, environmentally friendly helicopter that would be capable of performing multiple missions for a variety of customers—efficiently and inexpensively—while also meeting or exceeding established emissions and noise regulations. The result of their efforts was the Athena, a short-range, medium-speed, five-seat "SMART-COPTER" that was capable of vertical takeoff and landing from an unprepared area in less than 10 minutes from being positioned on a heli-surface.

The Athena was designed using an integrated product and process development methodology that considered the full life cycle of the design, from manufacturing to operation to maintenance. The team used a combination of SIMULIA, CATIA, DELMIA, and ENOVIA software from Dassault Systèmes to rapidly incorporate design modifications based on the results of preliminary analysis.

Fuselage & Landing Gear Analysis

The team decided to design the fuselage and subfloor with composite and foam-core materials due to weight savings, improved reliability, and the flexibility of composite materials, along with emerging developments in the areas of manufacturing and disposal.

Utilizing Abaqus for CATIA, a finite-element model of the fuselage was created for preliminary analysis. Initial static cases were run to determine critical load paths. The team simulated a 3.5g maneuver with loads applied from the main rotor, tail rotor, horizontal tail, and vertical tail. The model lent itself to continued static and dynamic testing. Loads placed on the global FEA model substantiated the strength of local fittings and other detailed parts.

When designing the landing gear, the team chose a skid landing gear—a simpler, lighter solution that provides energy savings through ease of manufacture and overall reduced power consumption. A trade study between structural steel, naval brass, tungsten carbide, aluminum, and composite landing gear was conducted. Structural steel was selected because of its favorable fatigue and energy-absorption characteristics. The skid landing gear was designed using hollow circular skid tubes and cross beams. To reduce drag, lightweight composite fairings were incorporated into the design.

The Abaqus FEA product suite was used to perform an analysis of the skid landing gear during a virtual drop test. The skid and cross beams were modeled as beam elements. A yield strength and ultimate tensile strength of 310 MPa and 517 MPa respectively with a Poisson ratio of 0.33 were assigned to the landing gear. The ground was modeled as a shell with a contact friction coefficient of 0.5. Using four rigid beam connectors, the landing gear was attached to a rigid node with assigned mass and inertial properties from the CATIA model.

The drop test analysis was conducted in accordance with both civilian FAR 27.725 requirements and US military requirements. Analysis cases were completed for several landing conditions, from a drop height of 27 in (0.69m), corresponding to an impact at 12 ft/s (3.675m/s). Based on the analysis, the team determined that the landing gear is capable of meeting the performance requirements for all landing gear conditions.

The efforts of the student team paid off—Georgia Tech captured second-place honors in the graduate category of the 2008 competition.

For More Information

http://www.vtol.org/pdf/studentDesign2008/grad_gaTech2008.pdf
simulia.com/academics

High Quality Support – Customer Satisfaction is Our Top Priority

Local Expertise Combined with 24/7 Online Support System Helps SIMULIA Provide the Best Technical Support in the Industry

Many of our customers consistently rate SIMULIA as better than other technology suppliers in providing high-quality customer support. Some of the reasons for the high ratings include our commitment to internal training of our support teams, the dedication of our support teams to providing detailed answers, and the proximity of our local support offices to our customers. With SIMULIA customer support professionals in more than 30 offices worldwide, our customers have access to experts who can speak their native language and who understand their unique industry challenges.

Our regionally-based professionals are supported by our headquarters-based Customer Services team in Providence, Rhode Island. Our headquarters support staff is well trained in the use of our software, has ready access to the SIMULIA development team, and possesses excellent industry knowledge—making it possible for our distributed support teams to provide quick answers to your most challenging simulation problems.

Another significant feature of our support offering is the convenient access to the SIMULIA Online Support System (SOSS). The SOSS is a password-protected Web-based support system that is available 24 hours a day, seven days a week. This system provides a self-service knowledge database where you can get answers immediately to many technical questions about our product suite.

Knowledge database articles are prepared by support engineers and developers of SIMULIA solutions, so you can expect in-depth and detailed information. A search engine allows you to drill down quickly to the answers specific to your problem. Currently, there is an extensive database of articles for the complete Abaqus FEA product suite. We are quickly adding articles for Isight, Fiper, and SLM.

Our online system also enables you to submit support questions, upload files for troubleshooting, and search the history of your support requests and related answers. Within the online system, you can also make requests for software enhancements, access the Abaqus Automation Portal, download SIMULIA Tech Briefs, view Web-based training videos, and view up-to-date product release information. We are planning to provide access to online support for users of our newest products within the coming months.

The members of our global Customer Support team are the backbone of our company. In addition to being focused on helping with your immediate needs, they are passionate about sharing their expertise over the long run to help you become more successful in the use of our realistic simulation solutions. As evidenced by the positive comments from our customers, it is truly our people who make SIMULIA's Customer Support the best in the industry.

"SIMULIA East has always been very prompt in their responses to my questions. When a particular applications engineer cannot satisfactorily respond or find a solution to my problems, they are quickly escalated to someone who can. Keep up the good work!"

– Brett Lussier, EMO Labs, Inc.

"The German offices in Aachen and Munich deserve an equivalent of 3 Michelin stars – there is no better team out there."

– Anders Winkler, IMS Gear GmbH



SIMULIA Customer Support and Service professionals from around the world gathered in Providence, Rhode Island in January 2009 to increase their knowledge of the technical details and industry applications for all SIMULIA products, including Abaqus, Isight, Fiper, and SLM.

Resources

Support Services Information:
simulia.com/support/support

Login or create a SOSS Account:
simula.com/support/mysupport

Regional office information:
simulia.com/locations/locations



2009 SIMULIA Customer Conference

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

Jaguar Land Rover and Abbott Vascular to Deliver 2009 SCC Keynotes

We are pleased to announce that our invited keynote speakers will be Mark Stanton, Group Chief Engineer – Vehicle Engineering & Attributes, Jaguar Land Rover, and Kelly Pike, Advisor, Research & Development, Abbott Vascular. They will provide insight into how realistic simulation is being used at their respective companies to drive research and innovation, provide performance insight, and help build better products in less time.

Who Should Attend

All users of Abaqus, Isight, Fiper, and SLM are encouraged to attend the 2009 SCC. This year's conference expands on our rich tradition of providing a valuable forum to learn how experts in engineering and academia are applying the latest simulation technology and methods to enhance product development.

Customer Presentations

Our customers' commitment to presenting their strategies and applications is the reason for the ongoing success of the SIMULIA Customer Conference. This year's agenda will feature technical presentations by engineers from more than 70 manufacturing and research organizations, including: The Boeing Company, The Coca-Cola Company, Corus RD&T UK, Edwards Lifesciences, Foxconn International Co., General Motors, GN ReSound, Halliburton, Honda R&D Co., Kimberly-Clark Corporation, Rolls-Royce plc, Samsung Electronics Co., Tetra Pak, and many others.

Advanced Seminars

On Monday, May 18, SIMULIA will offer four Advanced Seminars that will enable you to advance your knowledge and skills. The Advanced Seminars topics for your selection are:

- Calibrating Material Models for Improving Simulation Results
- Solving Contact Problems with New Capabilities in Abaqus
- Advanced FE Modeling and Process Automation with Abaqus/CAE
- Performing Process Automation and Design Optimization with Isight

Complementary Technology

SIMULIA partners will exhibit and provide presentations on their complementary technologies for simulation and computing. Microsoft is the premier sponsor for the 2009 SCC. Other sponsoring partners currently include: AVL, Beta CAE Systems, Bull, DatapointLabs, e-Xstream engineering, FE-Design, Granta Design Ltd., HBM-nCode, Hewlett-Packard, Safe Technology Ltd, Simulayt Limited, and Zentech International Ltd.



Networking at the Brewery

The 2009 SCC venue, while today a fully modern conference center, traces its brewery heritage back some 250 years. Today, the Brewery provides modern amenities and state-of-the-art audio-visual technology to ensure that the 2009 SCC will be comfortable as well as entertaining. There will be ample time during the conference to interact with your peers, our partners, and SIMULIA managers. Your conference registration includes two evening receptions, lunch each day, and refreshments during the breaks.

Early Bird Registration Is Now Open!
Register before February 27 and save!

Registration Fees	Early Bird	After 2/27
Conference Only	$895	$995
Advanced Seminar	$425	$425

On Wednesday evening, we will set sail for a scenic cruise on the River Thames! You'll be treated to an elegant dinner while enjoying the views of London's most iconic sites aboard the luxury Silver Sturgeon yacht.



Conference Proceedings



A valuable benefit of your attendance at the SCC is the annual Conference Proceedings. You will receive a high-quality, bound proceedings book and companion CD-ROM containing the customer papers prepared for the conference.

Register Today!
simulia.com/scc2009



Up to your eyeballs in simulation data?

Simulation Lifecycle Management from SIMULIA helps engineers and scientists organize and quickly find simulation data. SLM helps you document and automate best practices with tools that capture and reuse the intellectual property generated by simulation—which saves time, lowers costs, and maximizes return on investment.

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**

SIMULIA

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, 2009.