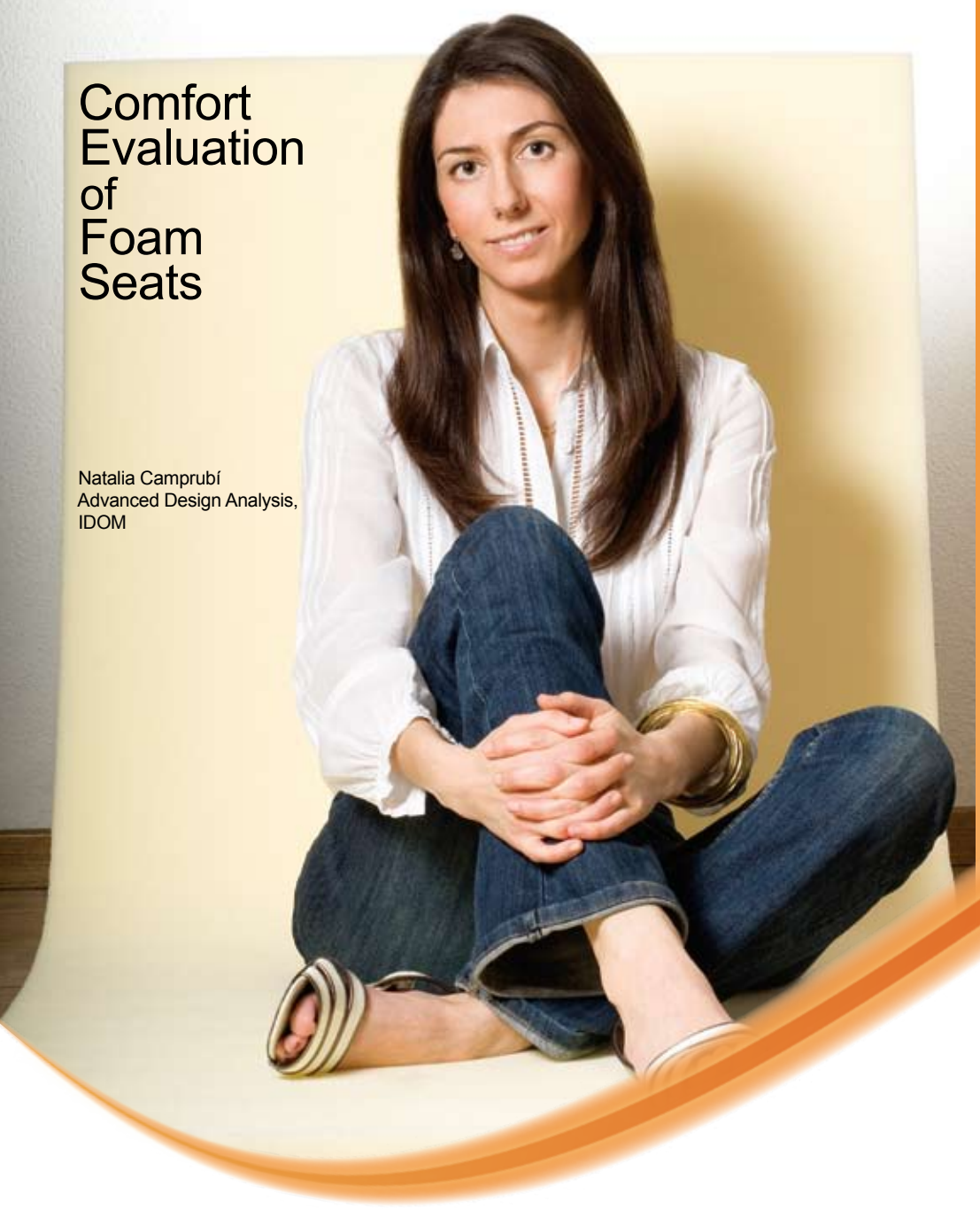
INSIGHTS

4
06 2008
Dassault Systèmes Realistic Simulation Magazine

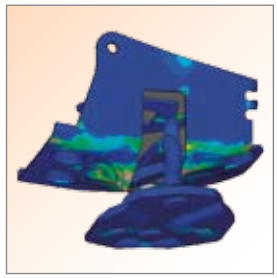


Comfort Evaluation of Foam Seats

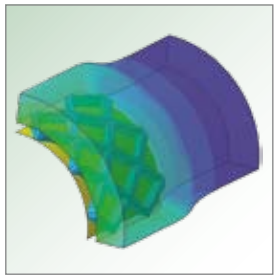
Natalia Camprubí
Advanced Design Analysis,
IDOM



Safe Train Design
Stadler Rail Group



Car Lock Safety
Kiekert



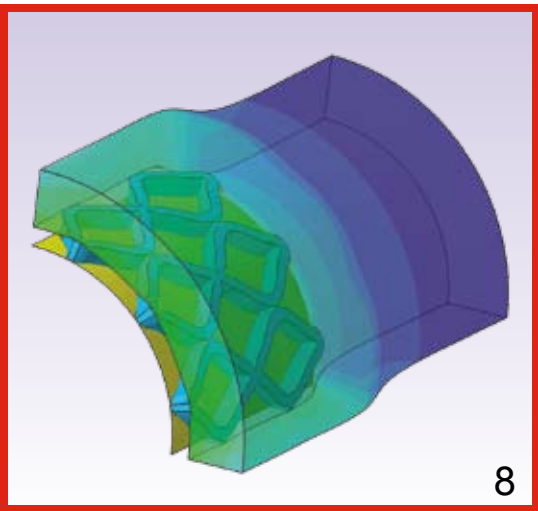
New Simulation Capabilities
Abaqus 6.8

SIMULIA

INSIGHTS

May/June 2008







Inside This Issue

12 Cover Story

Comfort Evaluation of Foam Seats Using Realistic Simulation

8-9 Product Update

- Introducing Abaqus 6.8
- Abaqus for CATIA V5 R18 SP4

6 Customer Spotlight

Realistic Simulation Makes a Safe Impact on Train Design

In Each Issue

3 Executive Message

Dale Berry, Director, Technical Marketing

4 In The News

- Industry Press Coverage
- Scott Berkey New CEO of SIMULIA
- Keeping the Arctic Safe
- Abaqus BioRID II Crash Dummy Model

10 Product Tutorial

Iterative Design Process with Abaqus for CATIA V5

16 Customer Case Study

Kiekert Locks onto Car Door Safety with Realistic Simulation

19 Alliances

- AutoForm Joins DS Software Community
- Windows Compute Cluster Server

20 Academics

- Loughborough University Studies Soccer Balls
- Rutgers University Accelerates Learning FEA Principles
- Cedarville University Researches Implant Stability

22 Services

- A New Era for Services
- Peter Nannucci Recognized by Boeing

23 Events

- Webinars
- RUMs

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:
Iván Alonso (Centro Tecnológico Grupo Copo), Dale Berry, Natalia Camprubí and Fernando Rueda (IDOM), Matt Dunbar, Brad Heers, Timothy L. Norman (Cedarville Univ.), The Parker Group, Mike Ricci, David Shreiber (Rutgers Univ.), Alois Starlinger (Stadler Rail Group), Thomas Waldmann (Kiekert), Jon Wiening

Graphic Designer:
Todd Sabelli

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

Realistic Simulation for Industrial Applications



Dale Berry
Director,
Technical Marketing,
SIMULIA

As my association with Abaqus FEA software enters its third decade, I'd like to give you a view of the evolution of the strategic planning behind our technology. I began using Abaqus software in 1988 as a support and consulting engineer with the company's Midwest distributor. Today, I am the director of SIMULIA's growing Technical Marketing team. These roles (and others in between) give me a unique perspective on how Abaqus technology has evolved and where it is headed.

In the late 1980s and early 1990s, Abaqus was well-established as the premier general purpose software for nonlinear FEA. While there was some adoption of nonlinear simulation in production environments, most companies used linear analysis methods for workflows that Abaqus was not specifically targeted toward. Most applications for Abaqus continued to be point solutions developed by experts in industrial R&D centers.

By the mid '90s, Abaqus FEA software was becoming adopted as a mainstream production tool, especially in Automotive Powertrain and Tire simulation. This was due in part to key influencers—both at customer sites and internally—who were driving development of new nonlinear Abaqus technology such as mis-matched meshing across contact surfaces, modified tetrahedral elements for contact, gasket elements, and bolt loading support—all revolutionary functionality at the time. Our team from the Midwest made yearly trips to the Rhode Island headquarters to give market-driven "core dumps" to the company's principals. I'm proud that our input played a role in the evolution of Abaqus technology targeted to solve industrial applications.

In 2001, an industry-focused approach to our product planning was initiated by the company's new executive team. Our Midwest consultancy also became a part of SIMULIA, and I joined the newly formed strategy and marketing organization. We began to develop industry "roadmaps," with an initial focus on Automotive Noise & Vibration and Aerospace Composites, areas dominated by traditional linear methods. It was our goal that Abaqus would provide complete functionality, including linear, nonlinear, and multiphysics capabilities, to solve a range of industry workflows and to enable the evolution of the associated methods needed to get closer to real-world behavior.

We believed that functionality such as the advanced Abaqus/AMS eigensolver, faster modal dynamics, and mesh-independent fasteners were key to enabling Abaqus to be used for both linear and nonlinear analysis workflows. The fruits of these labors are apparent today with the accelerating adoption of Abaqus Unified FEA for production use in both the Automotive and Aerospace industries—as demonstrated by the 2007 announcement that Airbus is accelerating the use of Abaqus for nonlinear structural analysis. At the 2008 Abaqus Users' Conference, The Boeing Company presented a broad overview of their use of Abaqus, especially for composite applications, and Toyota Motor Company presented how the company is evolving simulation methods to achieve high-fidelity results.

We realized that the technology and methods created to solve a specific application in one industry can—when properly implemented—be applied in other industries. Proximal workflows (for example, Electronic product drop simulation is proximal to Automotive crashworthiness) have helped establish Abaqus as the de facto technology leader in electronics packaging. We are taking our robust technology, innovative methods, and proven simulation workflows and applying them to design challenges in many industries. At the 2008 AUC, we held several industry-focused Special Interest Groups—not only to share our current status and future directions, but also to gather specific industry feedback from our customers.

Today, the SIMULIA Technical Marketing team is growing—and, in addition to Automotive and Aerospace, we are developing roadmaps for High Tech (Electronics), Energy (Offshore Oil and Gas), Consumer Packaged Goods, and Life Sciences (Medical Devices). From our experience, we know that these roadmaps will help us deliver increased value to our customers by providing the functionality and methods for the efficient and accurate simulation of industry-specific workflows. All of us at SIMULIA welcome your input and encourage you to participate at your upcoming Regional User Meetings. We look forward to working closely with you to solve your industrial challenges.



Industry Press Coverage













BENCHmark Magazine
January 2008, pp. 20–23
Building Blast Simulation and Progressive Collapse Analysis
University of Florida researcher Ted Krauthammer, Ph.D. teamed up with SIMULIA to write about using Abaqus FEA to simulate the effects of an explosion on a multi-story building. Such analysis aids the design of stronger structures without expensive physical testing. The authors conclude that current building design guidelines may not be stringent enough for severe blast loading on existing steel structures.

Mechanical Engineering
February 2008, pp. 39–40
No Climbing
Abaqus FEA software helped Stadler Rail Group of Switzerland satisfy tough new safety standards and fulfill an order of train cars in record time. Stadler designed a crash module with an anti-climb feature that prevents cars from riding over each other and damaging the passenger compartment. Engineers validated their design using a combination of dynamic physical testing and realistic simulation (see *INSIGHTS,* p. 6).

The Structural Engineer
February 2008, pp. 26–28
Improving Bridge Performance with FEA Software
This U.K. publication featured work by Penn State University Civil and Environmental Engineering associate professor Daniel Linzell that employs FEA to model and study the structural behavior of bridges. Linzell's research group found that Abaqus' advanced software provides the capabilities to incorporate nonlinearities that can more accurately account for the real stresses and deformations that impact the performance and service life of a bridge. Simulation results can also help with maintenance—or even forensics in the event of failure.

NASA Tech Briefs
February 2008, pp. 38–40
Creep Analysis of Lead-Free Solders Undergoing Thermal Loading
With the phasing out of lead in the electronics industry, researchers at Worley Parsons PTE Ltd., Singapore are using new methods to evaluate alternatives to traditional solder. Using Abaqus/CAE to develop a Ball-Grid-Array model of an electronic device, they performed creep analysis on the model to estimate fatigue life under thermal cycling and identify the areas under the greatest strain.

Machine Design
April 10, 2008, pp. 92–94
Quantifying Comfort
Spain-based researchers at IDOM and Centro Tecnológico Grupo Copo (see *INSIGHTS* cover story, p. 12) found that Abaqus software could model, and accurately quantify, the interaction between a foam seat and a weighted human form. Designing foam car seats for comfort poses a challenge to engineers because of the highly complex nature of the material and the difficulty of quantifying such a subjective variable as "feel."

Desktop Engineering
April 2008, pp. 50–52
SIMULIA Asks: Is SLM for You?
SIMULIA product manager Paul Lalor provides an in-depth Simulation Lifecycle Management (SLM) technology overview in this three-page feature article. The vision for SLM is to bring order to simulation processes and provide technology that ensures data integrity, traceability, knowledge capture, and collaboration. The end goal is to help organizations leverage their simulation assets more effectively.

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.

Scott Berkey New CEO of SIMULIA



On March 31, 2008 it was announced that Scott Berkey was named as the new CEO of SIMULIA, Dassault Systèmes' brand for realistic simulation. Previously, Scott was the Vice President of Worldwide Operations for SIMULIA. He replaces Mark Goldstein, who will continue working for Dassault Systèmes in a part-time strategic advisory capacity.

"Scott brings an impressive background of industry and management experience to his new role that will assure a continuation of SIMULIA's growth and momentum and the incredible opportunity that V6 presents," said Bernard Charlès, president and CEO of Dassault Systèmes. "Mark has been instrumental in establishing SIMULIA as a major Dassault Systèmes brand and has laid a firm foundation for its continued success."

Berkey joined SIMULIA in July 2006 as Vice President of Worldwide Operations. He has played a key role in improving global operations and increasing global market share. Previously, he was CEO at both Proficiency and Axentis and he held senior executive positions at Structural Dynamics Research Corporation (SDRC).

We want to express our sincere thanks to Mark Goldstein for his excellent leadership of SIMULIA for the past seven years. Mark is stepping down as CEO for personal reasons. We are pleased that Mark will remain a Dassault Systèmes executive, reporting to Bernard Charlès with responsibility for selected strategic growth initiatives. He will also continue as a member of the SIMULIA board.

Keeping the Arctic Safe

Pipeline engineering firm J P Kenny is using new multiphysics technology from SIMULIA to ensure that oil pipelines buried beneath the sea floor in the arctic are able to withstand gouging (damage) from icebergs, thus helping to protect the arctic ecosystem from oil spills.

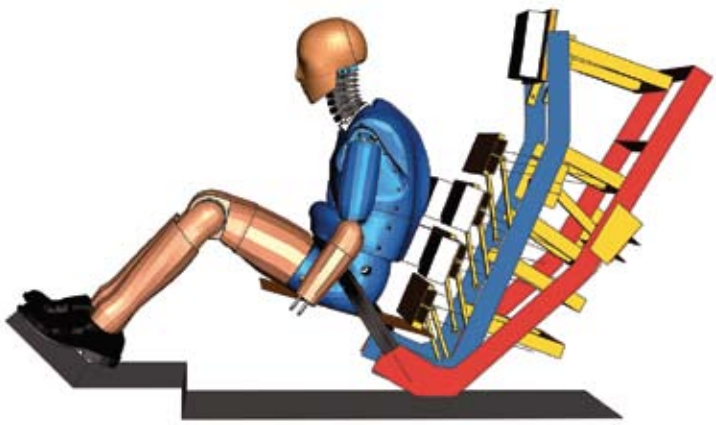


As part of J P Kenny's effort to provide expertise in arctic issues and solutions, the Advanced Engineering Group has recently implemented a new module based on the new Abaqus CEL technology that will be part of J P Kenny's SIMULATOR set of tools. The new module simulates the realistic response of buried pipelines under ice gouging conditions.

Comprising of a 3D finite element model of the pipeline, seabed soil, and iceberg, the module uses Abaqus CEL technology to analyze extreme deformations of the seabed undergoing iceberg gouging. The tool can help attain significant savings in pipeline burial in offshore arctic regions by realistically modeling sub-gouge soil deformation with no unnecessary conservatism. This could significantly reduce calculated required pipeline burial depth.

Abaqus BioRID II Crash Dummy Model

According to the Insurance Institute for Highway Safety (IIHS), whiplash injuries sustained in rear-end collisions account for approximately $8.5 billion in insurance claims annually. The new Abaqus BioRID II dummy model, in combination with the physical BioRID II crash test dummy, is used by automotive OEMs and suppliers to evaluate the realistic performance of complete seats (cushion and backrest), including head restraint systems that are designed to minimize whiplash injuries.



"Realistic simulation is proving to be ever more critical in the development of our seating systems to address market requirements and satisfy regulatory issues," states Laurent Guerin, Simulation Methods Manager, Automotive Seating Product Group, Faurecia. "The Abaqus BioRID II model is providing very accurate predictions and correlation against physical test results, and we are incorporating its usage as an integral part of our product development process."

The Abaqus BioRID II model has been developed and validated in cooperation with the German Association for Research in Automobile Technology (FAT). It takes advantage of unique Abaqus technology to represent the complex neck and spine construction of the physical BioRID II dummy. This state-of-the art technology enables accurate, yet computationally efficient, simulation of the accelerations and loads occurring in the spine, neck, and head during a rear-end collision.



Realistic Simulation Makes a Safe Impact on Train Design

Swiss-based Stadler Rail Group produces about 700 light and commuter rail vehicles per year. All of its products meet stringent requirements governing safety equipment, strength of train units (cars and engines), and, above all, passenger and crew protection from the force of impacts.

A recent order from the Netherlands for 43 of the latest generation of Stadler's GTW articulated rail cars presented the company with a new challenge: the train units had to meet as-yet unreleased crashworthiness standards that the country had adopted in advance of their approval by the European reviewing committee. These standards required that the units provide passenger zone protection during a 36 km/h (22.4 mph) front-end collision between two units with a vertical offset of up to 40 millimeters.

Two developments drove the new requirement. First, head-on impacts could easily include a small offset because two train units had differing amounts of wheel wear or braking inclination. A second reason was more urgent: a recent numerical simulation of an offset collision indicated that the previous design of a crash module (a safety device on the front of the train car) might not prevent damage to the passenger zone of the train units during such an impact.



Side view of two aluminum crash modules modeled in Abaqus FEA software to simulate a crash. The impact is offset to determine whether the teeth at the front of the modules can prevent either module from climbing over the other. Vertical stripes on the module sides represent trigger slots, points of weakness built in to induce controlled plastic failure.

"Numerical simulation suggested that the crash module could undergo global shear deformation and fail at the fixation point, falling off the front structure," says Dr. Alois Starlinger, head of structural analysis, testing, and certification at Stadler. "In such a shear mode failure, the module would not absorb any significant energy." In a worst-case scenario, both trains would climb over each other, deforming the passenger zones severely.

To satisfy the new safety requirement—which is scheduled to become the standard throughout Europe in 2008—Stadler Rail designed a new crash module with an anti-climb feature. Engineers validated the module design through a combination of dynamic physical testing and simulations in Abaqus finite element analysis software.

A "crash" design project

The crash module is a tapered rectangular tube that is 12 inches high and wide at the front, 30 inches long, and 14 inches high and wide at the rear, where it is welded to an end plate bolted onto the crash wall of the train unit. Partitions divide the module into chambers that provide stability to counter eccentric forces. On the front of the module are five horizontally aligned teeth, 70 mm apart with a depth of 40 mm, designed to engage the teeth of a similar module on an oncoming rail car and prevent climb.

Portions of this article have appeared previously in *Mechanical Engineering*.



Side view of the crash modules after physical testing of an offset crash. The modules performed successfully, absorbing the energy required to protect the passenger zone. The teeth kept the two modules engaged, and the aluminum deformed as desired, absorbing the impact of the crash with a controlled deceleration of less than 5 g. Note the close convergence of the Abaqus simulation and the physical testing.

Once the teeth have engaged, the rest of the crash module is optimized for controlled structural deformation from the front to the back. Targeted slots on the sides create intentional weak points that initiate buckling and subsequent energy absorption. In developing the design, engineers built on lessons learned while producing crash modules for previous generations of GTWs.

For the new design, the engineers selected an aluminum alloy, AW 5754. This alloy combines low yield strength with good plastic forming characteristics, enabling it to undergo large deformations without fracture. An important engineering goal was to create modules that could absorb up to 900 kilojoules of crash impact while decelerating the train unit at 5 g (g-forces) or less, as far as was practicable.

To capture the material behavior of the module, Stadler extracted information from its own materials database, compiled from exacting physical tests. Engineers incorporated the data into an Abaqus model, then calibrated the metals simulation by extracting aluminum samples from a series version crash module and testing the samples to create stress-strain curves. By comparing these curves to results generated by Abaqus simulations, the engineers were able to fine-tune the behavior of the FEA analysis so that it closely matched the real-world characteristics of the aluminum alloy in a crash module.

Now the engineers were ready to build a model of the crash module and analyze its behavior on impact. Simulation of the head-on offset impact followed a number of parameters:

- Collision masses (train units): 100,000 kg each
- Closing speed (combined speed): 36 km/h
- Maximum energy to be absorbed by crash components of both train units: 2,230 kj
- Maximum energy to be absorbed by a single train unit: 1,115 kj

Because of the complexity of the analysis, engineers chose to run nonlinear dynamic simulations with Abaqus/Explicit so they could observe the elastic-plastic behavior of the metal, measure progressive damage and failure of welding, analyze the large deformations of the module, and model contact and friction. “Abaqus was able to capture all the forces and materials behavior we needed,” Starlinger says. “General contact capabilities of the software were particularly useful.”

The finite element model and the analysis task before it were both dauntingly large. There were 450,000 elements in the model, and the dynamic simulations captured a period of 0.4 seconds broken down into 200,000 “snapshots.” To promote a speedy run time, the engineers ran the software on an SGI Altix 350 with 4 Itanium processors with activated parallel processing.

Train units were modeled in 3-D with running bogies (wheel, axle, and frame assemblies) and suspension characteristics to capture any lift-off of the wheels and axles on impact. Contact conditions were defined between the wheels and rails, as well as between the bogies and train unit body. Forces applied on impact by attached articulated units were modeled axially with 1-D and mass elements.

Safe, speedy arrival at results

Abaqus simulation results correlated very well with physical dynamic tests. The anti-climb teeth prevented either train unit from moving over the other, and the module body underwent controlled deformation to absorb 1.1 megajoules. Aluminum buckling decelerated the train unit at an average of 1.25 g.

“Our goal was to achieve an overall compressive strength for the train unit to 1,500 kilonewtons, without undergoing any yield and deformation in the passenger structure,” Starlinger says. “In fact, our crashworthiness engineering improved the compressive strength to about 3,600 kn, with only small amounts of plastic deformation in the passenger zone.” He adds, “And we proved out the anti-climb device against offsets as high as 80 mm.”

In addition to the accuracy of the Abaqus simulations, their fast run time (18-46 hours) was important. “We were able to release the crash module for production exactly eight months after the contract was signed,” Starlinger says. “The whole GTW Arriva went into operation ten months later, which is probably a record for starting a design from scratch in passenger train service.”

Stadler plans to build on its experience and continue making each new train design safer than the last. Starlinger sees Abaqus software as an important part of that process. “In its own way,” Starlinger concludes, “FEA is now as essential to ensuring train safety as brakes.”

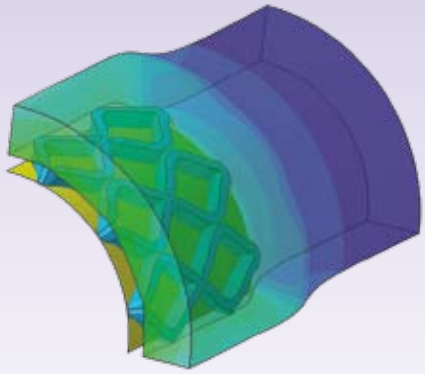
For More Information
www.stadlerrail.com

Gain Simulation Fidelity and Efficiency with New Release of Abaqus Unified FEA

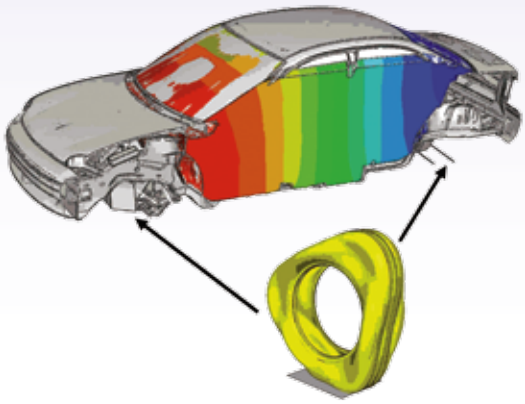
Abaqus 6.8 Delivers Enhanced Modeling and Visualization, Structural Analysis, Computing Performance, and Multiphysics Capabilities

The release of Abaqus 6.8 marks a 30-year milestone in the evolution of realistic simulation. First released in 1978, Abaqus software has undergone significant development over the years, resulting in the most robust and technically capable FEA software on the market. The latest release of Abaqus FEA continues this tradition of technology advancement by providing our customers with a broad set of realistic simulation solutions that have industry-specific application within an open and unified modeling and simulation environment.

"Abaqus FEA software provides the robust contact capabilities that we need to study complex loading and structural interactions while designing our large, two-stroke diesel engines," stated Per Rønnedal, Senior Manager, R&D New Design, MAN Diesel. "With the new capabilities in Abaqus 6.8, we have seen memory usage significantly reduced when running linear static analyses with multiple load cases. These improvements will allow us to test a number of design alternatives to optimize our diesel engine performance during early product development."



The new anisotropic hyperelastic material model in Abaqus 6.8 enables the complex modeling and analysis of biological tissues. This capability enables medical device manufacturers to improve the performance and reliability of medical devices such as stents.



Industry-unique capabilities in Abaqus 6.8 allow automotive engineers to capture full-vehicle noise and vibration response due to tire rolling effects and viscoelastic material effects from tires, bushings, isolators, and laminated steel.

New & Enhanced Features

With a focus on solving specific engineering challenges in industries such as automotive, aerospace, electronics, energy, and packaged goods, Abaqus 6.8 provides hundreds of new features and customer-requested enhancements in core areas, including modeling and results visualization, structural analysis, composites failure, general contact, computing performance, and multiphysics.

To effectively simulate composite crack propagation, delamination, and possible failure, aerospace engineers can leverage the fully built-in and improved Virtual Crack Closure Technique (VCCT). Previously available as an Abaqus add-on product, VCCT is now native in Abaqus/Standard.

Automotive engineers can use Abaqus 6.8's industry-unique linear dynamics capabilities to capture full-vehicle noise and vibration response due to tire rolling effects and viscoelastic material effects from tires, bushings, isolators, and laminated steel.

Electrical engineers are able to use the direct cyclic procedure to predict fatigue life in solder joints that are subjected to repeated thermal cycling. This method is also useful to automotive engineers evaluating powertrain durability.

Medical device developers can benefit from the new anisotropic hyperelastic material model to predict more realistic vascular tissue interaction with devices such as stents. This model can also be used to analyze materials such as reinforced rubber and wood.

The new Coupled Eulerian-Lagrangian (CEL) multiphysics capability in Abaqus 6.8 enables engineers to model materials that undergo extreme deformation. This capability can be used to predict the behavior of fluid-filled containers, hydroplaning tires, bird-strike on aircraft, and loads on earth-moving equipment during soil excavation.

Sensors and actuators are also now available to enable the modeling of feedback control systems used in applications such as robotics, suspension systems, and process modeling.

For More Information
simulia.com/products/abaqus_fea

Additional Features in Abaqus 6.8

Modeling and Visualization

- A new SolidWorks Associative Interface maintains the relationship between SolidWorks and Abaqus models.
- Enhancements to the Pro/ENGINEER Associative Interface provide a bi-directional capability to synchronize model changes between Abaqus and Pro/ENGINEER.
- The Material Library enables the efficient use and management of material properties from multiple sources, including third-party and proprietary databases.
- Modeling improvements accelerate the creation and definition of meshes, fasteners, connectors, cyclic symmetry, and cavity radiation.
- Visualization and output improvements have been made for free-body cuts, beam and truss elements, and contact analyses.

Computing Performance

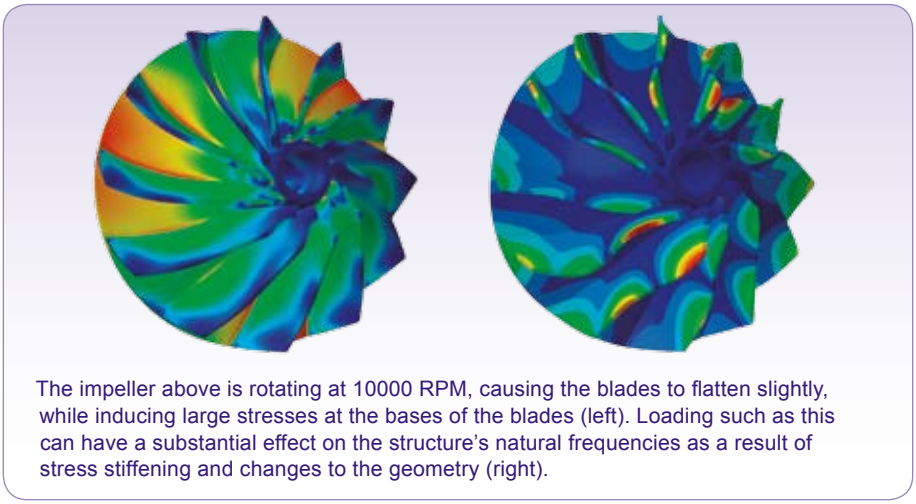
- High-performance computing is enhanced through improvements to the parallel direct sparse solver, which reduce memory usage and accelerate solving large models.
- The general contact algorithm provides enhanced performance for the evaluation of edge-to-edge contact and solid-element surface erosion.
- Additional performance improvements have been made to the Lanczos eigensolver, cavity radiation, constraint equation handling, memory usage, and disk management.
- Improved support of subspace-based steady-state dynamics and multiple load case analysis significantly reduces run time for large models with a large number of modes.

Abaqus for CATIA V5 R18 SP4

Latest Release Extends Realistic Simulation Capabilities in CATIA V5

The newest release of Abaqus for CATIA V5 continues SIMULIA's strategy of delivering scalable analysis solutions that allow realistic simulation to be used throughout the product lifecycle. Abaqus for CATIA V5 R18 SP4 provides improved usability and robust design analysis capabilities directly in CATIA V5 to accelerate the product development process.

Using this latest release of Abaqus for CATIA V5, expert analysts can define and deploy approved analysis workflows for use across the enterprise. This capability enables design and engineering teams to improve collaboration while evaluating design performance through the use of common FEA models, technology, and methods that are synchronized with their CATIA V5 design.



The impeller above is rotating at 10000 RPM, causing the blades to flatten slightly, while inducing large stresses at the bases of the blades (left). Loading such as this can have a substantial effect on the structure's natural frequencies as a result of stress stiffening and changes to the geometry (right).

New and Enhanced Capabilities

The Abaqus for CATIA V5 R18 SP4 release includes several new and enhanced capabilities, including:

- Natural frequency analysis on the unloaded structure or after any level of loading.
- Enhanced precision in setting up models involving contact—nodal adjustments made by the Abaqus solver can be previewed in the user interface.
- Improved CATIA Octree meshing to provide a higher level of element quality when working with Abaqus models.
- Element quality checks implemented in CATIA specifically for Abaqus users.

"Strength testing of automobile door latches is fundamental for us, but physical testing is slow and does not realize the whole job," says Thomas Waldmann, manager of the Technical Analysis and Simulation department at Kiekert, a global supplier of automotive door latches. Abaqus for CATIA V5 R18 SP4 represents SIMULIA's dedication to delivering a scalable suite of design analys is products that permit customers to start with basic design analysis and add functionality, such as nonlinear contact, thermal analysis, or coupled multiphysics as needed.

"We began our linear analysis of door latches with finite element calculations inside CATIA. By adding Abaqus for CATIA V5, we are able to perform more complex nonlinear contact simulations, which is what we need for accurate strength testing in three dimensions."

For More Information
simulia.com/products/afc_v5

Iterative Design Process with Abaqus for CATIA V5

During product development, design engineers often have the freedom to modify size, shape, orientation, and material properties of their product. However, each modification requires validation to ensure that the maximum stresses remain below the acceptable limit for all load cases. This article outlines two analysis approaches with Abaqus for CATIA V5 to evaluate a product design and achieve significant productivity gains.

The first approach uses CATIA's Knowledgeware capability to define parametric studies to assist in making engineering decisions. With this approach, geometry and load parameters can be captured in a design table to simplify model modifications.

The second approach makes proven analysis methods available in analysis templates that can be reused to analyze design variations efficiently with minimal effort. This approach facilitates the replacement of an entire geometric component in the analysis with another design from a different vehicle (Figure 1).

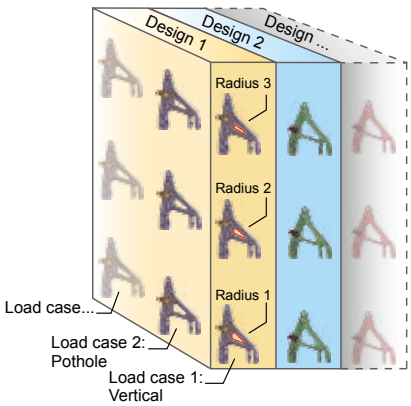


Figure 1: Using design tables and analysis templates to analyze multiple designs under various load cases.

Suspension Control Arm Assembly

For this example a suspension control arm is modeled with four geometric parts: the control arm, the shock absorber bracket, and two bolts (Figure 2). The control arm connects the steering knuckle to the chassis and allows the knuckle to pivot on a lower ball joint. The shock absorber, which controls vertical movement, connects to the control arm through the bracket.

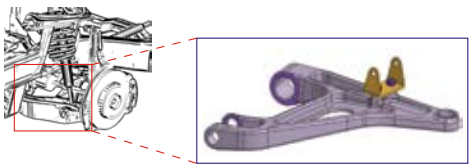


Figure 2: Suspension system assembly and control arm.

Rubber bushings are used for both the front and rear attachments to the chassis. The rear bushing is modeled using a linear spring in the axial direction and a nonlinear spring in the radial direction to simulate the bushing's carefully-tuned behavior.

The two bolts connect the bracket to the control arm. The stress in the control arm at the location of this connection is particularly important. Therefore, the effects of pre-tensioning the bolts using bolt-tightening connections must be modeled accurately. Defining contact properties between two surfaces helps ensure accurate results. In this analysis, the contact between the bracket and the control arm is modeled without considering friction effects.

Load Cases & Geometric Parameters

In a traditional design analysis process, the suspension model would be subjected to a variety of load cases, such as braking, accelerating, and driving over a pothole. To simplify this example, only pothole and vertical load cases are considered. These load cases consist of forces and moments applied at the chassis and at the lower ball joint's attachment points (Figure 3). In total, 15 load case parameters are implemented in the analysis: 9 force components and 6 moment components.

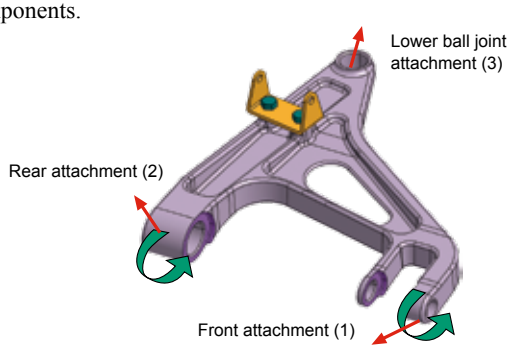


Figure 3: Load and moment locations.

The size of the control arm pocket can be a factor in weight and cost. Therefore, it is important to evaluate the effect of pocket dimension variations on the stress results. In this example three pocket radii are evaluated: 7 mm, 10 mm, and 12.5 mm.

Approach 1: Controlling parameters through a design table

A design table provides the designer with a tool to create and manage sets of model parameters. Design tables drive the parameters of a CATIA-based model from external values. These values are stored in a table in either a Microsoft Excel file or a tabulated text file. Each column in the table corresponds to a model parameter, and each row contains a set of parameters to be considered together as a group. These sets are called configurations (Figure 4).

DesignTable , configuration row : 1

Filter :

Line	Config.	Arm\Pocket_Radius	FB_Fx	FB_Fy	FB_Fz	FB_Mx
<1>	VerticalLoad+R1	12.5mm	50N	5600N	-3200N	24.3Nxm
2	PotHole+R1	12.5mm	30N	8400N	-2500N	24Nxm
3	VerticalLoad+R2	10mm	50N	5600N	-3200N	24.3Nxm
4	PotHole+R2	10mm	30N	8400N	-2500N	24Nxm
5	VerticalLoad+R3	7mm	50N	5600N	-3200N	24.3Nxm
6	PotHole+R3	7mm	30N	8400N	-2500N	24Nxm

Figure 4: Design table.

In this study six configurations are defined: each pocket radius value is tested against each of the two load cases. Each time a new configuration is selected, the model is updated automatically. The design engineer steps through the design table to obtain the results needed for each configuration (Figure 5). Alternatively, a Visual Basic script can be used to cycle through the various design configurations and run the analyses.

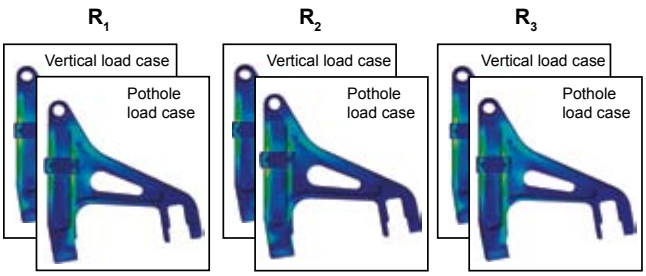


Figure 5: Von Mises stress result images.

Approach 2: Reusing Analysis Models with Analysis Templates

The second approach reuses the analysis model with geometry from another vehicle (Figure 6). The same stress evaluations can be conducted on different vehicle programs, allowing for great gains in efficiency with little effort.

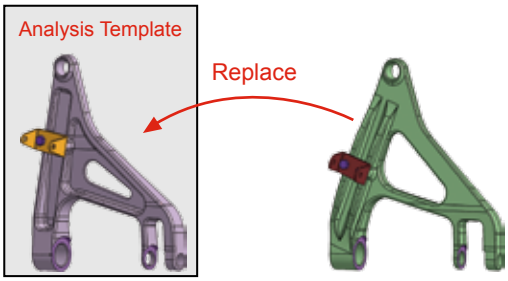


Figure 6: Different geometry evaluated using the same analysis template.

The CATIA publication capability is used to create an analysis template. Using this approach, a geometric entity (face, edge, point, etc.) is associated with a unique and descriptive name called a publication. For example, the lower ball joint attachment surface is associated with the "LowerBallJoint" publication. Bolt tightening connections, springs, contact definitions, and loads are created based on publications.

Figure 7 and Figure 8 illustrate the difference between the standard (non-publications) approach and the analysis template approach (with publications). Using the standard approach, analysis attributes are associated directly with the geometry. On the other hand, using the analysis template approach, all analysis attributes are applied to publications rather than to the geometry. The ability to abstract the analysis attributes from the geometry is a fundamental requirement for creating analysis templates.

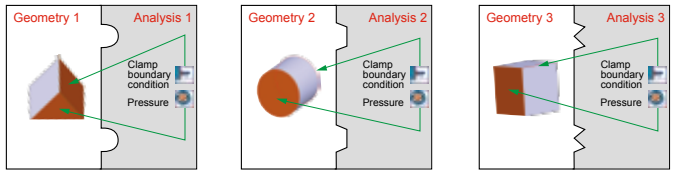


Figure 7: Standard approach. Analysis attributes are applied to the geometry.

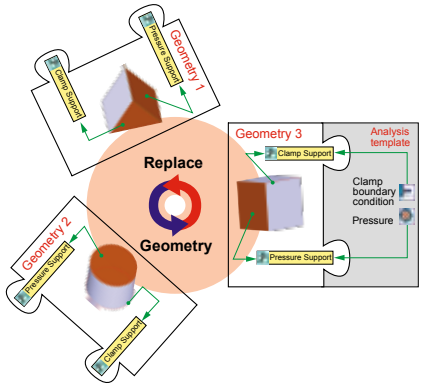


Figure 8: Analysis template approach. Publication objects are generated from geometric entities, and analysis attributes are applied to the publications.

Applying analysis attributes to publications instead of to the geometry enables a design engineer to modify or replace the geometry while maintaining full associativity between analysis attributes and geometry. Various designs can be tested using the same analysis model, provided that each design uses the same publication names.

Once an analysis is defined, it can be used as a template for other design evaluations. In this case the control arm and the bracket are replaced by a new design, while the analysis model and the design table remain the same. All analysis attributes update automatically to associate with the new geometry. The design engineer only needs to re-run each analysis for the new suspension system design. Results can be compared quickly to determine whether the design configurations meet the appropriate stress requirements.

Conclusion

Abaqus for CATIA V5 provides the capabilities to easily perform analyses and obtain accurate results, including nonlinear effects during the design process. The application also enables design engineers to iterate efficiently on an analysis, using tools such as design tables and publications, resulting in important time savings. If an alternate design must be considered, an analysis template with an updated design can be used to obtain the necessary validation.

To Download the Complete Tech Brief, Visit:
simulia.com/techbriefs

Comfort Evaluation of Foam Seats
Using Realistic Simulation

Finite Element Analysis (FEA) is accepted across a wide range of industries as an invaluable tool for product design and optimization. Design engineers from IDOM and Centro Tecnologico Grupo Copo are using Abaqus FEA from SIMULIA to evaluate the "feel," or comfort, of car seat foam—basically performing numerical modeling of tangible human criteria.

Cushions, backrests, headrests, armrests, and other foam parts that make up a vehicle seat are designed according to four principal criteria: integration within the vehicle, safety, aesthetics, and comfort. Satisfying all of these requirements, while working with complex foam material, makes the process of creating the seats complicated and time-consuming. In addition, measuring and evaluating results—particularly in the subjective area of comfort—presents a particular challenge to designers.

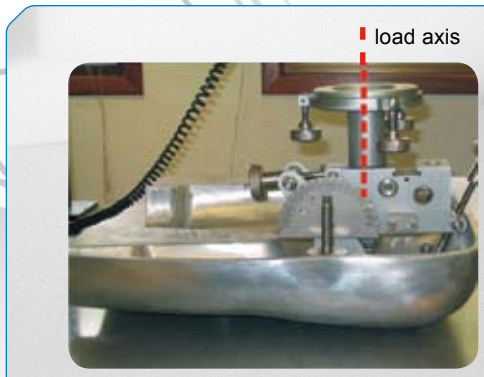
When designing car seats, most of the variables to be considered relate to either geometry or materials. Since the mechanical properties of foam are highly dependent on their strain level, multiple test specimens must be built to evaluate response and satisfy design requirements. Preparing a specimen for testing can take weeks: first, the mold with the desired geometry must be built, then the specimen is created in this mold, and finally, the specimen must be readied for the testing criteria being examined.

A valuable tool for facilitating and shortening this complex design process is numerical simulation using finite element analysis (FEA). Modeling the seats in a virtual environment integrates CAD with material databases and allows the input and evaluation of a variety of loads and stresses without the time constraints of reality testing. FEA can predict the response of a particular design under specific circumstances and supply data that can be used to optimize geometry and materials. But how can the simulation results be used to evaluate comfort?

One of the difficulties inherent to comfort assessment is to translate the sensation of comfort into quantifiable variables in order to measure comfort from the mechanical point of view.

Measuring seat comfort in vehicles can be carried out under both static and dynamic conditions. Typical variables for comfort assessment are related to occupant's position, such as hip-point or Seat Reference Point (SRP), to pressure distribution, or to the response of a seat specimen with an occupant model on it under the effect of a certain vibration spectrum. Variables related to comfort assessment can be obtained from CAD-geometry of the assembly vehicle-seat-occupant or from the mechanical response of the seat to certain tests. Since all these tests can be simulated numerically with FEA, comfort can be assessed in a design analysis environment.

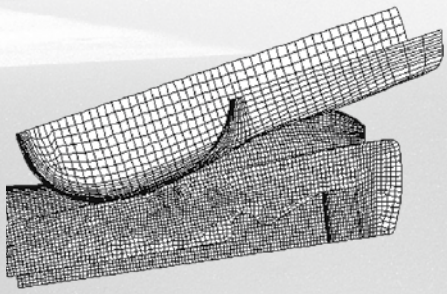
At IDOM, we conducted a preliminary feasibility study in the framework of a more ambitious program aimed at developing a virtual environment for foam seat testing. This study focused on the assessment of numerical simulations, primarily related to comfort, using Abaqus FEA software from SIMULIA.



Test form

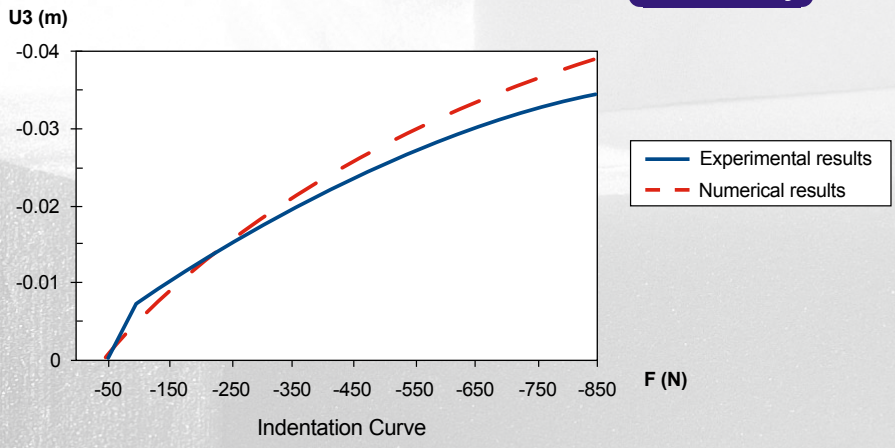


Seat cushion



Finite element model

Figure 1: The top and middle images show the test form used for the indentation test and the geometry of a seat cushion. The bottom image shows the finite element discretization of both parts (test form on top of seat cushion) used to simulate the indentation test. The test form was discretized with 1642 4-node rigid elements (R3D4). The reference node of the surface of the test form lies on the connection between the test form and the load axis. At this node, all degrees of freedom, except vertical displacement, are restricted.The seat cushion was discretized with 8-node brick elements (C3D8). To reduce computational time, only half of the seat cushion was considered in the finite element model, taking advantage of symmetry with respect to a vertical plane. The mesh has 125,164 elements and 138,172 nodes.



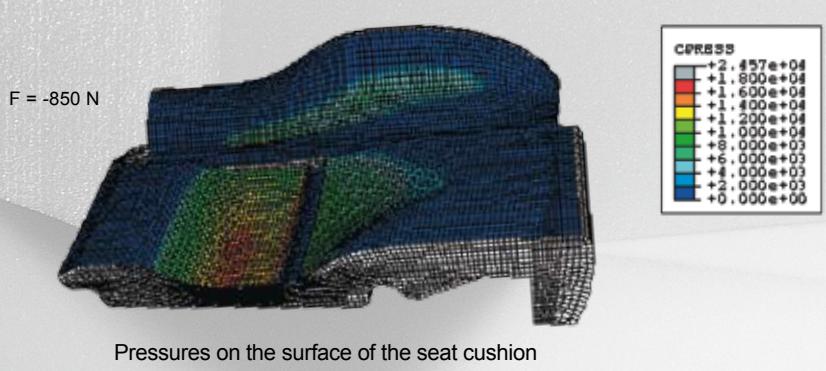


Figure 2: The top graphic shows the comparison of the indentation curve (vertical displacement U3 of the test form versus applied load F) obtained experimentally and numerically. The bottom image shows the contact pressure (Pa) on the upper surface of the seat cushion for an applied load of 850 N on the test form. Using an HP workstation with a 2.8 GHz IntelXeon processor and 1.5 Gb of RAM memory, the computational time required for the simulation of the indentation test with Abaqus/Explicit was 2 hours.

As part of our study, a physical test for static indentation was simulated numerically. The aim of this test is to simulate the mechanical response of a seat cushion when an occupant sits on it (Figure 1). Lab testing was carried out on a seat cushion positioned on a rigid support. A test form that reproduces the occupant's thighs was placed over the seat cushion. A vertical load was applied gradually on the test form, moving downward on the seat cushion to simulate the action of an occupant sitting on it. The penetration of the test form on the cushion was measured for certain loads applied to the test form.

In the general methodology of numerical simulation of the static indentation test on foam seats, two steps can be clearly differentiated: fitting of the general constitutive material model for elastomeric foams to the particular case considered, and simulation of the static indentation test itself.

Flexible polyurethane foams are hyperelastic materials. They have certain properties that make them very suitable to serve as seat cushions, because they significantly contribute to comfort. These foams can elastically undergo large strains, up to 90% in compression, and they have excellent energy absorption properties. They are highly nonlinear, they have viscoelastic properties, and they suffer from material softening in the first load cycles (Mullins effect).

The Ogden material model for highly compressible hyperelastic materials is contained in the Abaqus code. The three main hypotheses of this model are isotropy at a macroscopic scale, hyperelasticity, and non-hysteresis. The values of material parameters can be determined by means of a least squares fitting from stress-strain measures of simple experimental tests.

The material database used for this study contains a wide range of polyurethane foams, the specification of the polymer required to manufacture the foam, and different material and mechanical properties. The design process is significantly facilitated by means of this database, because of the capability of evaluating the performance of the same geometric design with different materials.

In our study, once the generic constitutive material model for elastomeric foams was particularized with experimental data of simple tests, more demanding tests, such as the static indentation test, were simulated. The curve-applied load—vertical displacement of the test form obtained in the simulation of the indentation test—showed good agreement with the one obtained in the experimental test (Figure 2).

Additional information relevant for comfort assessment can be easily obtained from such a simulation. Examples are contact pressure distribution on the upper surface of the seat cushion, vertical stress distribution in the seat cushion or contact area, and distribution of the load between seating plane and side wings.

As a consequence of the preliminary study, expectations about the potential of the application of numerical simulations with FEA in comfort assessment were met: experimental tests aimed to obtain results useful to compute comfort parameters can be successfully simulated numerically. Moreover, a great amount of information useful for comfort assessment can be easily obtained from these simulations.

The virtual testing approach also significantly reduces the time required for each design loop, allowing the design process to take advantage of the analysis to optimize the geometry in a virtual environment. FEA is definitely a valuable tool for assessing seat designs for comfort, and it will play a major role in this field in the future.

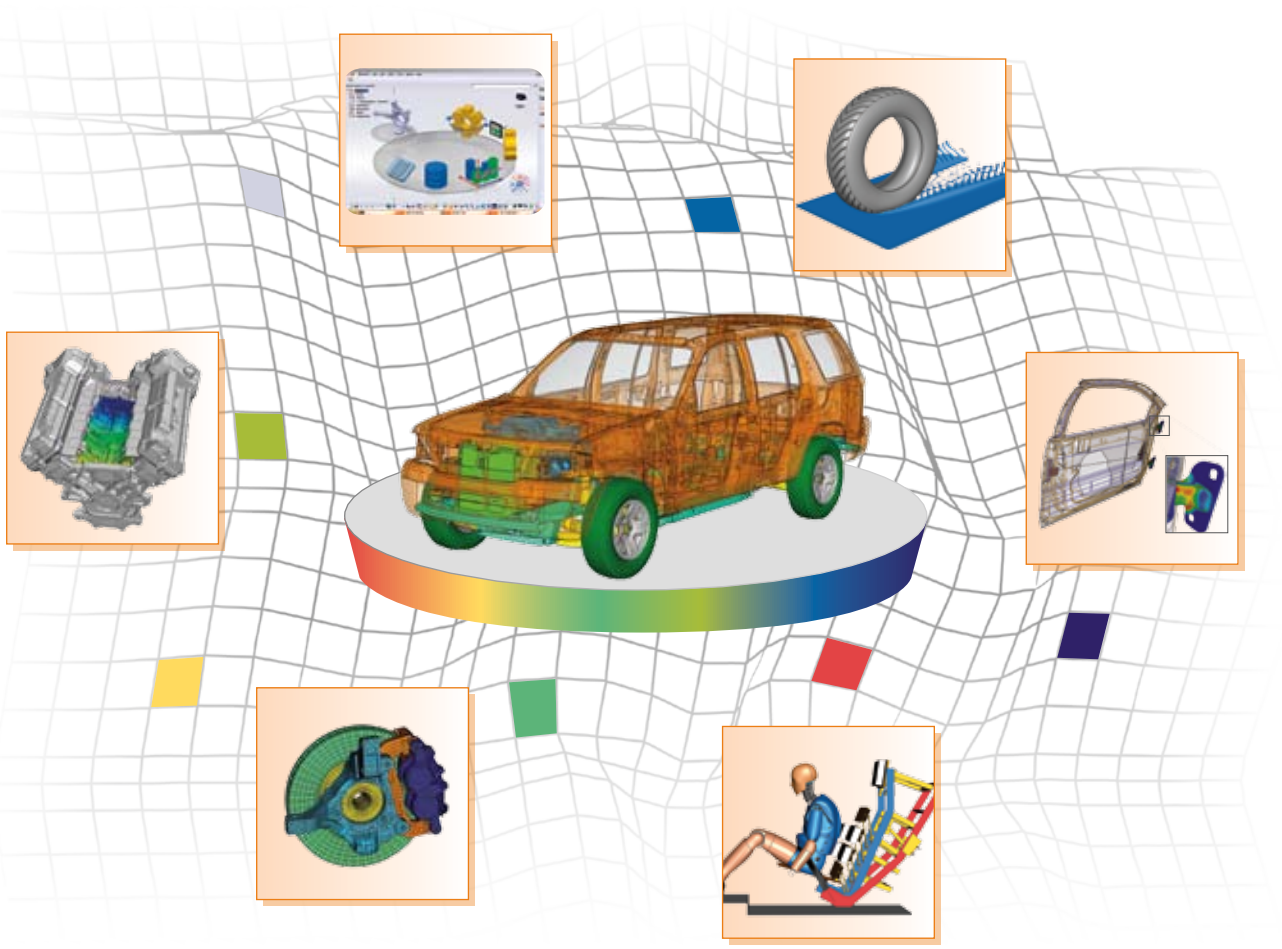
Natalia Camprubí and Fernando Rueda,
Advanced Design & Analysis Division, IDOM

Iván Alonso, Centro Tecnológico Grupo Copo

For More Information
www.idom.es/ada
www.grupocopo.com

Accelerating Automotive Innovation

Brad Heers, Automotive Lead, SIMULIA Technical Marketing



Automotive industry challenges of lower costs and improved reliability have created a need for increasingly accurate, complex, and realistic simulations. Over the years, this demand has been a key driver for enhancements in the Abaqus FEA software.

In early versions of Abaqus FEA, the nonlinear and contact capabilities in the software enabled engineers at Automotive OEMs and suppliers to solve problems in applications such as rubber seals and component durability. Today, SIMULIA is expanding beyond this initial foundation to develop a comprehensive unified FEA solution for the automotive industry.

From tires to powertrain, from chassis to crashworthiness, and from interiors to components, SIMULIA's automotive solutions have become an integral part of the industry's product development process. Many of the new features and enhancements in the new release of Abaqus Unified FEA are focused on performance, scalability, and new linear dynamic capabilities, which directly support our automotive customers (see Abaqus 6.8 Product Update, p. 8).

Building on Nonlinear Analysis Strengths

During the assessment of Powertrain performance, automakers realized that engine assembly, sealing, and durability simulations required advanced nonlinear numerical technology. Traditional linear methods were not providing the accuracy they needed to avoid leakage and costly assembly problems. Abaqus—with its market-leading nonlinear technology—quickly became the dominant platform for Powertrain simulation. Our Powertrain simulation roadmap is focused on delivering more valuable simulation tools to the Powertrain engineer in the future. At many automotive manufacturers around the world, Abaqus is used to improve gasket sealing, extend piston life, and predict the location of fatigue crack initiation in cylinder block threads.

Tire simulation is another application area for early success of SIMULIA in the automotive market. SIMULIA now leads the tire simulation industry, and we continue to grow in this area. The evaluation of the complex combination of rubber, steel, and

other layered materials makes testing even a few physical prototypes expensive and time-consuming. Abaqus Unified FEA is used to reduce the time and cost associated with evaluating tire durability, tread wear, rolling resistance, and other realistic behaviors. Optimizing tire performance plays a critical role in improving vehicle safety, reliability, passenger comfort, and fuel economy.

Exploring New System-Level Capabilities

With new and improved Abaqus technology, vehicle-level simulation capabilities have begun to emerge, including Noise & Vibration (N&V) and System-level Integrity (SLI) for moderate-to-severe event strength and durability. In Abaqus 6.8, the addition of multiple load cases in mode-based steady-state dynamic analyses has significantly improved performance in Abaqus: in certain benchmark models, analysis times have been reduced from 12 days to 4 hours—which means that Abaqus is now a competitive alternative to analysis tools traditionally used for these processes.

Other new developments that provide benefits to N&V engineers are a range of performance improvements to the Lanczos and AMS eigensolvers. The redesign of substructures allows unsymmetric stiffness and damping matrices, which introduces a unique capability to the market. Using this capability, tire manufacturers can now transfer the complex results of their tire analyses to vehicle OEMs in the form of a substructure—which conceals proprietary tire design information, while allowing vehicle OEMs to account for tire effects in their N&V analyses.

In addition, the inclusion of sophisticated frequency-dependent response within mode-based procedures allows customers to more accurately model such features as bushings and laminated steel in the frequency domain. This introduces material and component responses that are not readily captured with historical N&V finite element solvers.

In the area of SLI, Abaqus is also gaining traction with automakers who want to use Abaqus not only for traditional linear dynamics and mode-based simulations, but also for more advanced nonlinear workflows in which the detailed effects of assembly, nonlinear bushings, and tires are more accurately incorporated into suspension-level and vehicle-level durability models.

SIMULIA has put significant R&D efforts into enhancing general contact, mesh-independent fasteners, connector elements, high-performance computing, and other specific features to develop competitive simulation solutions for crashworthiness and occupant safety. We are working with First Technology Safety Systems (FTSS) to develop validated Abaqus crash dummies, and we recently announced the new BioRID II crash dummy for Abaqus that was developed in cooperation with the cooperation with the German Association for Research in Automobile Technology (FAT).

Leveraging Unified FEA

Unified FEA is the use of a single unifying data model that can be used to drive the creation of simulation models for all three domains. Today, product development organizations are often comprised of three separate engineering groups creating three separate vehicle models—one for N&V, one for SLI, and one for crashworthiness. Often, the three groups use different simulation tools. This can lead to duplication of effort, which is costly and often out-of-sync with today's demands for faster design approvals.

Unified FEA allows our customers to combine the traditional Abaqus strengths in powertrain, tire, and component durability with the capabilities in system-level N&V, SLI, and crashworthiness. Unified FEA is able to lower the cost of simulation by providing a single simulation platform. It can also contribute to improved simulation accuracy due to the use of common technology across multiple domains. Ultimately, it enables our customers to gain significant benefits earlier in a vehicle's design cycle.

Simulation Lifecycle Management Solutions

Over the years, the volume of simulations being performed has grown dramatically. This has led to an exponential increase in new methods development, as well as resulting simulation data. As a brand of Dassault Systèmes (DS), SIMULIA is leveraging existing DS technology available in ENOVIA to provide a solution for Simulation Lifecycle Management (SLM). SIMULIA SLM maximizes the value of company-generated IP through the capture, re-use, and deployment of simulation best practices for collaborative product development. The new release of SIMULIA SLM delivers unique capabilities to integrate and control the execution of simulation applications, carry out operations such as query and version control, administer access privileges, and perform and review simulations in a distributed, collaborative environment that provides significant value to our automotive customers.

With the significant growth in our realistic simulation technology, as well as in our automotive-related customer base, it is clear that our strategy of providing robust nonlinear FEA for the entire vehicle development process—plus the tools to manage and secure the resulting IP—is resonating strongly within the industry. It is our goal to meet with you regularly to understand your requirements and deliver solutions that you need today and tomorrow to meet the automotive industry's challenges of lower costs, faster development cycles, and improved reliability and performance.

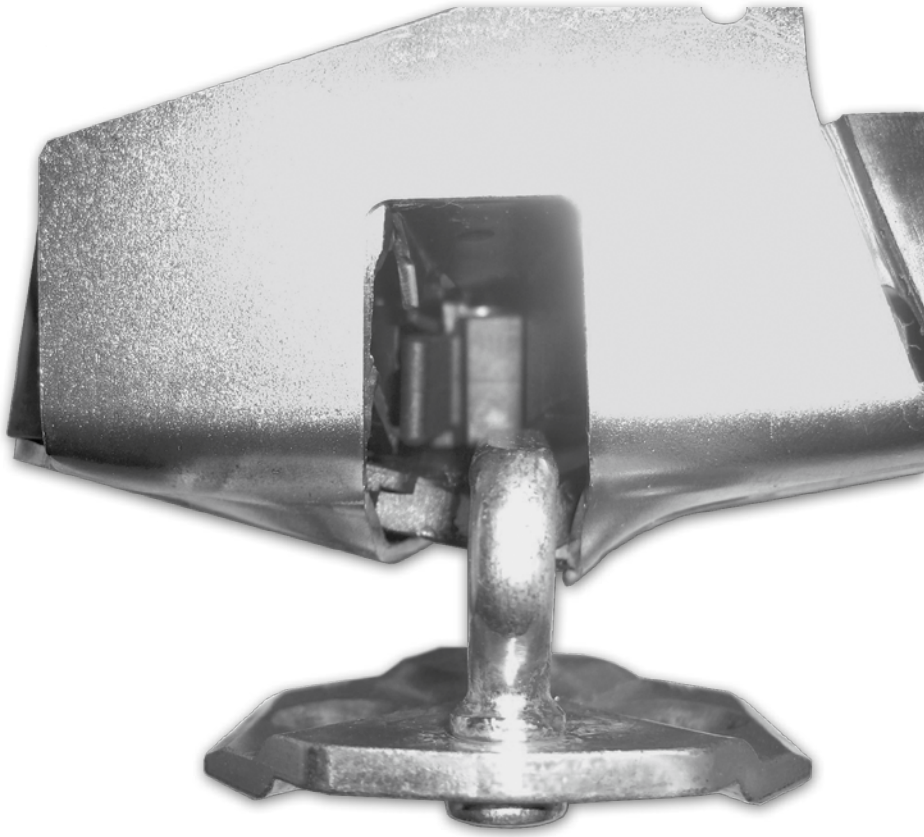


Brad Heers – *Automotive Lead*, SIMULIA

Brad joined SIMULIA in 2000. He is familiar with CAE automotive techniques across a variety of subsystems and disciplines, and works globally with major OEMs and suppliers. Prior to joining SIMULIA, Brad worked for General Motors Corporation for six years in the CAE Chassis Analysis and Vehicle Dynamics group. Brad received his B.S.M.E. from GMI Engineering & Management Institute (now Kettering University) and his M.S.M.E. from Stanford University.

For More Information
simulia.com/solutions/automotive

Kiekert Locks onto Car Door Safety with Realistic Simulation



Leading latch maker establishes virtual product validation process with help of Abaqus Finite Element Analysis

Keeping car doors securely fastened during automobile accidents is an important aspect of occupant safety. While airbags and seatbelts receive significant public attention, developing safe door latches is also a high priority for automotive manufacturers. Meeting pressing government safety standards and satisfying different OEM design specifications in a cost-effective, timely manner is a major challenge for automotive door latch suppliers.

German latch maker Kiekert, the world's most experienced manufacturer of automobile locking systems since 1857, produces 35 million door latches per year. Kiekert's customers, which include Daimler AG, Chrysler, VW-Group, BMW, PSA (Peugeot & Citroen), Renault, Ford-Group, and GM, also depend on their innovative technology for side-door latches, power sliding doors, remote operations, and liftgate-and trunk-lid actuators.

The critical components are those you can't see

The visible handle and lock cylinder of a car side door are connected to a complex inner system of cables, cams, levers, couplings, actuators, gears, pawls, and catches that is collectively known as a door latch system. When activated, the catch and pawl secure the door by clamping around the striker, a u-shaped piece of metal bolted to the vehicle's B-pillar (where the seatbelt is also fixed).

The side door latch enables a multitude of door functions, such as opening and closing from inside or out, locking and unlocking from inside or out, central-, child-proof-, remote-locking, and more. The entire system must continue to function for years at a wide range of temperatures, within specific noise and vibration limits, and also be strong enough to survive a crash. The integrity of the latch/striker connection is critical to that strength.

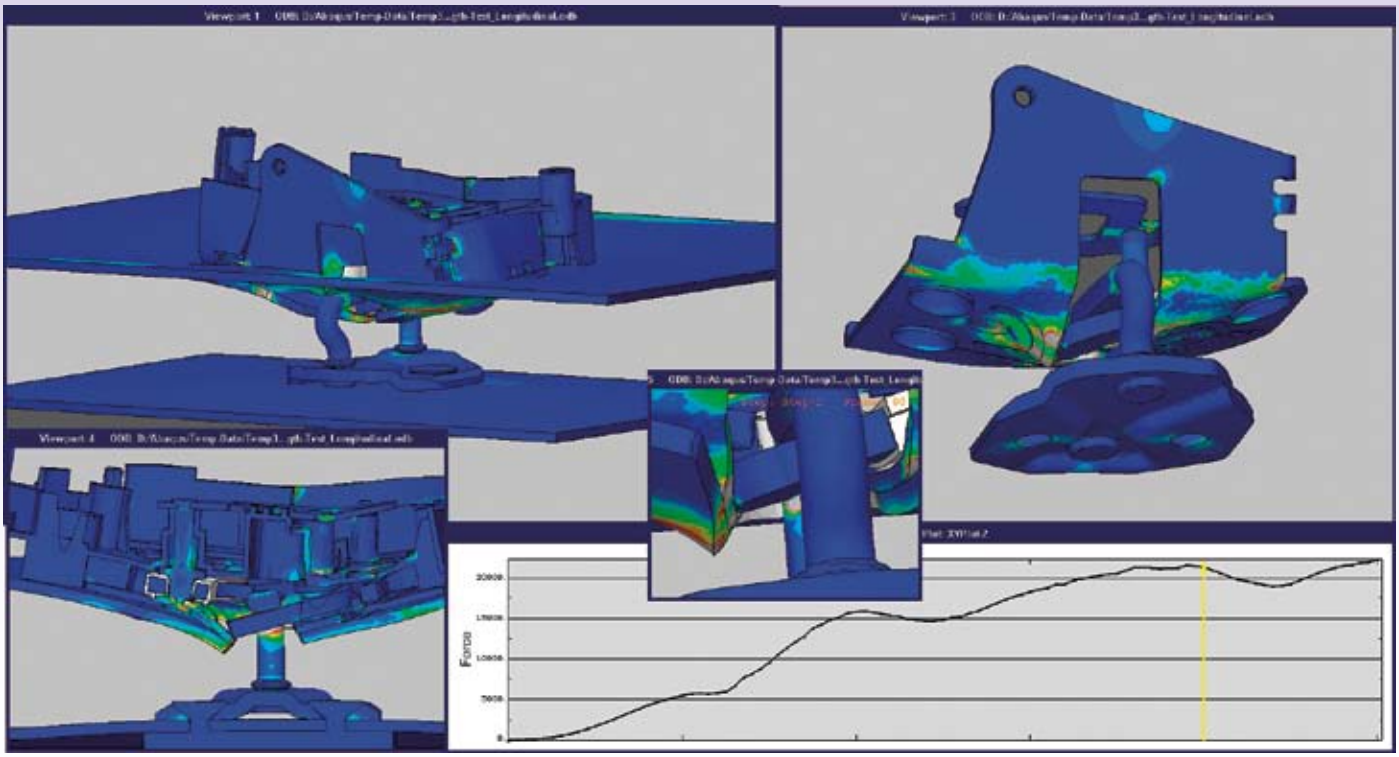
Governments and OEMs have different standards

"Strength testing of door latches is fundamental for us," says Thomas Waldmann, manager of the Technical Analysis and Simulation department at Kiekert. "Not only do we have to meet standards set by governments, but each car manufacturer has its own testing requirements as well."

Waldmann heads a global group that focuses on computer modeling of door-latch components with the primary emphasis on full virtual validation. The validation program (DSP&R, Design Simulation Planning and Report) was developed over many years and includes over 80 different simulations and calculations. "Kiekert's engineering tools provide a high-quality level of results," says Waldmann. "Our unique body of knowledge and our testing database ensure low project risks and the most efficient design process in the latch industry." Meeting global standards for strength testing (currently about 1.1 metric tons, or 11,000 Newtons, of force applied to the latch/striker connection from multiple directions) is relatively easy, he notes: the real work is passing the regulations from Kiekert's OEM customers. "Most customers' standards are about twice their government figures."

Quality, quickly

Kiekert begins with CAD models and runs them through a three-part process that includes finite element modeling, multibody dynamic simulations, and tolerancing—all of which must be done accurately in the



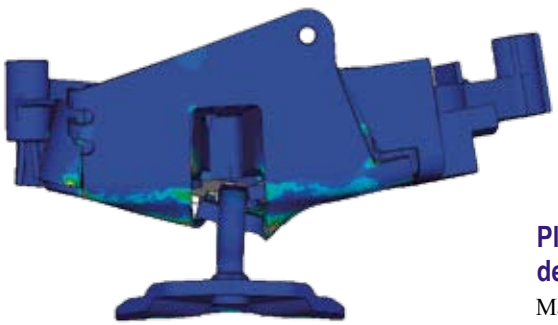
These images show different views of Abaqus finite element analysis of a Kiekert strength test in a longitudinal direction. The upper half of each figure is the latch with the test-plate (that simulates the door panel). The lower half of each figure is the striker on the test plate (simulating the B-Pillar). Left-upper figure shows the complete configuration. Right-upper figure shows only the latch and striker parts (without test rig). Left-lower figure is a cutting plane view through the catch. Small center inset image shows that the catch was pulled out. Graph at lower right indicates the reaction-force (in Newtons) reached during the test.

least amount of time. Use of the virtual validation process has enabled Kiekert to significantly reduce the number of tests for product design validation, cutting their time to market for a new latch to less than 18 months.

Abaqus unified FEA chosen for more complex analysis

The key to speeding up latch design and development at Kiekert has been Abaqus FEA software from SIMULIA. "Physical testing in a laboratory is slow and can't realize the whole job," says Waldmann. "We began our linear computer modeling of door latches with finite element calculations inside CATIA. Adding Abaqus enabled us to do more complex nonlinear simulation, which is what we need for accurate strength testing in three dimensions."

Abaqus Unified FEA works well with Kiekert's existing design software. "We can mesh complex parts in CATIA V4 and V5 and then export the meshes to Abaqus," says Waldmann. "We use Abaqus for CATIA V5, in conjunction with Abaqus/Standard, Abaqus/Explicit, and Abaqus/CAE. We never have any problems getting what we want, because we have different ways to get our solutions." Using the implicit and explicit solvers in Abaqus to solve the right type of problem is easy, Waldmann points out. "And modeling the nonlinearity of contact is something Abaqus does best."



Postprocessing Abaqus FEA image of a Kiekert latch module after strength testing shows how the striker starts to pull out the catch through the mouth of the frame plate.

Choosing the elements for meshing the lock design is often task-dependent. Automatic meshing is a must in order to build models efficiently for Kiekert's complex latch assemblies. By using tetrahedron elements, Kiekert can create a mesh in only half an hour—even though they need a little more CPU time for the job because they are modeling many small parts.

Waldmann's group has a 16 CPU Cluster and primarily uses a Linux operating system. Most Abaqus jobs at Kiekert are run on 4 CPUs, providing substantial simulation throughput and efficiency. "We can run up to four jobs at once, usually overnight," he says. They can also use UNIX platforms for calculation, or PCs with Microsoft Windows for Abaqus/CAE. An average job has between 700,000 and 1.5 million degrees of freedom.

Plastics versus steel and other design tradeoffs

Materials selection is important when modeling a virtual door latch. The first latches ever built were made entirely from steel and/or aluminum alloy. But today, multifunctional latches include plastic parts, even electrical motors, added for ease of function, noise reduction, or comfort.

Kiekert has established a unique materials database over the years. "Accurate material data is the basis of exact simulation results," says Waldmann. "Kiekert has invested

significant time and money in running scientific material tests to generate that data."

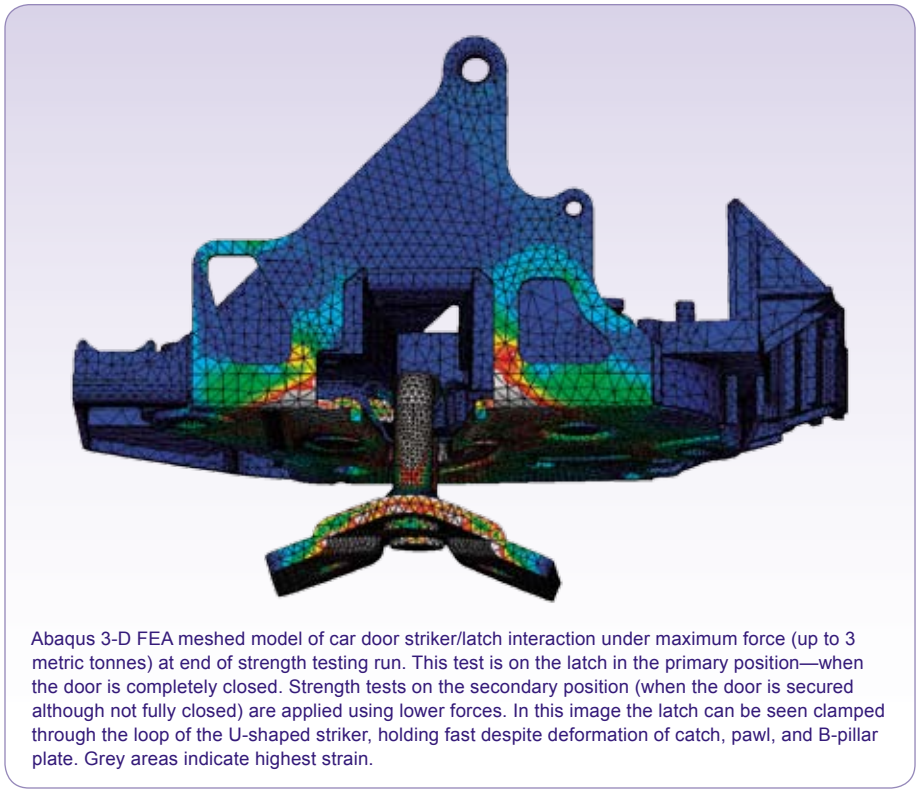
Using the Change Material function in Abaqus, Kiekert can evaluate plastics ranging from zero to 50 percent glass fiber content. But parts designed for safety must remain steel, Waldmann says, not only for strength but for fire resistance. "We have a long history working with different grades of steel with special heat-treatment capabilities. We search for the best balance between flexibility, strength, and abrasion. With the Change Material function, we can put a part through various stresses and see what works best. Sometimes we run the same job with the same geometry with five material variations."

Kiekert also uses FEA for evaluating temperature and noise/vibration, says Waldmann. "The parts of a latch, particularly the plastic ones, must continue to work between –40 and +85 degrees C (–40 to 185 degrees F). As for the noise testing department, they want more plastic to reduce noise whereas I want more steel for strength. We always need to satisfy both noise and safety considerations, but using Abaqus makes it easier to adjust your design and test the results."

Putting On the Pressure

When it comes to virtual strength testing, Waldmann's department comes down hard on their latch designs, loading 2–3 tons of virtual force (20,000N–30,000N) on the connection between latch and striker. All manufacturers require that strength tests are carried out along both longitudinal and transverse planes; sometimes vertical and/or transverse +45 degree rotation tests are included (rotational testing was a recent request from an OEM in the U.S.).

FEA testing provides a safer way to evaluate latches than pulling apart real steel parts in a laboratory. "When we do virtual loading with FEA, we can make cutting planes during testing to look inside and see exactly what happened without the danger of steel shattering during physical testing," points out Waldmann. Using FEA, Kiekert engineers can also evaluate the sequence of events more incrementally, as stresses are applied to a latch, and observe internal structural changes that may not be reflected in the appearance of an actual latch at the end of reality testing.



Abaqus 3-D FEA meshed model of car door striker/latch interaction under maximum force (up to 3 metric tonnes) at end of strength testing run. This test is on the latch in the primary position—when the door is completely closed. Strength tests on the secondary position (when the door is secured although not fully closed) are applied using lower forces. In this image the latch can be seen clamped through the loop of the U-shaped striker, holding fast despite deformation of catch, pawl, and B-pillar plate. Grey areas indicate highest strain.

Real Crash Data in the Future?

Since a latch is part of an entire door locking system, developing a theoretical forecast of complete latch system behavior during a crash will be the next step in increasing car door safety. "Kiekert has begun designing an integrated latch module in a holistic approach to optimize such door locking system behavior," says Waldmann.

For now, Waldmann's group has reached the point where FEA and other simulation tools they are using have enabled them to make the most realistic calculations in the latch industry. Such state-of-the-art simulation capabilities are one of the unique selling points that have made Kiekert the top latch supplier worldwide, according to Waldmann. "We spend a lot of time adjusting the geometry of our latches for the best result," says Waldmann. "The simulation department is an essential part of the Kiekert design process."

Postscript

Over 54,000 people a year in the U.S. are ejected from their vehicles during accidents, according to the National Highway Traffic Safety Administration. Although windows account for a majority of ejections, it is estimated that close to 7,500 people were thrown out of their car through an open door.

In an effort to reduce door-ejection statistics, the U.S., the European Union, Canada, and Japan have adopted the first global technical regulation (GTR) for motor vehicles to set minimum safety standards for door locks and retention components. Each country is aligning its domestic regulations with the world standard: voluntary compliance is in effect now in the U.S., becoming mandatory September, 2009, according to the National Highway Traffic Safety Administration. Most existing U.S. standards were established in the early 1970s.

For More Information
www.kiekert.de

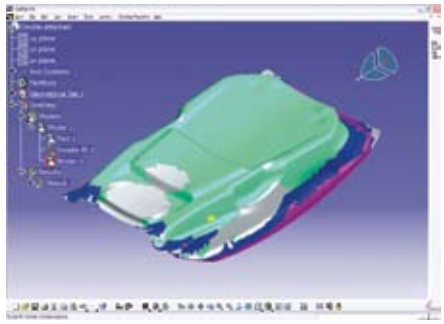
AutoForm Accelerates Sheet Metal-Forming Process in CATIA

AutoForm Engineering GmbH, a leading global supplier of software solutions, has recently become a partner in the Dassault Systèmes Software Community program. This partnership, managed by SIMULIA, enables AutoForm to deliver sheet metal-forming solutions integrated with CATIA.

"The integration of AutoForm capabilities within CATIA will enable our manufacturing customers to compress time in their design-to-production processes, which is especially valuable to the automotive industry," states Ken Short, VP Strategy and Marketing, SIMULIA. "As a Dassault Systèmes Software Community partner, AutoForm will work closely with our R&D teams to enhance the capabilities of their sheet metal-forming solutions directly in CATIA."

"The partnership with Dassault Systèmes is an integral part of AutoForm's strategy and will provide significant benefit to our customers who are leveraging CATIA for product lifecycle management," stated Dr. Waldemar Kubli, CEO of AutoForm Engineering. "By integrating and complementing our suite of solutions with the highly-productive CATIA portfolio, we will continue to deliver state-of-the-art solutions that enable our mutual customers to bring high-quality products to the market faster and more cost-efficiently."

AutoForm's solutions enable product designers, process planners, and tool designers to quickly and efficiently perform early manufacturability and cost assessments during their design process



in CATIA. They are able to set up the entire stamping process, make process modifications, and consider and evaluate different process layouts to identify the best ones within the context of their overall PLM process.

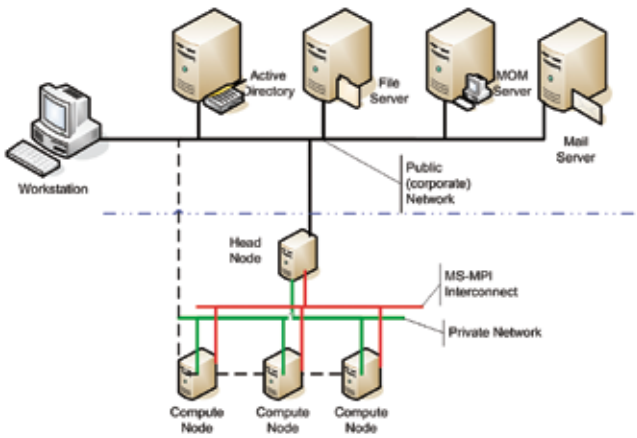
For More Information
www.autoform.com

Windows Compute Cluster Server 2003

One of the biggest issues that customers face today in adopting high-performance computing (HPC) solutions is the management and deployment of clusters and nodes. This problem has traditionally been a departmental or corporate-level problem, with a dedicated information technology (IT) professional staff to manage and deploy nodes, and users submitting batch jobs and competing for limited resources.

Windows Compute Cluster Server 2003 brings together the power of commodity x64 (64-bit x86) computers, the ease of use and security of Active Directory service, and the Windows operating system to provide a security-enhanced and affordable HPC solution.

A typical cluster configuration includes a single head node and one or more compute nodes. The head node controls and mediates all access to the cluster resources and is the single point of management, deployment, and job scheduling for the compute cluster. Windows Compute Cluster Server 2003 has been designed to provide a simple



Typical Windows Compute Cluster Server 2003 Network

and familiar interface for managing and administering the cluster. It uses existing corporate Active Directory infrastructure for security, account management, and overall operations management. For those companies already using Windows operating systems, it enables users to leverage their IT administrator's skill-set for managing an HPC system.

The combination of Abaqus distributed-memory parallel technology and Windows clustering solutions allows Abaqus users to decrease execution time for their jobs. With Abaqus version 6.8-1, the benchmark s2a, which is a mildly nonlinear static analysis of a flywheel with centrifugal loading, requires 30 minutes to run using four cores on a dual-processor/dual-core Opteron server. The same job running on a four-server Windows CCS cluster (16 total cores) requires 14 minutes.

Running Abaqus FEA Software on a Windows-based cluster provides engineers with an integrated, efficient, and easily scalable high-performance computing platform for solving some of the most complex simulation and multiphysics analysis equations in less time.

Key Features and Benefits

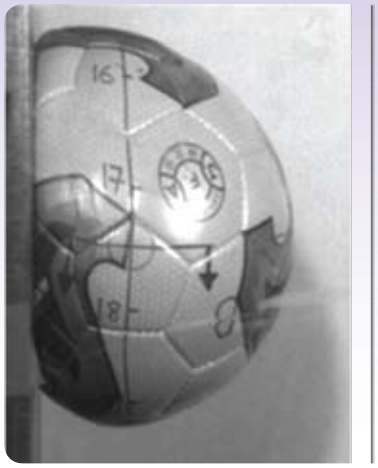
- Simplified Cluster Deployment and Management
- Better Integration with IT Infrastructure
- Broad Application Support
- Familiar Development Environment

For More Information
www.microsoft.com/hpc

Studying the Bounce of a Virtual Soccer Ball

With the explosive growth in popularity of World Cup Soccer, many sports companies are developing new high-tech designs and materials to improve the performance of soccer balls. Their goal is to create the optimum ball that is flight accurate, waterproof, travels fast, and transfers maximum kicking force to the ball rather than absorbing the energy. Yet the ball needs to have a soft feel, be safe for head hits, and adhere to specifications from governing sports bodies like FIFA.

The Sports Technology Research Group at Loughborough University in the U.K. has developed an FEA model of a soccer ball that is capable of predicting how the ball will bounce and deform under different playing conditions. This new breed of ball has an inner latex bladder like the standard type, but it has two layers instead of one on top of that. A middle layer, called a carcass, is made of stretchable fabric panels to which a third, outer layer of soft polyurethane panels is adhered.



This FEA model of a soccer ball from Loughborough University in the U.K. is capable of predicting how the ball will bounce and deform under different playing conditions.

The researchers used Abaqus FEA to model the middle layer of the ball, realistically simulating the many directions in which the fabric can stretch, depending on the alignment of its fibers. They next "bounced" the virtual balls against a hard surface from different angles—and found that when they repeated the experiment with real balls, the results matched their computer models very precisely. Now that they know how the inner parts of a soccer ball work, they can study the effect of adding the third layer of soft panels in the next-generation ball.

For More Information
www.lboro.ac.uk

Web-Based Abaqus Tutorials Accelerate Learning of FEA Principles at Rutgers University

In his graduate biomechanics class at Rutgers, the State University of New Jersey, Professor David Shreiber introduces students to concepts from continuum mechanics to the theory of linear elasticity. His students—typically first-year Ph.D. and M.S. candidates—come from diverse undergraduate backgrounds with a spectrum of experience in mechanics, and many have not developed the instincts or intuition to visualize states of stress and strain within a deformable body.

Having used Abaqus extensively in his own graduate studies and research, Professor Shreiber theorized that Abaqus could be an effective tool to help his students "see" the concepts he was describing. When he received a promotional offer from SIMULIA for a 20-seat Abaqus Teaching Edition license, he welcomed the opportunity to use Abaqus in his classroom.

Shreiber demonstrated—and his students soon discovered for themselves—that by



using Abaqus, numerical solutions to the analytical problems discussed in class were quickly achieved. From the simple deformation of a hanging bar under its own weight, to more complex problems that include geometries, boundary conditions, and loading conditions relevant to real-life biomechanics, Abaqus could be used to design and solve a wide range of problems in assignments. "I have been very pleased at the response and results," said Professor Shreiber. "The students were able to quickly build models, run analyses, and effectively communicate results."

To accelerate the learning curve, Professor Shreiber created a series of tutorials using a shareware program called Jing (www.jingproject.com). The software takes screen captures while Professor Shreiber builds, executes, and analyzes a model in Abaqus, combining the images that follow his mouse-clicks and menu selections into a Shockwave video for student viewing.

"After using Abaqus in class, some of my students were motivated to purchase the Abaqus Student Edition. Others are seeing the potential to use Abaqus in their academic research," Professor Shreiber continued. "The students also indicated that the Abaqus projects helped them understand the analytical concepts from the classroom, and I found that exam grades were improved compared to previous years. I am looking forward to incorporating Abaqus into our curriculum at Rutgers even more next year."

For More Information
www.rutgers.edu

Easing the Pain of Hip Replacement Surgery at Cedarville University

3-D Analysis Assists in Predicting the Stability of an Implant

A group of undergraduate engineering students at Cedarville University, led by Professor of Mechanical and Biomedical Engineering Timothy L. Norman, used Abaqus to model the push-out load between a hip prosthesis stem and bone during a senior design project. When used along with a device that maps out the geometry of the femoral intramedullary canal, the information generated from this analysis could aid an orthopedic surgeon in real-time prosthesis size selection during hip replacement surgery.

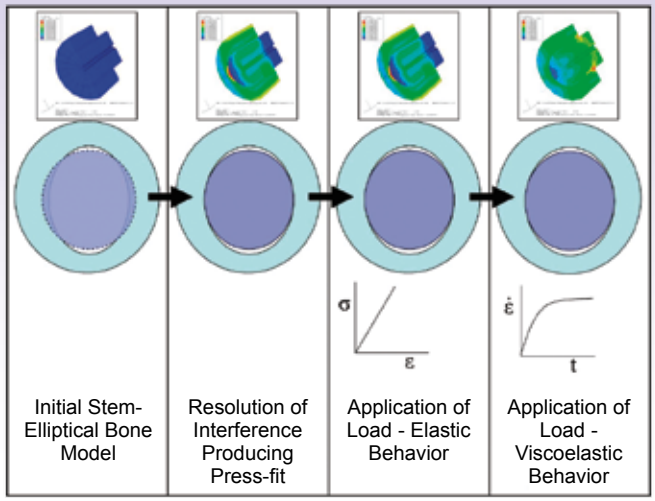


Diagram of the Stem-Elliptical Bone Model (SEBM) at various analysis steps.

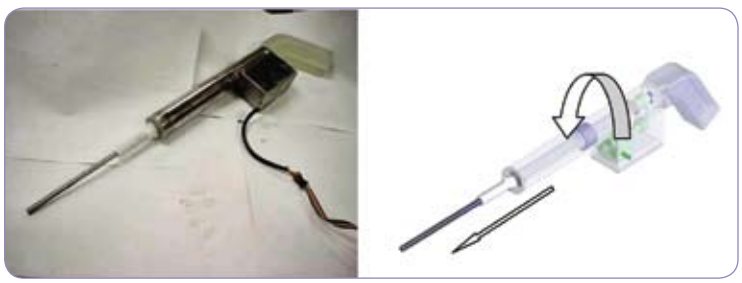
Total hip replacement is a common surgery used to correct an arthritic or severely damaged hip joint. In the cementless type of this surgery, the prosthetic device depends upon stem-bone contact area and "press-fit" for initial stability, and then later on bone growth into the porous stem for long-term stability. Failure to achieve long-term stability after surgery requires subsequent revision surgery, so it is critically important to both surgeon and patient to get it right the first time.

A potential cause of post-surgery instability is inadequate press-fit, which may be due to too little contact area and interference caused by the difficulty in sizing both the reamer—the device used to create the hole in the femur—and the prosthetic shaft that is inserted into the reamed bone. Often, the reamed intramedullary canal has an irregular oval-shaped cross-section that may be different than the shape of the stem if the bone is not completely reamed. If the geometries differ more than the planned interference, the contact area is reduced, and the press-fit will be compromised. Better knowledge of the shape of the intramedullary canal and stem prior to reaming and stem installation could aid surgeons in selecting proper reamer and stem sizes.

At this time, the most common means of estimating the size and shape of the intramedullary canal is by x-ray, which provides a 2-D view of an irregular 3-D space from which the surgeon estimates the best fit for the prosthetic shaft. As a result, the surgeon may resort to trial-fitting different reamers and shafts to obtain the best feel, which has its risks: if the shaft is too large in comparison to the canal, the bone will fracture, and if the shaft is too small, there may be inadequate contact area or press-fit and the implant may be unstable. The difficulty in determining the proper press-fit may be one of the major causes of implant instability.

To help solve this problem, the Cedarville University Undergraduate Design Team wanted to find a way to provide orthopedic surgeons with real-time 3-D information about the shape and size of the femoral canal that can be used in prosthesis selection. The undergraduate engineering students designed a femoral canal sizing device (Oculus) used to collect femoral canal measurements, and performed the analysis using Abaqus over the course of a two-semester capstone senior design project. Jason Auyer, John Simmons, Matt Spena, and Mary Todd Hoffner were advised by Dr. Tim Norman on the project, and Mary Todd Hoffner performed the finite element analysis. Abaqus was used to predict stem push-out loads, and the analysis was split into three non-overlapping individual steps, which occurred in chronological order: resolution of the interference, application of the load with elastic response, and application of the load viscoelastic response.

Project results showed that correlations between contact area—generated by the Oculus—and push-out loads calculated using Abaqus can be used to approximate the stability of an implant. These data can give a surgeon immediate access to accurate contact information based on actual intramedullary canal measurements and push-out load predictions based on Abaqus FEA and can, therefore, be used to aid the surgeon in determining the optimal stem size.



Femoral Canal Sizing Device (Oculus)

For More Information

simulia.com/academics/t_projects_canal
www.cedarville.edu

A New Era for Engineering Services at SIMULIA

Mike Ricci, Director Global Services,SIMULIA

As SIMULIA grows, in both the number of people we employ and products we offer, we are placing renewed emphasis on our global services strategy and offerings.

Due to our established technical expertise, global operations, and affiliation with our parent company, Dassault Systèmes, our engineering professionals serve our customers in a variety of areas. We are able to provide assistance in the use of Abaqus Unified FEA software through analysis project consulting, and we also support the implementation of enterprise simulation solutions. Our services are proven to improve quality, reduce cost, and deliver measurable efficiency gains to your organization.



Continuous Care

Our regional service teams are well-known for performing engineering analysis projects on an as-needed basis. We are adopting a "continuous care" services strategy where our engineering professionals work with you to review your current processes, challenges, and goals, then create a plan that can be followed to achieve the greatest efficiencies and long-term return on your investment. This strategy benefits our customers by planning the right service resource at the right time.

From consulting projects, to on-site engineers, to training services, our teams are available to apply the right level of resources to fit your demands and schedules. It is our goal to make you more productive and efficient in using realistic simulation tools. To that end, we have created new training classes for Fluid-Structure Interaction and Simulation Lifecycle Management, as well as courses that support specific industry workflows for Offshore Oil and Gas and Medical Devices.

SLM Implementation

With the release of our new Simulation Lifecycle Management product, our engineering services teams are available to provide SLM implementation services. This type of service helps your organization integrate your simulation processes, data, and applications within a managed environment. SIMULIA SLM can be leveraged as a standalone solution or integrated into existing Product Lifecycle Management environments. The ultimate goal of this product—and of our services—is to enable you to secure your simulation-generated intellectual property and transform it into a controlled corporate asset.

In the coming months, when you are wondering how you will get all of your work done, consider having a conversation with one of our regional Business Development Managers about which of our service offerings is right for you.

SIMULIA Services Portfolio

- Quick Start Program
- Engineering Analysis Consulting
- Training: Standard, Custom, and On-site
- Packaged Abaqus Extensions
- Methods Development
- Mentoring Program
- Abaqus Customization
- Analysis Process Automation
- Onsite Engineering Consulting
- Project Management
- Simulation Lifecycle Management Implementation

Sample Consulting Projects

- Linear and Nonlinear Stress
- Nonlinear Contact, Including Drop or Impact
- Fluid-Structure Interaction
- Thermal Analysis
- Fatigue, Fracture, and Failure
- Durability Analysis
- Noise and Vibration
- Manufacturing Processes, such as Metal Stamping, Spin Forming, Forging, Cutting, Rolling, Embossing, Material Handling, Boring, Milling, and Welding

For More Information
simulia.com/services

Peter Nannucci Recognized by Boeing



The Boeing Company recently awarded Peter Nannucci, Sr. Services Engineer at SIMULIA Eastern Region, their Quality Pride Award. This award recognizes Peter's outstanding performance in his role as a Boeing On-site Senior Abaqus Support Specialist.

"Peter exceeded our expectations," stated Frank Smith, Associate Technical Fellow, Structural Technology and Prototyping at Boeing Rotorcraft. "He developed an outstanding reputation for the quality of his work—both directly supporting our various programs and in his consulting capacity."

"We are extremely pleased to hear that Boeing has taken the initiative to recognize Peter for his outstanding contributions to their programs," stated Bill Brothers, Boeing Account Manager, SIMULIA. "Peter's award reflects SIMULIA's commitment to providing high-quality engineering services that make a positive impact on our customer's product development processes."

The Quality Pride Award was awarded to Peter by Frank Smith, Ahsan Iqbal (Sr. Manager, Structural Analysis), and Mike Warburton (Sr. Manager, Mechanical and Structural Engineering), all of the Boeing Rotorcraft division in Philadelphia.

SIMULIA Training Schedule & Registration

SIMULIA Regional Offices and our Representatives offer regularly scheduled public training courses on the use of Abaqus, Multiphysics, and SIMULIA SLM. There are also Affiliate training courses provided by our partners on products used to extend the capabilities of Abaqus. An extensive schedule of courses and locations is available, ranging from basic introductions to advanced courses that cover specific analysis topics and applications.

Examples of courses available include:

- Introduction to Abaqus
- Abaqus/Explicit: Advanced Topics
- Automotive Powertrain Assembly Analysis with Abaqus
- Coupled Eulerian-Lagrangian (CEL) Analysis with Abaqus/Explicit

Please check the course availability online to find the course and location to fit your needs (note: you can select a 6-month or full-year view). Customized or on-site training courses are offered as a services engagement by the regional offices.



For More Information
simulia.com/services/training_services

2008 RUM Schedule

Attend the upcoming Regional User's Meeting in your area. Learn about the latest enhancements to Abaqus FEA and the future direction of SIMULIA. For additional information on the Regional Users' Meeting in your area, visit your local office website.







Americas

Date	Location
September 18–19	West Lafayette, IN
September 23	Beechwood, OH
September 24–25	Cincinnati, OH
October 1–2	Minneapolis/St. Paul, MN
October 15	Houston, TX
October 28–29	Toronto, Ontario
November 5–7	San Diego, CA
November 12–13	Plymouth, MI
November 13–14	Baltimore, MD
TBA	São Paulo, Brazil

Europe/Middle East/South Africa

Date	Location
September 22–24	Bad Homburg, Germany
September 25	Athens, Greece
September 25–26	Helsinki, Finland
October 1–2	Bristol, UK
November 4–6	Milano, Italy
November 6–7	Istanbul, Turkey
November 10–11	Linz, Austria
November 13–14	Antwerpen, Belgium
November 13–14	Madrid, Spain
November 20	Paris, France
November 20–21	Poznan, Poland
November 25	Herzelia, Israel
TBA	Johannesburg, So. Africa
TBA	Prague, Czech Republic

Asia Pacific

Date	Location
October 7–10	Kyungju, Korea
October 28–29	Taipei City, Taiwan
October 29–30	Tokyo, Japan
November 6	Beijing, China
November 10–11	Bangkok, Thailand
TBA	Bangalore, India

For More Information
simulia.com/events/rums



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, 2008.