INSIGHTS

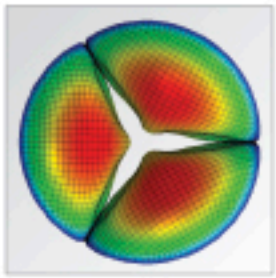
11
9 2010
Dassault Systèmes Realistic Simulation Magazine



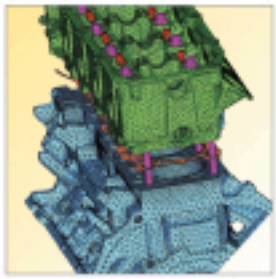
Smith & Nephew Improves Its Knee Replacements



Bausch + Lomb
Visualizing Eye Surgery



University of Pittsburgh
Advancements in Material Models

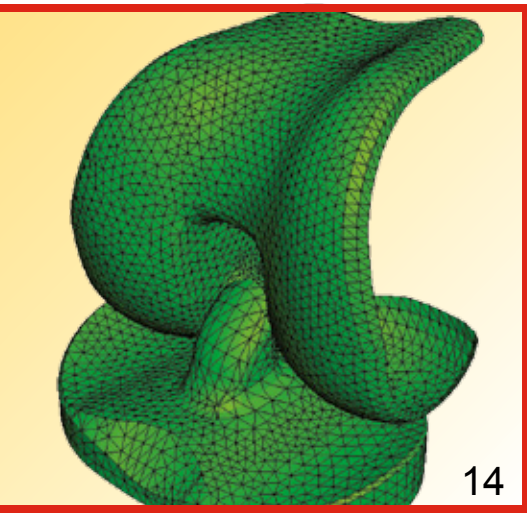


Ford Motor Company
Improves Powertrain Design

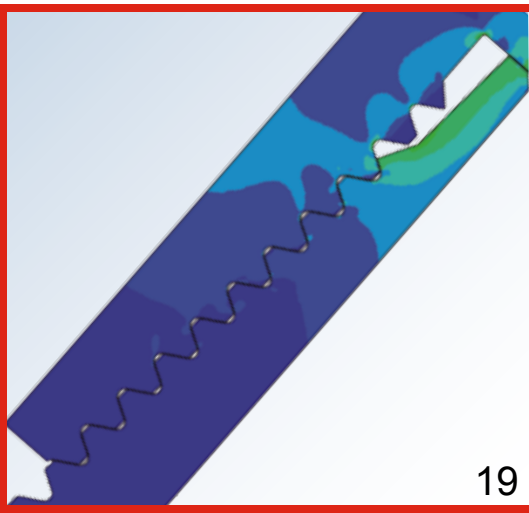
SIMULIA

INSIGHTS

September/October 2010







Inside This Issue

14 Cover Story
Smith & Nephew Studies Performance of Replacement Joints with Abaqus FEA

On the cover: Bernardo Innocenti

8 Customer Spotlight
Bausch + Lomb Brings Prosthetic Lens Insertion into Focus with Abaqus FEA

19 Product Update
Abaqus Extension for Threaded Connections

In Each Issue

3 Executive Message
Dale Berry, Director, Technical Marketing, SIMULIA

4 Customer Viewpoint
Michael S. Sacks, Ph.D., John A. Swanson Endowed Chair in Bioengineering, University of Pittsburgh

6 Customer Case Study
Ford Improves Powertrain Design with Abaqus for CATIA

10 Solution Brief
Mobelife Leverages Realistic Human Simulation for Hip Revision Implants

12 Realistic Human Simulation Strategy Overview
Subham Sett, Life Sciences Lead, SIMULIA

16 Customer Case Study
Sumitomo Calls on SIMULIA for Answers to Cell Phone Cable Design Challenges

18 Customer Spotlight
Foster Wheeler Improves Efficiency and Design Quality with Customized Automated Simulation Tool

20 Customer Case Study
Wölfel Improves Seat Design with Realistic Human Simulation

23 Alliances
- Simpleware
- Intel

24 Academic Update
- Hadassah University
- Technical University of Denmark

26 In The News
- Research and Development Establishment (Engineers)
- Senergy

27 Events
2011 SIMULIA Customer Conference

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

www.simulia.com

Editor:
Tim Webb

Associate Editor:
Karen Curtis

Contributors:
Bernardo Innocenti and Luc Labey (Smith & Nephew), Robert Stupplebeen (Bausch + Lomb), Tim Clijmans and Frederik Gelaude (Mobelife), Jinming Xu (Foster Wheeler), Alexander Siefert (Wölfel), Rebecca Bryan (Simpleware), Frank Ding (Simpson Strong-Tie), Eran Peleg (Hadassah University Medical Center), Lars Mikkelsen (Technical University of Denmark), Niels Lynnerup (University of Copenhagen), Michael Sacks (University of Pittsburg), Parker Group, Dale Berry, Subham Sett, Gaetan Van den Bergh, Mike Schubert, Mahesh Kailasam, and Matt Ladzinksi (SIMULIA)

Graphic Designer:
Todd Sabelli

The 3DS logo, SIMULIA, CATIA, 3DVIA, DELMIA, ENOVIA, SolidWorks, Abaqus, Isight, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners. Copyright Dassault Systèmes, 2010.

OCT_INS_Y10_VOL 11



Advanced Technology + Industry Focus = Innovation

It is likely that everyone reading this issue of INSIGHTS knows a family member or friend who has received some kind of medical implant such as a knee replacement to ease joint movement or a stent to improve blood flow. The accelerated pace of such medical advancements is, for the most part, enabled by the efforts of research engineers at world-leading universities and medical device manufacturers. SIMULIA is actively engaged with these experts and has a strategic focus on enhancing our realistic simulation technology to help drive medical device and implant innovation.

This engagement and commitment to the Life Sciences industry is an integral part of our brand's strategy to provide complete realistic simulation functionality, including linear, nonlinear and multiphysics capabilities, which enables the evolution of the associated methods to simulate real-world behavior of materials and products. In addition to creating technology roadmaps for Life Sciences, Aerospace and Automotive, our Technical Marketing team has also created roadmaps for Electronics, Energy and Consumer Packaged Goods. I'm now pleased to report that we have further expanded our industry focus to include Shipbuilding, Defense and Turbomachinery as well. Our goal in doing so is to deliver increased value to as broad a cross-section of customers as possible by providing the functionality and methods that meet the demands of their industry-specific workflows.

This issue of INSIGHTS highlights several of our customers' accomplishments in realistic human simulation and interaction with medical devices. While there are more simulation technology advancements still to come in this area, the success already being achieved by our customers is astounding. The cover story features researchers at Smith & Nephew who are helping expand range-of-motion and useful lifespan of knee replacements with realistic simulation (page 14). You'll also find thought-provoking articles on the use of Abaqus by researchers at the Technical University of Denmark, Hadassah University, Mobelife and Bausch + Lomb. While these groups are working on diverse projects, they share a common commitment to gaining a deeper understanding of the function of the human body and to developing innovative medical treatments that improve the quality of life for our families, friends and society.

Collaboration and technical exchange with our customers is a critical part of our ability to deliver solutions to solve specific industry workflows. As this issue goes to press, many of the SIMULIA Regional User Meetings (RUMs) have taken place, but there are still several others coming up. With nearly 30 RUMs being held this year, we expect that close to 3,000 members of our global customer community will have met to share their experiences and learn about our current and upcoming technology enhancements for unified FEA, multiphysics, design optimization, and simulation lifecycle management.

In these meetings with our regional professionals, you will learn about our continuing commitment to enhancing our products to handle advanced materials and complex analyses, such as crack growth and large-scale linear dynamics. You will also hear about our strategy for leveraging high-performance computing clusters to help you tackle larger models, with higher level of realistic details.

Looking forward to 2011, I am happy to invite you to attend the annual international SIMULIA Customer Conference (SCC) to be held in Barcelona, Spain, on May 16–19. By participating in the 2011 SCC, you will be able to meet with SIMULIA's technical and industry leaders and share your industry-specific experience and requirements. Our Technical Marketing team will be interacting with as many of our customers as possible in the coming months to continue fulfilling our commitment to meet your needs. We view this as a global community effort with you, our loyal and valued customers, as key members of our extended team.



Dale Berry
Director, Technical Marketing,
SIMULIA

Cardiovascular Soft-Tissue Simulation Requires Advancements in Constitutive Material Models

Michael S. Sacks, Ph.D., John A. Swanson Endowed Chair in Bioengineering, University of Pittsburgh



Predictive computational modeling of the cardiovascular system has often been used as a powerful investigative tool. Motivated by the need for a deeper understanding of the underlying physiology, identification of pathological initiators, as well as the development of bioprosthetic devices, a broad variety of modeling approaches have been introduced into the literature. Central to system- and organ-level functional simulations is the need for robust and physiologically-meaningful constitutive models of the underlying soft tissue structures.

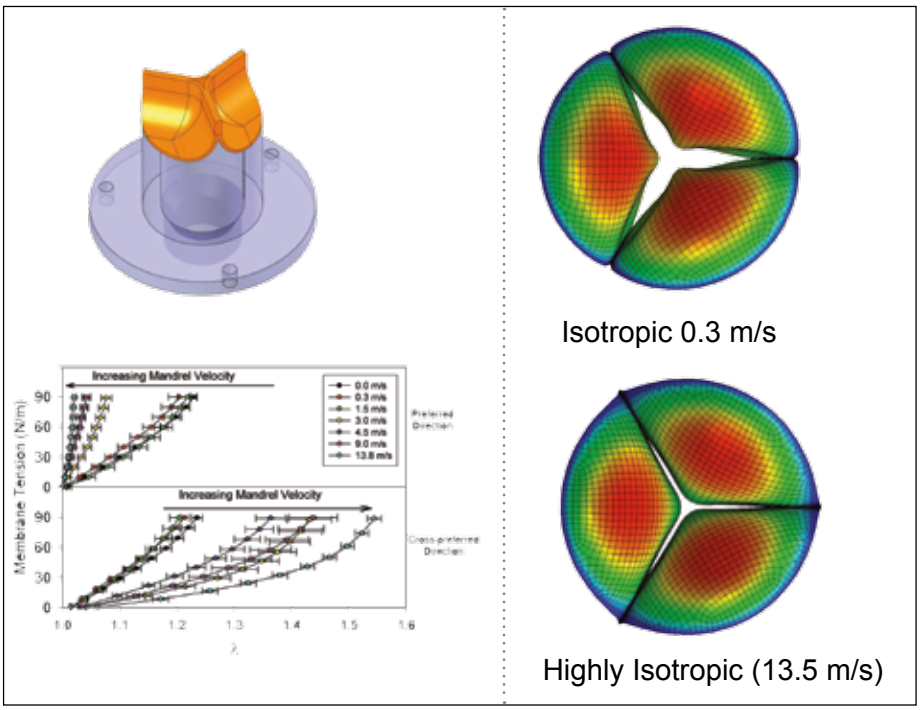
The field of soft tissue mechanics has been popularized by Y.C. Fung through a range of influential books that demonstrate the unique challenges involved in the mathematical characterization of living tissue mechanical behaviors. Particular challenges in soft tissue constitutive modeling are encountered due to their complex mechanical behavior. For example, because of their oriented fibrous structures they often exhibit pronounced mechanical anisotropy, nonlinear stress-strain relationships, large deformations, viscoelasticity, poroelasticity, and strong mechanical coupling. Taken as whole, soft biological tissues defy the use of simple material models. This has stimulated in recent years the desire by the biomechanics community to develop and implement soft tissue models within a computational framework. As the field embraces the challenges of developing robust models suitable for computational implementation, issues such as convexity and stability have motivated researches to recast models into forms that support a continuum approach.

At present, soft tissue biomechanical modeling in the cardiovascular area is an active area of research, as investigators struggle to bridge the gap between the mature field of mechanics and the evolving biological sciences. In particular, there is a recent trend for investigators to increase their focus on models based on tissue microscale features and function, as well as mechanisms contributing to mechanical responses associated with varied length scales. Of primary interest is the complex function and micro-structural architecture of the extracellular matrix fibrous constituents. The collagen fiber network is of interest because it is the primary load-bearing tissue constituent and exhibits multi-scale characteristics that influence organ level anisotropy as well strongly interacting with the cellular constituents. In a predictive model framework, relating these characteristics to scale-relevant tissue behavior can produce a higher level of realism. This includes efforts to address pathology and engineered tissue development that looks towards microscale based models for insight and guidance. Relating the observed mechanical response to tissue structure is perhaps more paramount than in other traditional material applications, where the continuum scale is usually, at most, the size of large polymer molecules. In contrast, biological soft tissues are comprised of a dense network of primarily collagen and elastin various fibers, which indicates a continuum scale at the fiber-scale (typically ~1 μm). In addition, the fibers can undergo large rotations and exhibit nonlinear stress-strain behavior that can induce complex behaviors at the macro-specimen scale not easily accounted for in classic elastomeric material models. Accounting for these behaviors in both experimental evaluation and formulation of appropriate constitutive models continues to be challenging.

The Study of engineered tissue

Originally introduced to describe the behavior of skin tissue, the strain energy function proposed by Tong and Fung (now known as "Fung Elasticity") has provided a reliable foundation for soft tissue mechanics. Researchers have successfully used variations of this model to describe a number of soft tissue applications including skin, blood vessels, myocardium, and heart valvular tissues. As an example of the utility of Abaqus in the analysis of soft tissue, we conducted simulations on engineered heart valve tissues. Tissue engineering (TE) describes the process of combining engineered materials with living cells to produce viable structures for the replacement of diseased or deficient native tissues. The challenges facing TE researchers involve the contrasting requirements between favorable tissue growth conditions and functional in vivo properties. Scaffolds fabricated by electrospinning synthetic polymers, or polymer blends have received widespread attention, due in part to the ability to impart controlled mechanical anisotropy by variations in the fabrication process. This is extremely beneficial in mimicking native tissue architecture and has even been shown to approximate the highly nonlinear biaxial mechanical response of collagenous soft tissues. Computational simulations need to be directed at determining the scaffold properties that can provide the maximum benefit from a biomechanical point of view. If the fabrication process can control these properties, what properties should be targeted to mimic the homogenous strain field of native tissue? And, to what practical extent can this information guide the fabrication process to produce TE material with improved qualities? In addition to material anisotropy, the initial undeformed scaffold shape can be manipulated.

To begin to address these questions, finite element simulations using a stented leaflet design intended for organ level bioreactor



(Left) TEHV in-vitro bioreactor stent design, along with stress-strain data of ES-PEUU scaffold showing the influence of fabrication mandrel speed on the resulting biaxial mechanical properties. (Right) The influence on the resulting leaflet coaptation is demonstrated by the deformed shaped of scaffolds prepared with mandrel speeds of 500 and 2300 rpm.

studies were conducted under 80 mmHg trans-valvular pressure for isotropic and anisotropic scaffolds (Fig. 1). The circumferential direction was taken as the preferred material direction for all simulations. The biaxial mechanical properties of ES-PEUU scaffolds were modeled using the Fung model, as for pericardium, in Abaqus through custom written UMAT subroutines along with the material parameters the scaffold type. Quadratic hexahedral elements were used to model the leaflet. Overall geometric characteristics indicated that without sufficient mechanical anisotropy, leaflet response in the radial direction cannot develop the strain magnitude required to permit engagement with adjacent leaflets. Note that the initial scaffold geometry was constant for each case and was representative of a "near" closed condition.

Simulating to scale

While these phenomenological models have been successfully applied to a broad spectrum of soft tissue applications, they lack the ability to capture the underlying mechanisms contributing to tissue behavior. Understanding the link between these underlying physiological functions is central to the development of meaningful constitutive models. Examination of pathological states and engineered tissue through computational simulation require a model foundation based on microstructural architecture. A class of models, referred to as "structural", attempts to characterize organ level tissue response in terms of fibrillar scale properties. These models are based on physiological microstructural features quantified through experiential means. Key tissue features such as fiber orientation and crimp period are directly incorporated into the model.

Researchers have used structural-based models to describe heart valve mechanical response. Structural constitutive models have been developed for a variety of intact tissues and tissue components including lung, collagen, cartilage, passive myocardium, heart valves, and maturing skin. Perhaps the most complete approach for structural constitutive modeling for soft tissues has been developed by Lanir et al. In this approach, the tissues total strain energy is assumed to be the sum of the individual fiber strain energies, linked through appropriate tensor transformation from the fiber coordinates to the global tissue coordinates. However, the description of fiber scale properties such as orientation and crimp are cast in terms of statistical distributions due to a microstructure complexity that prohibits individual representation. This stochastic description is based on data homogenized at a representative element scale of approximately 100 μm. At this scale, characterization of the fiber microstructure is relative to the fiber ensemble. In this sense, structural models have a meso-scale focus as they characterize an organ scale response in terms of fiber structure attributes at a scale in-between cellular and organ.

However, critical structural information (such as fiber orientations) modeled using assumed statistical distributions estimated the fit to the mechanical testing data. We have shown that through the introduction of strategically applied structural data, such as fiber orientation, analytical functions suitable for direct implementation can be developed for detailed numerical implementation. In the near future, direct incorporation of such structural data on multiple scales will allow a level of bioengineering wherein computational approaches can begin to realistically be applied.

Acknowledgements

We would like to acknowledge funding by NIH/NHBLBI grants HL68816 and HL071814, as well as National Science Foundation Grant BES-9978858.



Michael S. Sacks, Ph.D., holds the John A. Swanson Endowed Chair in Bioengineering at the University of Pittsburgh. His research is focused on the quantification and modeling of the structure-mechanical properties of native and engineered soft tissues, with a focus on tissues of the cardiovascular and urological systems. Dr. Sacks holds a Ph.D. in Biomedical Engineering from the University of Texas Southwestern Medical Center at Dallas, 1992, as well as a M.S and B.S in Applied Mechanics from Michigan State University.

For More Information
www.engr.pitt.edu/bioengineering
Email: msacks@pitt.edu

Powertrain Symphony in CAD & CAE

An initiative at Ford to use Abaqus for CATIA with model templates ups the tempo of powertrain design



Designing and building a car is like composing and performing a symphony. CAD designers and CAE analysts act like a team of composers, drawing upon their creativity, the laws of physics, and a host of engineering software tools to create a design "score." Then the instrumental sections—the powertrain, electrical, exhaust, steering, and other systems—bring the composition to life. Revisions and rehearsals follow. Finally the premier arrives, and when everything is in sync, the orchestra of components produces an on-the-road performance in perfect pitch. With the right talent and tools, the results can be music to the market.

In more straightforward terms, designing an automobile is an extremely complex endeavor. Ford Motor Company's North America Engine Engineering Organization, for example, has more than a hundred CAD designers and CAE analysts in the Powertrain Analytical Design and Six Sigma (ADSS) department alone. This team has responsibility for the design of all of the powertrain components, including the cylinder block and head, connecting rods and crankshafts, pistons, turbo chargers, and valvetrains. To manage this task with a Six Sigma mindset and develop the most robust designs in the shortest amount of time is a challenge that requires precise inter- and intra-departmental coordination, robust engineering tools, and well-tuned processes.

With those coordination goals in mind, Ford created a global program to improve product development efficiency, increase throughput, and deliver 100 percent geometric compatibility. As part of this PLM effort, they implemented a series of digital innovation initiatives, one of which—Digital Vehicle Engineering (DVE)—includes development of multiple intelligent CAD templates with tight integration of CAE and optimization modules. The goal is to promote enhanced collaboration among engineers, designers, and analysts in a virtual product design and verification environment.

1st Movement: CAD and CAE integration

About five years ago Ford made the decision to migrate all CAD model building to CATIA, the Dassault Systèmes' brand for virtual design and product innovation. "What made the difference for the management team was the capability of CATIA to integrate CAE tools," says Jeffrey Bautz, Ford's ADSS manager. "They recognized that the resulting efficiency improvements would be significant."

The ADSS team saw potential benefits for the powertrain system and was one of the first groups to use the integrated CATIA CAD/CAE solution for production inside Ford. To implement this solution, the team choose Abaqus for CATIA (AFC), a solution from the Dassault Systèmes' SIMULIA brand that brings the FEA capabilities of Abaqus into the CATIA environment through two CATIA workbenches—nonlinear structural analysis and thermal analysis.

When the ADSS team began using AFC to integrate CAD and CAE within CATIA, they were able to greatly accelerate the analysis process. With the CAE model and the CAD geometry easily accessible in one interface, the workbenches enabled the team to do multiple iterations quickly. Using an integrated platform, there is associativity between the geometry and analysis models, no time delays between steps, and a much more streamlined workflow.

2nd Movement: template development

In search of even greater efficiencies, the ADSS team next focused on automating the CAD and CAE processes and recognized the Abaqus for CATIA environment as being robust enough to handle this task. With this in mind, the team embarked on a path of developing software templates to facilitate process automation—starting with CAD templates about two years ago, followed by the CAE templates roughly one year later. The team realized, in developing templates to use inside CATIA, that there were tremendous opportunities to improve product development cycle times.

"With integrated CAD/CAE templates as part of our DVE strategy, we are able to accelerate the initial geometry and analysis generation process," says Sassan Khoubyari, PLM Strategy and Implementation Manager. "This allows CAE to drive design upfront, rather than validating later in the design process."

Since an analysis is only as good as the analyst's assumptions, the team spends a tremendous amount of time developing their methodology. The process consists of multiple iterations between physical test data and the model to ensure correlation. Once assumptions are validated, capturing their value is important. Templates do that, standardizing the conditions and variables for a model or simulation. They can then be used to guide each engineer on the team down a single analysis path that's proven and repeatable.

Once a method for building a complicated model—like for a cylinder head— has been developed, the analyst must still apply a huge number of different boundary

conditions, contact elements, and loads. It is possible to have cases with 250 to 300 different types of boundary conditions in a single model. Before templates, most of that work was done manually. With hundreds of components in the powertrain and multiple iterations for many analyses, it's easy to imagine the extra time spent cross-checking designs. Simply put, templates minimize the potential for human error while saving time.

To get started, the ADSS team used a 6-Sigma-like approach and developed value stream maps for all of the major engine components—for the cylinder head, the block, the connecting rod, the exhaust and intake manifolds, to name just a few. The value stream maps were used to prioritize component template development, the goal being to identify those templates that could potentially improve the product development cycle most dramatically.

First, the team chose to test an oil filter adaptor analysis, because it was relatively simple, could be completed quickly, and could serve as a template proving ground. This analysis template included parameterized ribbing (see Figure 1). They also chose a cylinder head lift deck rigidity analysis, a much more complicated project, because it would test CATIA's ability to handle complex CAE templates. This analysis template included a variety of features: automatic set-up of 51 contact pairs and 71 constraints; creation of parameterized components including steel plates, head bolts, plugs for spark plug and injector holes; elastic-plastic material property of head bolts; geometry partition and grouping of a combustion chamber surface to define the mesh boundary for a cylinder pressure application; five analysis steps, including press-fit of valve seats, bolt down of a cylinder head with steel plates and, finally, peak combustion pressure application in each cylinder head respectively. Both analysis templates were attractive candidates because they are current production programs in the early phases of development where there is a tendency to do many iterations.

The team uses Excel spreadsheets (that are attached to the templates) with all the key parameters. The CAD template defines the geometry. The CAE template includes the basic information for the simulation—the mesh, load, and boundary condition requirements.



Figure 1. Model for an oil filter adaptor NVH (noise, vibration, and harshness) analysis.

Because the templates are linked, a CAE analyst can easily change the key parameters, which then automatically update the geometry along with the mesh in the analysis model. In addition, for further consistency across the department, the CAD and CAE groups now have standardized hardware and are working on high-end PCs in a Microsoft Vista environment.

3rd Movement: benchmarking efficiencies and savings

Two years into the CAD/CAE integration, the ADSS team has made improvements in the product development cycle for a number of powertrain components. The team now has CAD templates for all the major components and has begun to actualize the return on investment, with long-term impacts lining up to be significant.

For an accounting of the specific improvements, Bautz turns to his team leaders. According to John Norcut, CAD template development manager, the oil filter adaptor analysis has been greatly improved. "By eliminating the CAD-to-CAE-to-CAD hand-offs, there has been a savings of three to four weeks overall in product development cycle time," Norcut says.

"For the cylinder head deck lift analysis," says Alex Tang, CAE technical specialist in charge of the CAE template development effort, "it used to take an analyst one to two days to set up the model. But with the template and a CAE-ready model, set-up time has been reduced to less than 30 minutes."

Improvements are equally dramatic for other components. To mesh the connecting rod for a dynamic analysis, it used to take an experienced analyst as much as four to eight hours. With templates, it can now be done in as little as 10 minutes if the CAD model is clean. For an intake manifold burst analysis, mesh generation has been collapsed from three weeks to only about two hours. For a connecting rod durability analysis, a one-and-a-half week mesh-time has shrunk to minutes.

As a result of these gains, the ADSS team is looking at ways of bringing additional analysis tools inside CATIA as well. Their plan for the future includes the use of SIMULIA's Isight optimization software. This tool provides a visual and flexible process of automating the exploration of design alternatives and identifying optimal performance parameters.

4th Movement: quality on the road

The CAE integration and template effort will have a number of long-term impacts, including a change in workflow: CAD designers and D&R engineers, rather than CAE analysts, will be able to handle many of the simple analyses. As a result, analysts will gain time to tackle more difficult problems—such as higher-end analyses and new methods development—that require their level of training and expertise. This work-load balancing will further improve design validation efficiency because every new method will allow the team to eliminate a hardware test. And fewer hardware tests mean substantial cost savings.

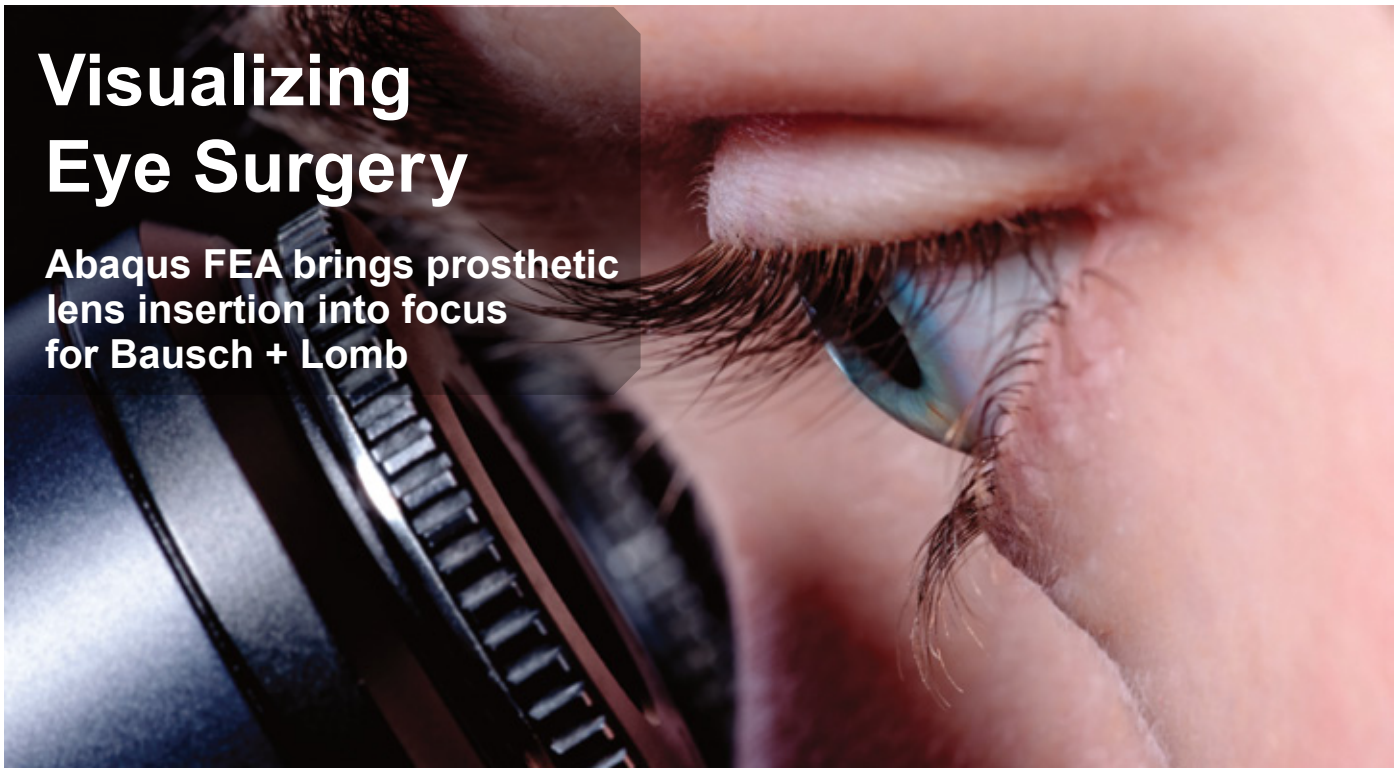
The template initiative has had such positive results that it is now being deployed throughout Ford's operations globally. What's more, the initiative is in synchrony with Ford President and CEO Alan Mulally's ONE Ford plan to accelerate development of new products that customers want and value.

From powertrain solo to full automotive system symphony, the results of design improvements at Ford are already on the road today, and the consumer is the ultimate beneficiary. For the past three years, Ford vehicles have been statistically proven to be equivalent in quality to those of its leading competitors. With CAE integration now added to the design composition, the harmonies of automotive performance are only going to get that much tighter and sweeter.

For More Information
www.ford.com
www.simulia.com/products/afc-v5

Visualizing Eye Surgery

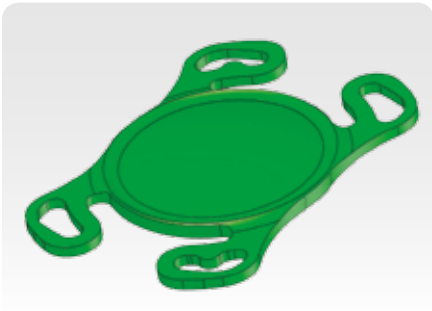
Abaqus FEA brings prosthetic lens insertion into focus for Bausch + Lomb



The medical establishment already has a very clear picture of cataracts and how to treat them. That's encouraging, given the fact that by age 80, more than 50 percent of all Americans will have developed a cataract, and every year more than three million will undergo eye surgery to correct it. What is also encouraging is the surgical outcome: the success rate is 95 percent, with vision typically restored within a 20/20 (normal) to 20/40 (good) range. Those are excellent results, especially given how far treatment has progressed in such a short time.

Modern cataract surgery was first performed in the late 1960s, enabled by the development of an ultrasound technology that emulsified the eye's diseased natural lens, along with the discovery of a suitable replacement-lens biomaterial, polymethylmethacrylate (PMMA). Since the first prosthetic intraocular lens (IOL) was rigid, however, the incision required to insert it into the eye was large (encircling roughly half the cornea), required sutures, and made recovery long and outcomes variable.

When deformable materials, such as hydrophobic acrylic and silicone, replaced PMMA in the early 1990s, incision size decreased dramatically due to the new materials' ability to be rolled, folded, and bent during insertion. As IOL materials evolved, the insertion process did too, shifting from forceps to a tapering tube (similar to a syringe) that pushed the lens into the eye. Lenses are now being delivered through increasingly smaller incisions ranging from 1.8 to 2.8 mm.



Geometry of a typical replacement lens (IOL) used in cataract surgery. Standard dimensions for the acrylic lens are 6 mm diameter with a 1 mm center thickness. The lens needs to be compressed inside an inserter tip that delivers it into the eye through a corneal incision that averages 2.8 mm.

With the size of the incision directly related to post-surgical aberrations in vision, engineers at Bausch + Lomb in Rochester, NY continue to look for improvements and have recently set an ambitious 1 mm incision goal. To achieve this, ongoing research and development is focused on new lens materials, improved IOL geometry, and better inserter designs. That's where finite element analysis (FEA), with its capability to realistically simulate a wide variety of physical phenomena, enters the picture.

Simulation sees what can't be measured

Engineers at Bausch + Lomb have been using Abaqus FEA, in biomedical applications for about ten years. It was first employed to model the conformation and deformation of contact lenses on the cornea; this helped evaluate lens performance, including optical properties. Other applications have included improving cataract surgery tools and modeling manufacturing procedures.

"We use FEA in our iterative design process to shorten development time by analyzing each design or by developing design rules-of-thumb," explains Robert Stupplebeen, design engineer and analyst at Bausch + Lomb.

In general, to create its FEA models, the Bausch + Lomb team first builds 3D CAD models in SolidWorks and then uses the software's Associative Interface to import the model into Abaqus. From there, simulations are often coupled with other programs, such as SigFit, an optomechanical pre- and post-processor (developed

by Abaqus Integration Program member Sigmadyne), and ZEMAX, a comprehensive optical design software package.

When starting a new product design project, "getting sufficient biological test data can be problematic," says Stupplebeen. "With just about any biomedical product or process development, there are a lot of assumptions that need to be made."

In the case of cataract surgery, the Bausch + Lomb product development team is focused on two primary modeling issues that can be confirmed by testing: the insertion force required to implant the IOL, and the geometry of the lens as it emerges from the inserter. But they also are interested in what can't be measured in real life, such as the geometry and internal stresses of the lens when it's in the inserter.

"We validate our model on the things we do know and then utilize the rest of what the model tells us to gain a better understanding of the physical behavior," says Stupplebeen. "Without FEA, all of these things are just unknowns."

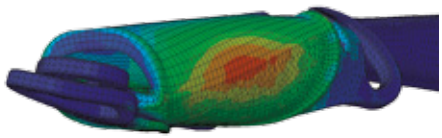
The cataract surgery simulation setup requirements are stringent, says Stupplebeen. "The analysis is highly non-linear with large deformations, difficult self-contact, sliding contact, and hyper-elastic material properties. To handle all this in one model, we chose Abaqus/Explicit."

The model: lens, inserter, incision

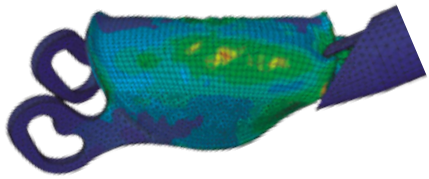
From an ophthalmologist's point of view, the cataract surgical procedure is relatively simple: Take a standard IOL, which consists of a circular lens with four appendages (haptics) that stabilize the lens in the eye; load the lens in the inserter and fill the inserter tube with a viscoelastic lubricant; make a small corneal incision and remove the damaged crystalline lens using an ultrasonic device; then place the inserter in the incision and push the plunger, inserting the IOL. The surgery is outpatient and typically takes under 10 minutes.

From an engineering perspective, however, the procedure is quite challenging given the geometry: an industry-standard precision lens has a 6 mm diameter, a center thickness of 1 mm, and four haptics; an average incision is 2.8 mm.

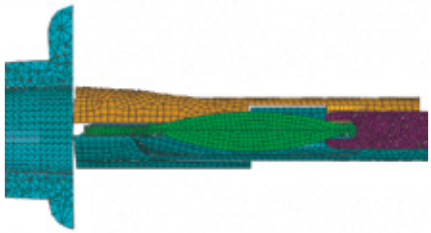
"It's like trying to suck a Frisbee through a vacuum," says Stupplebeen. "During the insertion, the lens can experience strains in excess of 60 percent."



Lens strain is illustrated as the IOL is being pushed by the plunger inside the inserter during a cataract surgery simulation. Rare lens tear has been observed to occur at points of stress where the plunger contacts the IOL or on the trailing haptics.



Strain on the lens is shown as the IOL emerges from the tip of the inserter. The areas of highest stress correlated with the location of rare lens scratching. Values represent 0-60% strain.



Section view of the FEA model used to simulate lens (IOL) insertion during cataract surgery. The IOL (green) is being pushed by the plunger (purple) and is about to enter the inserter tip (blue) through which it is delivered into the eye. The lens, tip, and plunger are modeled using deformable elements; the loading door (brown) and loading area (dark green) are modeled using rigid elements.

To simulate the lens insertion process, the Bausch + Lomb team modeled an acrylic lens, with average lens and inserter dimensions, and applied the following parameters to the model: hyper-elastic Neo Hooke material properties; Rayleigh damping to reduce low frequency oscillations; general contact with zero friction (because of the smooth surfaces and lubricant); a nonlinear pressure-overclosure relationship to reduce contact penetration; and mass scaling to reduce solve time by a factor of 10.

The loading area of the inserter was treated as rigid and modeled using R3D4 elements, while the lens, plunger, and tip were all treated as deformable using C3D8R and C3D4 elements. The model of the lens and inserter, which are designed in tandem because of their close interrelatedness, is highly complex with approximately a quarter of a million elements with over 100K increments. To run the five-hour analysis, the team used a Cray CX1 with Windows HP Server 2008.

Validating lens strain and inserter forces

By using Abaqus, the team was able to calculate the force applied on the inserter versus the displacement experienced by the lens and then compare it with test data. The analysis yielded results that correlated well with the tests. The team was also able to measure strain on the lens while visualizing its deformation as it traveled through the inserter. These peak strain measurements correlated well with extremely rare failure modes (tip fracture, lens tear, and lens scratches) and were also found to occur in the same locations where past real-world failures had occurred.

"Given the agreement between simulation results, physical tests, and observations, the validated model is being used to reduce the likelihood of failure modes, reduce insertion force, and develop the next generation of IOLs and inserters," says Stupplebeen. "As surgeons strive for smaller incisions, we have to develop a more compressible material, thinner lenses, and/or new inserter geometries."

FEA benefits are clearly visible

Whatever the product development direction (and Bausch + Lomb is pursuing them all), Abaqus FEA is helping their designers and engineers make predictions about what will work and what will not. Since cataract surgery product design cycle time is typically about a year and a half, with an additional year for clinical trials, accelerating prototyping with realistic simulation provides tremendous bottom-line benefit.

"We recognize the significant return gained from continued investments in simulation," says Stupplebeen. "Without a doubt, it has helped us shorten our time to market, decrease our development costs, and improve our product performance."

For More Information
www.bausch.com
www.simulia.com/cust_ref

No Two Hips are Alike

Mobelife leverages realistic simulation for patient-specific design and analysis of hip revision implants



Of the 6.2 million people in the U.S. treated annually for bone fracture 220,000 of these patients receive a total hip replacement. Typical lifespan of a hip prosthesis is between 15 to 20 years, which means more and more patients are outliving the implant. When a prosthesis fails, a revision surgery is carried out to replace the components.

A significant challenge faced during implant revision surgery is that the pelvic bone stock is often significantly reduced. Ten-year failure rates of primary hip replacements are estimated at 11.4%. This number more than doubles to 25.6% in case of revisions. In over 58% of those revision failures the acetabular component, the cup-shaped cavity in the hipbone into which the ball-shaped head of the femur fits to form a ball-and-socket joint, is involved.

To deal with today's challenges, such as massive bone loss and multiple revisions of the hip, a custom approach is most suitable.

One-of-a-kind joints: a personalized solution

To assist in improving patient outcome, Mobelife®, a Belgian high-tech company, has developed a completely customized approach based on Computed Tomography (CT) data by combining state-of-the-art image processing tools (from Materialise) and Abaqus FEA. The Acetabular (hip-joint) implant design process by Mobelife allows for personalized restoration in terms of anatomy, stability and mobility in most devastated pelvic bones.

Mobelife has unique experience in offering not only the software aspects of converting medical images to geometry to use in simulations, but also delivering the parts that the surgeons use under such extreme conditions.



An example of a patient-specific acetabular revision implant, showing the spherical cup with three fixation flanges, and part of the porous structure at the back side.

The process begins with the creation of a 3D model of the patient's pelvic bone based on CT images. Mobelife employs Mimics medical imaging software from Materialise, enabling users to quickly generate surface meshes from the CT images. Based on the CT-data, the bone is reconstructed automatically and the implant is designed in close collaboration with the surgeon for unique fit and functionality.

The precise orientation of the newly created hip socket is anatomically analyzed. In cases of significant pelvic bone loss, the implant extends onto the major bones of the pelvis for fixation. A custom porous structure is used to fill the gap and a thin porous layer of titanium interfaces between the implant and the bone. Flexibility and compatibility are provided for either press-fit or cemented-liner integration.

Before the surgery even starts, the optimal screw positions and lengths are determined based on variable bone quality. Once the design phase is finalized, the implant is patient-specifically analyzed with Abaqus FEA for mechanical integrity and interaction with the surrounding bone based upon fully individualized muscle modeling and finite element simulation.

Abaqus provides the personal touch

Mobelife has developed a dedicated and automated preprocessor to link their 3D models with the Abaqus FEA software. Patient data is analyzed for the location of muscle attachment regions on the bony structures and the interconnection trajectories of the muscles. The outcome is translated into specific forces acting on the pelvic bone and the implant based on patient weight and muscle activation.

The thickness of the bone's compact cortical shell and the properties of the spongy, trabecular bone beneath are automatically calculated from the CT-data and imposed on the local elements of the bone model. Finally, material properties of the titanium implant components are assigned. Once the part is assembled and contact has been defined, the stresses, strains and displacement of the patient-specific model in relation to the bone are analyzed. This process helps to investigate mechanical integrity of the design and avoid bone resorption (stress shielding).

Simulation technology that touches human lives

So how does this ability to design and create individualized hip replacements impact the patients themselves? In the case of one woman, it meant the difference between possible immobility or pain-free walking. The 50-year-old patient was diagnosed with a pseudotumor after Resurfacing

Arthroplasty for osteoarthritis of the left hip joint. The revision failed after one year and she developed a pelvic discontinuity (a distinct form of bone loss separating the pelvis). Using conventional methods, the extreme bone loss would make it even more difficult to replace the prosthesis. The steps below outline how Mobelife relied on realistic simulation to help repair the damage.

Step 1: Imaging

Advanced 3D-image processing presented the bony structures and implant components. Analysis showed the extent of the pelvic bone loss. The former implant migrated back and off center, dislocating the joint. The automatically generated reconstruction proposal showed the missing bone stock and the anatomically correct joint location.

Step 2: Custom implant proposal

In the second step, a custom acetabular metal backing implant was proposed. The bone defect (35ml) was filled with a patient-specific porous structure rigidly connected to a solid patient-specific plate. The proposed implant shape was determined taking into account the surgical window and surrounding soft tissues. Cup orientation was anatomically analyzed. Screw positions and lengths were pre-operatively planned depending on bone quality. This information was transferred into the actual surgical procedure using custom jig-guiding technology from Materialise.

Step 3: Design analysis

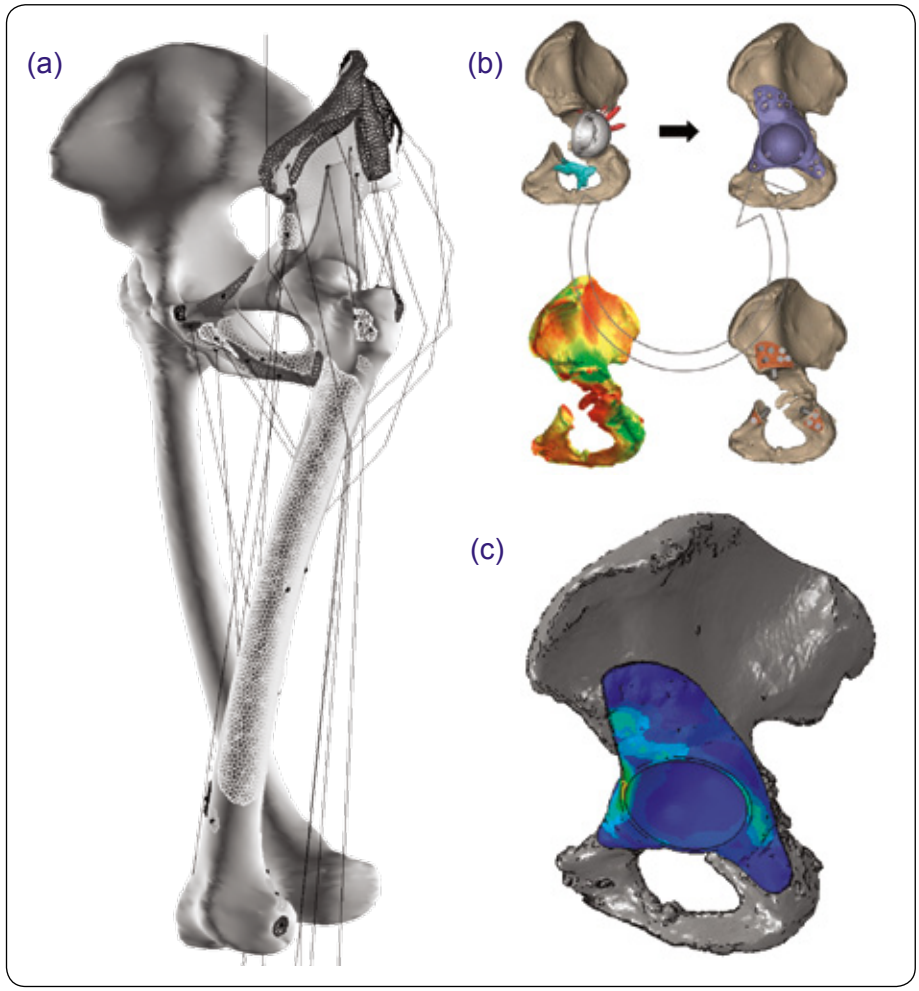
In the third step, the implant design was virtually tested with Abaqus to see how it would perform for this specific patient. Implant integrity proved to be adequate as the bone loading did not exceed the safe range.

Step 4: Production

Mobelife produced the implant parts and jig with Additive Manufacturing techniques under ISO 13485 certification, using respectively the Selective Laser Melting (SLM) technique in medical grade Ti6Al4V material, and the Selective Laser Sintering (SLS) technique using medical grade epoxy monomer. The parts were cleaned ultrasonically, optically scanned for quality control and sterilized in the hospital.

Step 5: Surgery-ready solution

The complete solution package included the implant, individualized instructions and the jig for pre-drilling of screw holes into the bone. During surgery, the old implant was removed and the patient-specific implant inserted easily. All screws were applied according to the plan. The liner was integrated and the joint reduced. Functionality and mobility of the hip joint were tested during the operation with positive results.



(a) Muscle model comprising muscle attachments and trajectories. (b) Fifty year old female patient with pelvic dislocation. A large pelvic dissociation (top left) is reconstructed by a patient-specific implant (top right), with a porous filler and thin porous layers as well as optimally positioned screws (bottom right) based on the particular bone quality (bottom left). (c) Von Mises stresses in the patient-specific revision implant analyzed. Stresses do not exceed the implant's material safety range.

Step 6: Recovery

A few days after surgery, the patient was able to carefully take her first steps completely pain free. She is recovering extremely well, exceeding usual primary revalidation patterns.

About Mobelife

Mobelife was founded in October 2008 as a Belgian high-tech company with the purpose of serving the health care professional directly – and hereby the patient indirectly – by a completely customized product development process. It offers innovative all-in-one patient-specific orthopedic implant solutions which are individually evaluated for optimal fit, stability and mobility. The company intends to sensitize surgeons of the need for patient-specific solutions to improve the patient's quality-of-life after complex reconstruction surgery.

Mobelife. custom orthopaedics

For More Information
www.mobelife.be
info@mobelife.be

Realistic Human Simulation—A new Frontier in Biomedical Innovation

Subham Sett, Life Sciences Lead, SIMULIA Technical Marketing



Image courtesy of Simpleware

In a 2010 survey conducted by SIMULIA, nearly 500 customers from industries such as Life Sciences, Consumer, and Consumer Packaged Goods provided feedback on their current usage and future use of realistic human simulation (RHS) as part of their product development process. Their positive response indicated a strong interest in applying Abaqus for RHS.

In a previous strategy article titled *Rx for Medical Innovation* in October of 2008, I discussed the challenges faced by medical device customers and briefly touched upon the need to include tissue-device interactions. Today, with an increased interest in leveraging Abaqus for RHS, I would like to share a more in-depth look at this requirement including the current role of FEA in this field, the contribution of the research community in advancing our understanding of human modeling, and SIMULIA's strategy to address the challenges and requirements of making realistic human simulation an integral part of the realistic simulation product development process.

Complexity of human tissue modeling

If you think metals, alloys and plastics have complex properties and that products designed using them are innately nonlinear, take a step back and consider human tissue behavior. It is inherently nonlinear and hence difficult to define and analyze. However, the traditional strengths of Abaqus for modeling highly nonlinear behavior including complex materials, severe deformation, and contact make it uniquely suited to modeling and simulating the realistic behavior of human tissue.

We recently added support for anisotropic hyperelasticity to account for the fundamental nature of tissue—different fiber orientations and different responses to mechanical loads. Tissue behavior is characterized not just by a mechanical response to a stimulus but also by other physics—heat transfer, transport, reaction-diffusion, electrical signals and many more. Most of these physics can be captured in a single analysis model using built-in Abaqus features, while some of the more advanced behavior, such as muscle activation, require user subroutines. The Radiofrequency ablation (RFA) of tissues—which is used to selectively destroy tumors without damaging adjacent healthy tissue—is one example where multiphysics comes into play. The application requires the study of phenomena such as tissue damage, blood perfusion and cooling effects in conjunction with resistive heat effects. All of these effects can now be modeled through a co-simulation approach that combines a thermal-electrical analysis using the new Abaqus/CFD capability with simple user subroutines to account for blood perfusion and tissue damage.

Modeling and analyzing human systems

The general-purpose capabilities in Abaqus provide a significant advantage to anyone needing to perform realistic human simulation. The same set of tools and analysis functionality can realistically simulate the complete range of human anatomy including cells, organs, muscles, blood vessels, bones, and joints as well as full-body.

For example, researchers in National University of Singapore[1] have used

Abaqus to model and analyze the structural integrity of malaria-infected red blood cells. At Boston University[2], researches are evaluating how the human brain behaves during an EEG. At UCLA[3], the same underlying Abaqus technology, in conjunction with third-party tools, is being used by their researchers to build a hierarchical multi-scale approach to modeling the human femur.

Modeling human systems would be incomplete without the ability to look at the drivers of human systems—the musculo-skeletal system. While Abaqus traditionally has been known for its ability to look at fully deformable systems, the biomechanics community is quickly catching on to its ability to provide a single modeling paradigm that can easily evolve—from match-stick representation of the human system for kinematic modeling of activities such as gait, jumping or sitting to modeling complex, deformable systems including contact—all in the same modeling and analysis framework. In addition to kinematic modeling, more sophisticated behavior incorporating muscle actuation and/or muscle set optimization can be performed using logical-physical modeling capabilities in Abaqus. We also realize that the life sciences research community needs the ability to link their Abaqus analyses to specialized musculo-skeletal numerical tools. At SIMULIA we have a strong alliance program and are partnering with industry leaders such as AnyBody Technologies to enable researchers to perform such coupled analyses.

Building a realistic anatomical model

Realistic human simulation relies on anatomical data. In the traditional product development process, design engineers have access to CAD, either 2D or 3D drawings, as well as industry-standard material databases. Engineers attempting to perform human modeling often need access to biological data from cadaver testing, CT-scans or MRIs. Medical imaging data, often provided as gray-scale images, must be assessed and differentiated to segment out different tissue types and then converted to a 3D representation. Depending on the tissue type, this is a challenging task and requires special image processing software. Two SIMULIA partners, Materialise and Simpleware (see article on page 19), provide medical imaging software capable of creating a finite element or CFD mesh representation of the human body that can readily be imported into Abaqus for completing the model definition and performing highly accurate simulations.

Calibrating tissue material

As with any engineering analysis, a calibration of the tissue model is critical. Traditional material calibration techniques and parameter fitting via optimization techniques can be performed in a time-efficient manner for in-vitro test data. For in-vivo data obtained either through force-plate measurement or fluoroscopy techniques, one has to resort to more involved inverse FEA methods. Such methods frequently rely on integrating test data with Abaqus and other computing tools such as Matlab, Excel or even simple calculators. By using Isight—SIMULIA's solution for process automation, workflow integration, and design optimization—researchers have the ability to easily capture material calibration steps in a single workflow without needing to be a scripting guru. Optimization techniques within Isight can then be applied to the workflows to run hundreds of analyses, efficiently using cluster-computing resources to obtain the best material fitting parameters.

Commitment to research is critical

The use of Abaqus is well-established within the production environments of classical engineering and manufacturing industries such as Aerospace and Automotive. However, to achieve realistic human simulation, research plays a critical role in our ability to model, analyze, and understand tissue behavior. Biomedical researchers at the world's leading universities are collaborating closely with professionals in medical facilities and medical device manufacturers to gain a clinical appreciation of the requirements and translate their research into engineering terms.

An extensive network of research professionals across the globe is relying on Abaqus FEA and Isight to expand the knowledge-base of human tissue mechanics. The research community adds a tremendous value to this area and SIMULIA is actively supporting this process through travel grants to major biomedical research conferences, internships and a dedicated bioengineering program to improve access to Abaqus for research, teaching and student educational purposes. We are also encouraging mutually beneficial collaborations. Very recently, we introduced the Extended Finite Element or XFEM method to enable model fracture without knowing a priori crack path. We see this as a valuable tool in the prediction of risks to bone fracture whether due to osteoporosis, injury, or resulting after a surgical implant procedure. However, the damage criteria that apply to bones are still not well known. We are now supporting researchers with the right software tools so they can focus on applied research to make computational tools applicable to realistic human simulation.

We are committed to providing our customers with the best-in-class unified FEA, multiphysics, process automation, and optimization solutions to advance their use of realistic human simulation. We are accelerating our collaboration with current and new partners to improve the integration of best-in-class solutions and streamline the development of robust biomedical simulation workflows. By actively engaging with our customers both at the commercial and research level, we will make rapid advancements in our realistic human simulation technology and accelerate the development of innovative research methods, treatments and medical devices that improve patient care and the quality of life. We look forward to hearing your ideas, thoughts and feedback on how best we can serve the needs of this emerging field.



Subham Sett
Life Sciences Lead,
SIMULIA

Subham is responsible for developing our Life Sciences simulation strategy and roadmap and in this role works with academia, industry, and regulatory agencies. He has 10 years of engineering simulation experience including methods development for the medical device industry. He joined SIMULIA in 2003 from Coventor, Inc. where he was a MEMS product development engineer and earned several patents. Subham holds a M.S. from the University of Colorado, Boulder, and a B. Tech. from the Indian Institute of Technology, Kharagpur.

Download Life Sciences-related customer papers at: www.simulia.com/cust_ref

Smith & Nephew Put New Knees through Their Paces with Realistic Simulation

Researchers study performance of replacement joints with Abaqus FEA

The largest joint in the body, the knee, bears five times our body weight with each step we take. Even without suffering a sports injury or serious accident, many people will experience that time alone can bring lifestyle-changing wear and tear to the anatomical structures of their knees: aging can cause severe arthritis producing significant pain and greatly limited mobility. In that case, total knee replacement—performed about 580,000 times a year in the U.S. alone—is currently the solution that provides the most relief, as evidenced in the medical literature.

The American Academy of Orthopaedic Surgeons calls knee replacement "one of the most important surgical advancements of the 20th Century." The technology has continued to evolve since the first artificial knee was implanted in 1968. The current most-used procedure, Total Knee Arthroplasty (TKA), replaces damaged or diseased joint surfaces of the knee with metal and plastic components shaped to mimic the function of the original articulation. Sized and shaped to fit, knee implants have been shown, in so-called patient registries, to perform well for at least 15-20 years in more than 95 percent of patients, most of whom can achieve a range of motion of from zero to about 120 degrees.

But since the physiological range of motion of a 'normal' knee is a wider zero to 135 degrees, some TKA patients find they can't return to their previous levels of full functionality or activity. Some people's bones show an atypical response to implantation of the metal, even though it is biocompatible. And as lifespans get longer, the durability of implants becomes increasingly more important. To drive research and innovation and achieve a greater understanding of knee kinematics, improved mobility and device robustness for knee patients, Smith & Nephew, the U.K.'s largest medical technology company, founded the European Centre for Knee Research in Leuven, Belgium in 2007.



Post-surgery patient undergoing videofluoroscopic analysis of the function of their replacement knee. Image courtesy of Dr. Sergio Romagnoli.

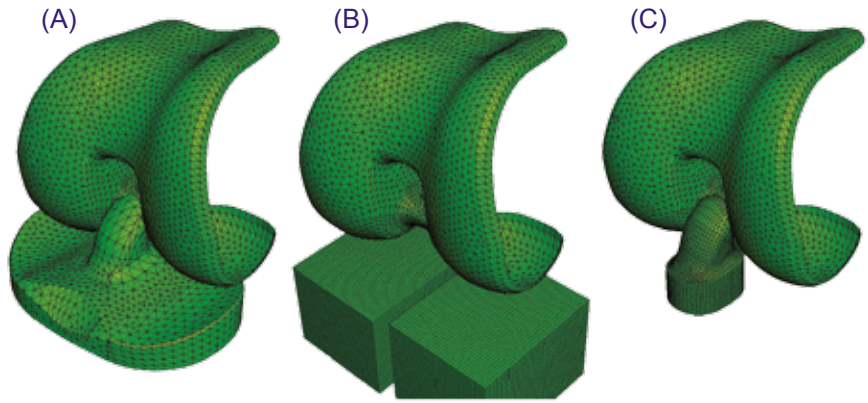
150-year-old Smith & Nephew is an industry leader in orthopaedic reconstruction and trauma, and operates a number of R&D centers around the world. But "the Knee Centre is unique because it's focused solely on research," says Bernardo Innocenti, M.E., Ph.D., the Centre's project manager for Numerical Kinematics. "We submit all projects to a scientific advisory board, in which several high-level orthopaedic researchers are involved. This advisory board supervises our protocols to ensure that all research is done in the most scientific and ethical way. Our results are important not only from the researcher's point of view but from the designer's point of view as well. We provide great research tools for surgeons, scientists and companies."

Dr. Innocenti is a perfect fit for the job: he's felt pulled to research since childhood. "When I was really young I wanted to be a doctor, but I don't like blood," he explains. "Instead, since I've always been good at mathematics, I became interested in numerical modeling and how it can be applied to medical issues."

A typical knee model? No such thing.

A successful research project by Innocenti and his colleagues won the Knee Society's Mark Coventry Award for the best Basic Science Paper in 2009. Their study of the kinematics of an in vivo replacement knee used a novel combination of videofluoroscopy (a type of radiography, which, unlike a static X-ray, gives a real-time look at bones inside a moving leg), and numerical modeling with finite element analysis (FEA), to look at contact position in patients who had undergone a full knee replacement. Another study used FEA for realistic simulation of leg bone resorption occurring where the tibia comes in contact with a metal implant. Still other areas of research have included comparing different geometries of implant models and their effects on gait and knee kinematics.

"We work very closely with surgeons who come to us if they find a particular issue with a patient that they want to solve, or they see something out of the ordinary in their clinical practice and are looking for an explanation," says Innocenti. "What I like best about my



Abaqus FEA models of Smith & Nephew replacement knee components used for evaluation of the contacts between the different parts. The full model (A) with the original femoral and tibial components used for the sensitivity analysis was then modified to (B) one for the condylar contact points and (C) another for the post-cam contacts.

work is that there's really no such thing as a "typical" model—every project is different and exciting."

Abaqus FEA helps go inside the knee

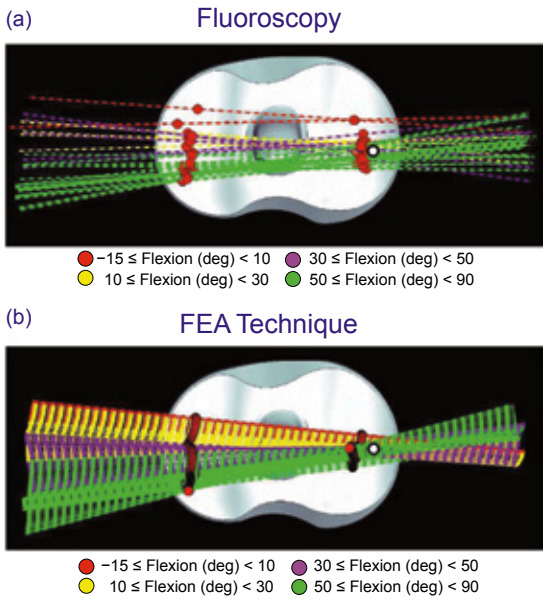
What all these Knee Centre studies do have in common is the use of Abaqus Unified FEA; engineers at Smith & Nephew have used the software for product design and development for years. "Abaqus FEA is fundamental in this game because it enables us to estimate rapidly and precisely the effects of different parameters in the design or performance of a TKA," says Innocenti. "When I joined the Knee Centre I had not used Abaqus before, but I found it very easy to work with. Modeling is very straightforward, yet it adapts to whatever complexities I want to introduce and design changes are easy to execute."

"When you replace a knee, you are trying to replicate the behavior of biological materials, like bones, cartilage and ligaments, with non-biological ones such as titanium, stainless steel and polyethylene," he says. "I have everything I need for simulating the performance of all these materials in Abaqus, whether it is bone or metal or something more complicated like the viscoelasticity of soft tissues or polyethylene."

A notable problem with modeling the artificial knee is that its mechanics vary greatly over time because, as the joint moves, the loads and stresses on the contact points keep changing over the entire range of motion. And every replacement knee is operating in a unique body environment. Videofluoroscopy of a TKA patient's leg in motion is an accepted technique for monitoring this functionality. But videofluoroscopy only shows the behavior of the leg bones and metal inserts, not the soft tissues—or, most critically, the polyethylene insert that cushions the contact between the upper and lower parts (the femoral and tibial components) of the prosthesis. This is the challenge that Innocenti and his colleague Luc Labey, M.E., Ph.D., overcame with their award-winning research.

Visualizing the challenge with FEA

Their study examined five osteoarthritis patients who had each received Smith & Nephew's Journey® Bi-Cruciate Stabilized Knee System, a guided motion knee implant specifically designed to produce more "natural" kinematics after TKA. The patients performed a number of exercises while being analyzed with fluoroscopy—rising-sitting, stair climbing and step up-down—and the resulting kinematics data was used as input for Abaqus FEA models of the knee implants.



Replacement knee component contact points location (red dots) and contact line rotation (in different colors according to the corresponding flexion angle) are compared using traditional fluoroscopy (A) and the Abaqus FEA model with fluoroscopy based kinematics input (B) from a typical patient during chair rising-sitting. The results support the basic reliability of fluoroscopy, but also demonstrate the importance of using FEA models for a more realistic estimation of the contacts and for deeper understanding of the loads and stresses that occur during in-vivo post-cam engagement.

Putting the FEA models through the same movements as the patients allowed Innocenti's team to estimate, very accurately, the contact points between the femoral and tibial components, taking into account the modulating effects of the "invisible" polyethylene part that's undetectable with videofluoroscopy. The FEA analysis supported previous contact point displacement measurements derived from fluoroscopy alone, but with smoother, more credible and consistent patterns—demonstrating that the Journey BCS patients' new knees were working as intended.

In addition, for the first time, the models enabled the invivo analysis of the contact between the femoral cam and the tibial post. "We were able to validate our technique with experimental results that produced a very high quality metric," says team testing head Luc Labey. "Our findings can be incorporated into both future design refinements and recommendations we make to surgeons today."

Validation of their Abaqus models has given Innocenti's team confidence to extrapolate their data to a wider range of questions about TKA longevity. "How these materials behave over time is critical to our work because an understanding of wear is very important with prostheses," says Innocenti. Physical prototypes of artificial knees have historically been subject to laboratory wear testing in the same way as many other manmade products: repeated motion in a test rig over time. But since it takes over one million cycles of knee "steps" to replicate the wear and tear of a single year of walking, it takes many months to collect enough real-world data to be useful. However, by "walking" their Abaqus virtual knee prostheses through accelerated test cycles, the Smith & Nephew team has been able to simulate the effects of five years of walking in just one week.

Future research with realistic simulation

Innocenti sees great potential for Abaqus in future research as well: "To try to be able to model more specifically and accurately all the biological systems around the knee bones, the soft tissues, the menisci, is a major goal," he says. "FEA could be a fundamental piece in this refinement due to its intrinsic ability to provide rapid output and sensitivity studies."

The ultimate aim of total knee replacement is to have a prosthesis that behaves as naturally as possible, he points out. "Abaqus is helping us get ever closer to designs that let TKA patients do everything they want to for a full, active life after surgery."

For More Information
global.smith-nephew.com

Sumitomo Calls on SIMULIA for Answers to Cell Phone Cable Design Challenges

Realistic simulation with Abaqus FEA provides solutions down to the wire

So many people feel they can't live without their cellular telephones these days, but how do cell phones survive all that use – and abuse – from people? These days the devices have to provide ever-greater functionality fitted into an increasingly smaller package. One solution has been to add a flip/twist or sliding panel that opens up to provide more surface area. Yet the laws of physics are constant: electricity can only travel along an unbroken path. This means that the wires keeping everything inside a cell phone running smoothly have to stay intact no matter what.

The solution? Coaxial cables. Made up of hundreds of bundled fine wires, and of less than 0.3 mm in diameter, they are highly compact and particularly well suited to use inside cell phones because a protective layer around each signal wire shields it from electromagnetic noise. The individual wires within a coaxial cable, each measuring just dozens of microns in diameter, stand up well against all the bending and twisting of a typical cell phone panel – at least for a while.

Although coaxial cables are made to withstand tens of thousands of opening/closing cycles, the recent trend towards even thinner products has limited cable routing options, and increased the loads on the cables themselves. So, as specialists in electronic wire products, engineers at Sumitomo Electric Industries saw a further need to enhance cell phone cable durability and decrease wire breakage from fatigue.

"Cell phone product design must include an evaluation of the lifetime of the phone's cables," says Shigeki Shimada, a member of the team at the Analysis Technology Research Center at Sumitomo. "Since the optimum layout of the cables ensures that their lifetime exceeds that of the product, we need this information at the earliest design stages."

In the past, the Sumitomo team evaluated cable lifetime using fatigue test equipment that physically stressed the wires over long periods of time. But the highly competitive electronics industry is demanding ever-shorter design cycles, while advances in cell phone durability are producing longer-lasting, more complex phones whose cables must stay functional for the duration.

Computer-aided engineering solutions from SIMULIA, the Dassault Systèmes brand for realistic simulation, provided Sumitomo with a strong response to these pressures: Using Abaqus Unified Finite Element Analysis (FEA) software, the group created computer models to simulate the behavior of everything from a single wire to a full bundle of coaxial cables. The models provided an inside view of many of the stresses and strains arising in the dynamic environment of a cell phone body. The results of their analyses provided engineers with a cause-and-effect vocabulary with which to examine design and material modifications resulting in improved performance.

The Sumitomo research group started with FEA models of a single cylinder-shaped wire rod, similar to an electrical cable (Figure 2). Simulating the response of the model to deformation caused by extension, bending and twisting (using Abaqus/Explicit for a quasi-static analysis), they measured the number of virtual wire breakages. These results helped them quantify the relationship between strain amplitude and lifetime.

Moving up to more complex FEA models of a coaxial cable with detailed internal structures (Figure 3), the Sumitomo engineers researched the relationship between the wires and the insulating layer under different degrees of strain and deformation. They used elastic material

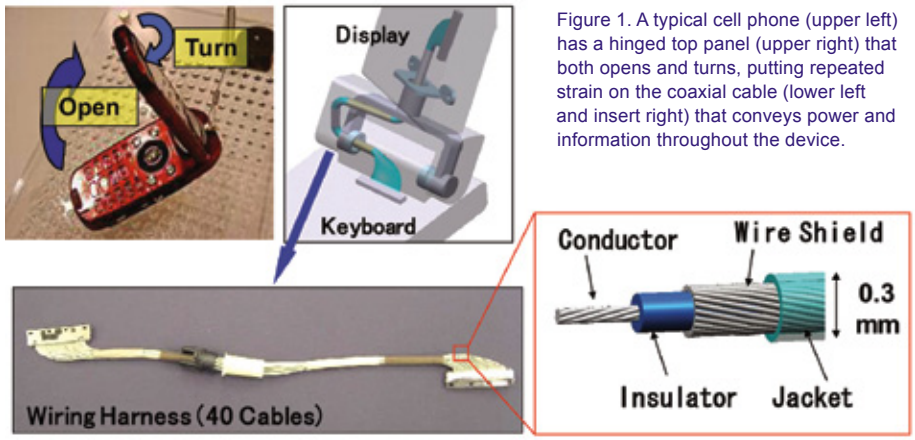


Figure 1. A typical cell phone (upper left) has a hinged top panel (upper right) that both opens and turns, putting repeated strain on the coaxial cable (lower left and insert right) that conveys power and information throughout the device.

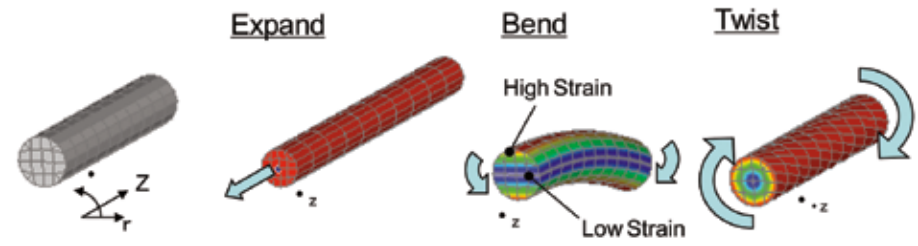


Figure 2. Deformation modes and main states of stress in a wire rod in Abaqus FEA.

models for the copper alloy of the wires composing the conductor and shield, and the Teflon resin of the insulating layers; the more than 100,000-cycle lifetime needed for the wires required a small strain (below 1%), meaning plasticity could be neglected. Combining these results with the data from the first models imparted deeper insight into the lifetime of an entire cable under fatigue tests that included left/right bending and other deformations. Real-world fatigue testing plus measurements of actual cable behavior validated the FEA predictions.

Finally, in order to look at the behavior of coaxial cables in the context of their actual layout inside a cell phone, the group modeled the cable and its insulation jacket as a macro beam with specific tensile, bending and torsional stiffness, and calculated the deformation at specific points in time at each area during the simulated movement of the cable. They next modeled the wall and cable bundle in the vicinity of the hinge – where the majority of problems occur in actual cable layouts of cell phones.

Simulating the behavior of the cable during opening/closing cycles that included a 180-degree rotation, they compared their Abaqus FEA results with X-ray images of actual cable cycles under fatigue testing (Figure 4). "We can now make qualitative evaluations of the lifetime of an actual layout of electrical cables," said Ken Manabe, another member of the research team. "In the future we plan to work on modeling kink-initiated breakages as well as cable breaks caused by wear."

With the lifetime evaluation methodology they developed using Abaqus Unified FEA, the engineers could then turn to even more complex design challenges, such as the effects of cable layout on lifetime in the newest thin, slide-type cell phones. They discovered that the sliding action of the phone caused the cables to meander both sideways and axially, extremely shortening lifetime. The design solution was a protective tape sheath that kept loose cables aligned together (Figure 5), but did not restrain their flexibility as the phone slide moved.

"Our CAE expertise is now helping us develop applications for similar types of cables, such as those used in robots," says Shimada. "As demands for cable performance become more diverse and rigorous, realistic simulation will help us develop responses to the various design challenges that arise."

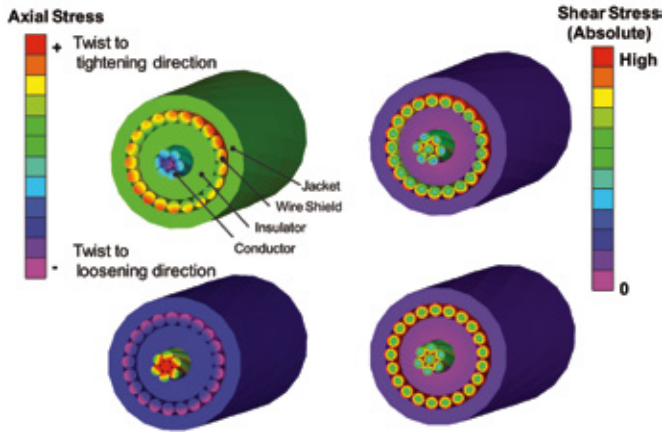


Figure 3. Distribution of axial and shear stress in full cable model under twisting (18 degrees/mm).

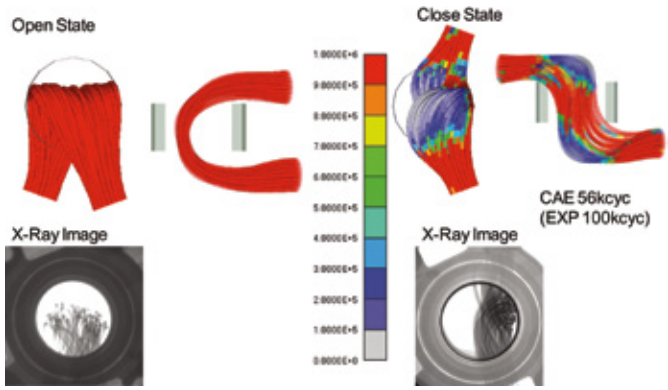


Figure 4. Results of lifetime evaluation and X-ray image of opening/closing cycle.

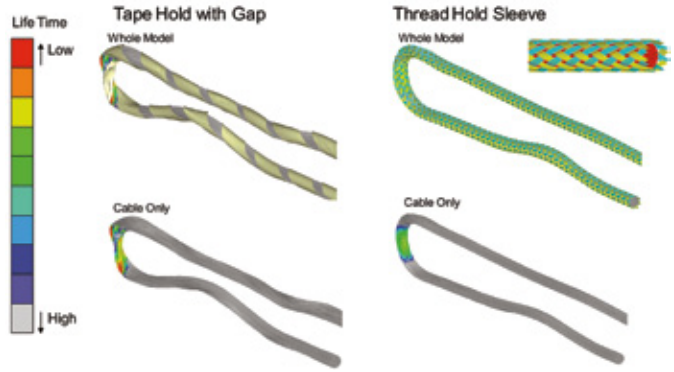


Figure 5. Design evolution of the cell-phone slide motion harness: Previous iterations of protective tape used an overlapping tape hold (not shown) that stacked the co-axial cables, increasing strain and shortening lifespan. The first improvement (left, top and bottom) was to change the way the tape was wound, opening a gap to release the cables. But cables could flow out the gap so lifespan did not increase much. Next, thread hold protection was tested (right, top and bottom). The friction of the threads is low enough not to stack the cables and the gap is too small for the cables to flow out. CAE and experimental testing confirmed that a long cable lifetime resulted from this design.

For More Information
global-sei.com

Rigorous Analysis, Flexible Design Lead to Creation of Optimal Mudmats

Customized automated simulation tool, integrated with Abaqus FEA, improves efficiency and quality of plate mudmat design for offshore production system



Most people are familiar with images of the upper part of an oil or gas platform visible above the sea water— but there's more complexity at work backstage, beneath the surface. In addition to the pipelines and risers, a mechanical support team— pipeline end termination and/or manifolds, often referred to as PLETs and PLEMs—is needed. And if the sea bottom is too soft to bear the load, which is often the case, the support team may itself need additional support. This is where mudmats come in.

The commonly used plate mudmats, made with carbon steel, consist of a top plate and a number of perpendicular vertical stiffeners that function as load-bearing beams (See Figure 1). But the simplicity of the design belies their importance.

"Mudmats are just as vital as the subsea equipment resting on them," says Dr. Jinming Xu, Engineer at Foster Wheeler Upstream (Houston, Texas). "They must be designed to have sufficient strength, particularly buckling strength, and that requires considerable design analysis."

Since most mudmats are custom-designed for individual load performances, subsea soil properties, and installation requirements, nearly every mudmat requires its own unique stress and buckling analysis. The analysis process used to be largely manual, time-consuming, and unwieldy—until engineers (Xu et al) at Foster Wheeler created their own software, Mudmat*ician*, for plate mudmat design, and integrated it tightly with Abaqus Unified Finite Element Analysis (FEA). The result was an automated process that cut analysis time from weeks to days and significantly reduced the potential for human error in hand calculation.

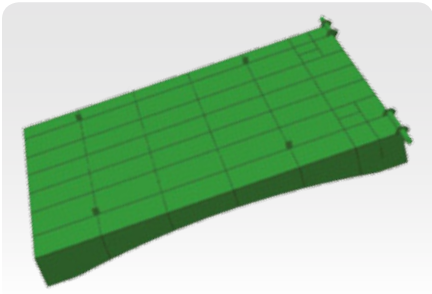


Figure 1. Mudmat*ician*-generated model of mudmat (non-plate accessories added manually), showing top view.

Number crunching during crunch time

Under the old method, designers first obtained panel stresses (checking von Mises stress against the yield stress) directly from an Abaqus FE analysis. Then they had to perform buckling checks (determining panel strength against sudden failure due to buckling) by hand, based on the stresses derived from the analysis at the first step and using industry codes such as API Bulletin 2V to guide the buckling calculation and verification.

To check a single panel, they would extract normal and shear stresses at nine different locations (See Figure 2) based on the panel aspect ratio (length to width). These 27 stresses were then plugged into a long list of complex if-then-else equations that varied with the stress values. In order to properly identify the panels, as well as the stresses at the correct location on each panel, designers had to hand-enter a code for each node and element number.

"The manual intensity of the analysis process made it excessively cumbersome," says Xu, "and like all manual data entry, it could be prone to human error." Also, it was too time-consuming to permit much design modification and iteration, especially given the importance of meeting tight deadlines in the offshore oil and gas industry. "It was clear that we would benefit from shortening the process," Xu adds.

Shorten it they did.

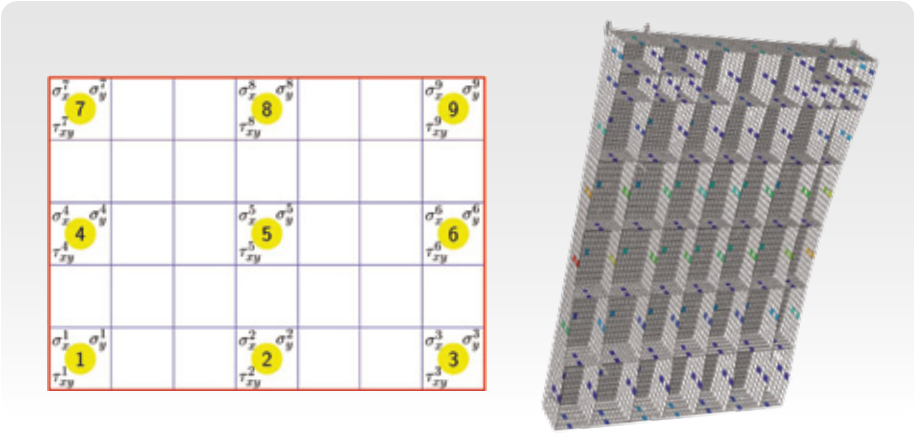
Taking a load off the design cycle

The application programming interfaces (APIs) within Abaqus software helped engineers at Foster Wheeler develop their Mudmat*ician* software. "Abaqus has very good APIs for customized development," Xu says. "It allowed us to write software that let us work with either the geometrical Abaqus model itself or the results in the output database. That's a great advantage, and it enabled us to automate the analyses from finite element model development to post-processing and panel-wise buckling assessment."

In fact, the new automated design cycle for mudmats involves four steps:

1. The designer specifies a series of pre-designed mudmat parameters (size, stiffener layout, and so forth) and creates a Python script design file.

2. The file is then run in Mudmat*ician*, which follows the parameters and generates a meshed model for use in Abaqus/Standard. The model uses spring elements to capture the load-bearing behavior of subsea soil. Model sizes may vary from 10,000 to 100,000 elements.



(Left) Figure 2. A single mudmat panel with 27 normal and shear stresses defined at 9 locations. (Right) Figure 3. Mudmat*ician* output of mudmat buckling checks based on stress analysis results from Abaqus FEA. The utilization factors are shown as a discrete field "B" in contour. Each factor is shown right at the point (element) where it is calculated. The color represents the value of the utilization factor, with least being in blue and greatest being in red. Above, some panels, with utilization factor(s) greater than, are over the buckling limit.

3. The designer adds boundary conditions and loads that are determined by subsea soil properties and by load points for equipment (and, in some cases, by lifting points such as padeyes or trunnions) and a Python script transfers the data into Abaqus. The FEA software then analyzes the mudmat, determining panel stresses.

4. Once the analysis is complete, Mudmat*ician* is employed as a post-processor to extract panel stresses (again, via a Python script) from the Abaqus output database. Mudmat*ician* also performs a buckling check based on the industry code implemented in the software. Although it involves normal and shear stresses at nine different locations per panel, the buckling check is performed only at a few weak points: the center of the panel for the top panels, and the center of the panel and the middle of the free edge for the vertical panels (See Figure 3).

During this process, Mudmat*ician* also reads the design file to identify each of the panels. After each buckling check, results are expressed as utilization factors, the ratios determining whether buckling strength is within acceptable levels. Being less than unity means within the limit, while being greater than unity denotes buckling inadequacy.

When the last step is finished, the designer can evaluate the overall integrity of the mudmat. If the strength seems insufficient or overly conservative, the engineer can return to the first step and quickly analyze a re-designed mudmat. Remodeling is easy and involves changing one or more parameters in the design file to build a new model with Mudmat*ician*, and re-running the FEA and the buckling check. This capability helps designers to easily identify and reduce the likelihood of potential weak points in the mudmat.

Automation = fast, trustworthy results

To validate the new technique, the engineers benchmarked Mudmat*ician* analyses against existing manually designed mudmats and compared the manual and software results. The correlation was good, and the first mudmat designed entirely with Mudmat*ician* has now been completed.

"The amount of man-hours spent on the design using Mudmat*ician* was dramatically reduced," says Xu. "What once took weeks or even months now only takes days. That gives us the freedom to re-design if necessary, while meeting tight deadlines."

In one case, the new automated analysis path enabled Foster Wheeler Upstream to offer emergency services to a client who requested an extremely rapid mudmat design for an urgent project. "We wouldn't have been able to help them so quickly without Mudmat*ician* and Abaqus," Xu says.

"Together, Mudmat*ician* and Abaqus have streamlined our analyses and shortened our design process," Xu says. "With improved engineering efficiency and design modification, it is now much easier to optimize mudmats. This allows us to deliver the highest-quality mudmat designs to our clients, quickly."

Abaqus Extension for Threaded Connections

Customized extension for Abaqus/CAE accelerates modeling and analysis of threaded connections



The performance of threaded connections is critical in several industries including the Oil & Gas industry, where they must operate reliably under high pressure and temperature conditions. FEA can be used to improve the reliability of threaded connections, but simulation set-up can be complicated, requiring accurate representation of the thread, box and pipe geometries, as well as the material properties. Often, analyses are not limited to a single connection, but need to be carried out for an entire family of connections.

The Threaded Connection Modeler (TCM), one of many Abaqus Extensions offered by SIMULIA, was created by our Southern region office in the US to provide a tool for quickly building entire families of threaded connection models and performing the make-up and loading steps.

The TCM plug-in is implemented using a Model-Tree approach. As each step in the process of building a threaded connection is completed, icons throughout the model tree are modified to provide immediate feedback on the user's progress. As an integrated Abaqus plug-in, all of the features of Abaqus/CAE can be used to enhance the model.

Additional Extensions for Abaqus

The robust architecture of Abaqus FEA provides users with the ability to create custom interfaces and automate analysis processes. In addition to the Abaqus extension for threaded connections, our regional offices have developed and supported a variety of additional extensions for specific industry applications, including the Abaqus Welding Interface, Adjustable Rigid Torus (ART), and Bolt Studio.

For More Information
www.fwc.com
www.simulia.com/products/extensions

Meet CASIMIR, the Human-Body Model

Wölfel uses Abaqus FEA and adds anatomical detail to improve design of vehicle seats

The human body is an elegantly complex structure. An adult has 206 bones, 639 skeletal muscles, countless tendons and joints, and more than 50 trillion cells. Humans are capable of running a mile in less than four minutes, jumping four vertical feet from a standing position, and bench pressing 1,000 pounds.

Even if a design team could put together a fully-detailed set of specifications for such an amazing, living machine, the human body would be impossible to build from scratch. So creating a realistic 3D computer model of it is extremely challenging. However, in the pursuit of more comfortable car seats, that is just what engineers on the Seating Comfort and Biomechanics team at Wölfel in Höchberg, Germany, are doing. Their solution, a 3D human-body model called CASIMIR (Calculated Sitting Man in Research), uses finite-element-analysis (FEA) to assess seat designs (Figure 1).



Figure 1. The CASIMIR family of human-body models includes several versions determined by the height and weight percentile that they represent: f05 corresponds to a small female; m50 a man in the 50th percentile; and m95 a large man. Adaptations can be made to account for physical variations of other nationalities and races, and an individualization algorithm further customizes the model.

Why car seat comfort is important

Seating comfort is an elusive yet increasingly important and competitive feature in vehicle seat design. Complex in nature, it is dependent on a wide array of factors: sitting position when driving, seat materials, the structure and shape of the seat, vibration from the vehicle, and the length of time spent driving, to name a few. While car seats used to be taken for granted, the simple act of a customer getting behind the wheel at a dealer can be a make-or-break moment in a potential car sale.

For those who spend a lot of time driving—such as traveling salesmen, bus and taxi drivers, airline pilots, and construction workers—issues of comfort can also become issues of health and safety. A recent survey of U.S. and Swedish drivers, for example, found 50 percent reporting low-back pain. In the EU, heavy-equipment operator safety is now regulated by the 2002/44/EC standards, which quantify daily exposure limits to vibration. At stake are issues as serious as neurological symptoms (tingling and numbness), circulatory disorders ("white finger"), and musculoskeletal problems (wrist tendinitis and back surgery).

To automotive designers, the basics of seat design—such as shape, adjustability, and the right balance of firm and forgiving foam—are important when considering comfort and health. So are the hand-arm and whole-body vibration that vehicle occupants are subject to. Other important questions include: How can seat comfort be determined so that an objective and comparative assessment is possible? And how can comfort be predicted early in the design stage before manufacturing expensive seat prototypes?

The evolution of a car-seat model

The CASIMIR human-body model has been the answer to such questions for Wölfel since 2003, as a stand-in for living human subjects in virtual analyses and optimization of vehicle seats. "CASIMIR's advanced finite-element-analysis capabilities have proved invaluable in helping us to assess seating comfort and examine the potential health effects of a wide variety of vibrations and loads on drivers and passengers," says Alexander Siefert, assistant manager for Seating Comfort and Biomechanics at Wölfel.

Born in the mid-1990s, CASIMIR was the brainchild of the Darmstadt University of Technology and the Federal Institute of Occupational Health and Safety (FIOSH) in Germany and was originally used to compute forces on the spine. But its value to vehicle seat designers was soon recognized, and

at the Society of Automotive Engineers' (SAE) annual Digital Human Modeling Conference—where CASIMIR has been a longtime virtual attendee—ideas for its finite-element-analysis evolution have been an ongoing topic of presentations for years.

The earliest CASIMIR models were fairly abstract versions, using lumped masses connected together to represent the body. Subsequent models increased in complexity until, in the current model, volumetric parts representing the tissue of the body can now interact in three dimensions with each other as well as with the seat itself. The model's posture can now even mimic that of its human counterpart, varying from erect to normal to slouching, a feature important for evaluating low-back pain.

The elements of a seating comfort analysis

Simulating the interactions of a human-body model with a seat model is a complex process involving: material selection for nonlinear and frequency-dependent representations of seat foam, padding, trim, and human tissue; analysis of contact in the interaction zones between the metallic seat frame and foam, and between seat upholstery and the human body; and accounting for large deformations in the seat foam and human tissue. To handle such complexity, the Wölfel seating comfort team chose Abaqus unified FEA from SIMULIA, the Dassault Systèmes brand for realistic simulation. "We investigated other FEA solvers," says Siefert, "but only the capabilities of Abaqus satisfied all of our requirements."

For evaluating seating comfort, a prerequisite is to first build a model of the seat. Over time, the Wölfel team has done that, developing an extremely accurate representation that defines key properties: the material parameters of upholstery materials; and the static and dynamic stiffness and damping values of structural joints (e.g., for the recliner and seat rail).

Once the seat model is defined, a typical seating comfort analysis involves two separate studies. The first, the seating-process analysis, is static and looks at the way the body contacts the seat under gravity. With the most important aspect of this analysis being the seat pressure distribution, this calculation helps designers evaluate the fit of body parts with seat cushions, the ergonomic position of the body, and the distance between the seat frame and the skin.

The second study, a dynamic simulation, computes how the human body in combination with the seat moves up, down, and sideways when the vehicle is in motion. This simulation looks at the way in which the seat translates the motion of the car into the movement of the body, and what the subsequent loads are on the body, especially the spine. The seat-transfer function, an important component, illustrates how the seat magnifies the vibrations that pass from the road and engine through the chassis to the seat and eventually the person. A large amplification is bad. A small amplification is good (Figure 2).

For both types of analyses, the team has utilized Abaqus Standard, based on an implicit solver scheme: the static analysis uses a nonlinear solver algorithm and the dynamic analysis a steady-state solver in the frequency domain. Four CPUs with Linux 64-bit Red-hat operating systems and approximately 64GB total RAM, powered the calculations.

In the end, the seating comfort analyses using a homogeneous version of the CASIMIR model (in which different tissues like fat and muscles are not differentiated) have shown good correlation with real measurements. Using this tool, the Wölfel team has successfully employed simulation for years to design seats (Figure 3). Virtual prototyping has helped them zero in on features, compare different designs, and rapidly optimize for both static and dynamic comfort. As testimony, many automakers—including BMW, Mercedes-Benz, and Ford—have seats on the road that have benefited from CASIMIR's input.

In addition, simulation has minimized the need for expensive physical prototypes. "Designing and building a single hardware seat prototype can cost 10,000 to 20,000 euros and take from several days to weeks, while model setups take less than a day," says Siefert. "And car-seat development always involves a lot more than only one design iteration. So the savings from using simulation can be significant."

Furthermore, according to Siefert, by using a human-body model for car-seat analyses, resonance/vibration problems can be detected early on, expensive design changes late in the cycle can be avoided, comfort simulations can be standardized, and outside-the-box designs can be considered.

Continued on page 22

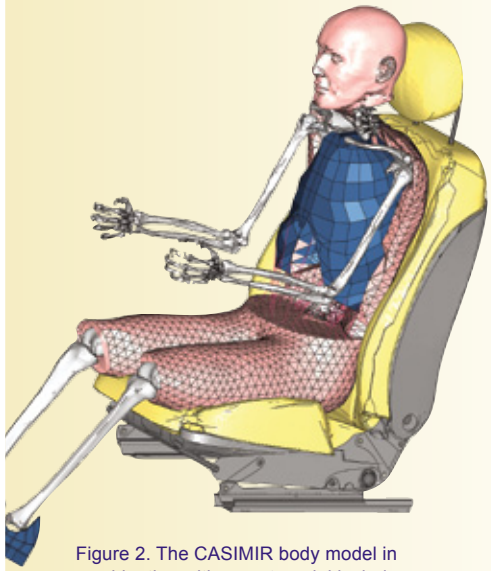
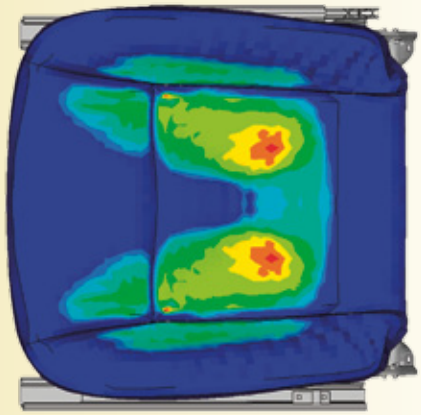


Figure 2. The CASIMIR body model in combination with a seat model includes contact definition between the structural frame of the seat and the upholstery, as well as between the upholstery and the human-body model.



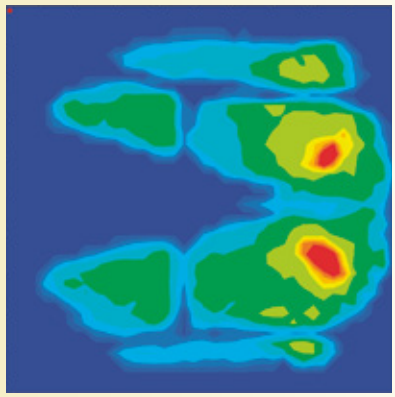


Figure 3. A typical seat-pressure distribution on a standard seat (from a static seating comfort analysis) shows a good correlation between CASIMIR (top) and measurements from a test driver (bottom).

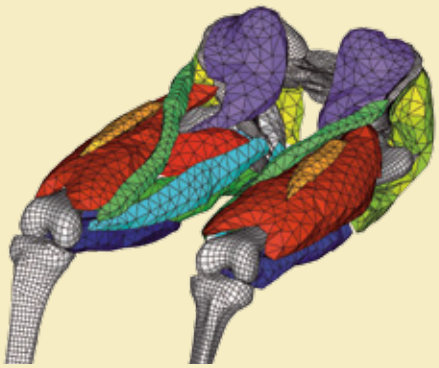


Figure 4. MRI imagery is assembled into 3D geometries, meshed, and then imported into CASIMIR's enhanced thigh and buttocks continuum model. In the model, muscles are represented in different colors.

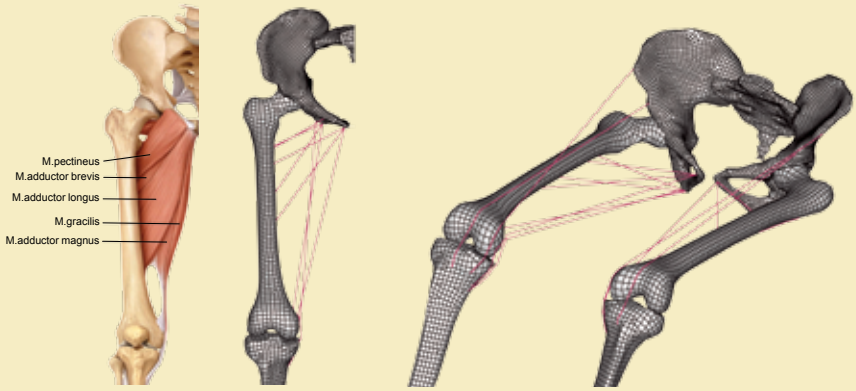


Figure 5a (left). Anatomical presentation of the thigh musculature and the realization in CASIMIR's discrete muscular model. Muscles that are connected to the skeleton on a line, rather than a single point (e.g., M.adductor magnus), are implemented by using more than one string in the model. Figure 5b (right). Discrete muscle model of the thigh and buttocks for CASIMIR includes 52 strings.

"CASIMIR combined with the power of Abaqus FEA gives us the ability—and the freedom—to cost-effectively pursue innovative concepts in car-seat designs," says Siefert.

More anatomical data creates better realistic models and simulations

While CASIMIR has already provided automakers and their customers with better designs and improved comfort, the Wölfel team believed that an enhanced CASIMIR could provide even more. When looking to improve the model's capabilities, the rule-of-thumb was this: where the functionality and response of the human body is concerned, the more realistic the model's anatomy is, the more accurate the results of the simulation. "By adding additional anatomical data to CASIMIR, we are able to increase the accuracy of our simulations," says Siefert.

As a first step in enhancing CASIMIR, the Wölfel team developed a continuum model of the thigh and buttocks area, which represented the fat, muscles, and skin separately. To make the model more realistic, they took data from the U.S. National Library of Medicine's Visible Human Project (photographs of 600 cross-sections at 1 mm thick intervals, supplemented by CT and MRI images), assembled the 2D cross-sections using SCAN-IP, and then imported the digitized anatomy into the model. The result was more lifelike 3D muscle volume detail. They then meshed the model with the HyperMesh pre-processor, using 43K linear tetrahedron elements, with a typical size of about 20 mm and only 47K DOFs for the buttock tissue, to shorten computation time (Figure 4).

The team further enhanced CASIMIR's thigh/buttocks area, developing a discrete muscle model that more accurately represented the complex musculoskeletal behavior of the body's hip and thigh. Starting with a standard anatomy atlas as reference, they used spring and damper elements (similar to those used in automotive fatigue analyses) to simulate the active muscle behavior. To define the muscle's line of action, they used strings (Figure 5). And around the joints, where the muscle action is most complex, they defined a muscle path (the shortest distance over the joint) and attached it by connector elements to the skeleton.

Applying both enhanced thigh/buttock models within the standard simulations, the team found that, in the dynamic analysis, muscle activity influenced seating comfort. More specifically, they determined that as the muscles of the thigh/buttock's area were flexed, there was a reduced rotation of the pelvis and a higher compression on the intervertebral discs of the lumbar spine. In addition, they established that these internal forces, which can cause low-back pain, were accentuated at the human body's natural resonance of 5 Hz, a frequency that is excited by normally occurring driving conditions. Overall, the results gave the team a better window into how the body experiences internal forces while vibrating and moving on the seat.

The analyses also gave the team guidelines for how to improve seat designs. When a driver, for example, pushes on the gas and brake pedals, the muscles of the leg flex and stiffen, affecting the comfort of the seat. Passengers, on the other hand, are more passive and do not activate their leg muscles. Making this distinction, the team can now use the original homogeneous CASIMIR model for passenger simulations and the enhanced CASIMIR for driver simulations.

The enhanced driver simulation, in turn, will provide more accurate guidelines for improving driver-seat designs, such as adding length to the seat to better support the thighs.

To advance their car-seat simulation capability even further, Wölfel engineers continue to refine CASIMIR. The team is already modifying it to combine the volumetric and discrete muscle models and include their interaction. They are also planning enhancements for the shoulder-hand-arm system, as these parts of the body are integral to comfort and health as well.

In related applications, Wölfel is even planning to return CASIMIR to its biomedical roots and use it for the development of implants to stabilize the lumbar spine. There is also talk of turning their increasing tissue- and skeletal-modeling knowledge to a variety of other biomechanical simulations, such as implant designs for the knees and hips, as well as design improvements for wheelchairs, office seats, and mattresses (to prevent bedsores).

With CASIMIR in the driver's seat and still evolving, Wölfel continues to point automotive engineers toward new and improved seat designs. When innovations in seat designs are brought to market, drivers of all kinds—commuters, commercial, and industrial—can look forward to a more comfortable ride, less vibration and loads, and the promise of long drives with less driver fatigue and low back pain.

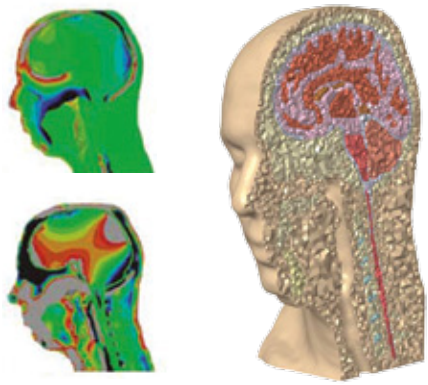
For More Information
www.woelfel.de

Anatomically Accurate Computational Models from Simpleware Provide Insight into Brain Injury

To save the lives and improve the recovery of its service men and women, the military wants to better understand the prevention and treatment of brain injury. The U.S. Naval Research Laboratory has recently worked to further apprehend the brain's response to shock waves, such as those produced by explosions. Advances in medical imaging have pushed conventional modeling techniques to their limits.

The U.S. Naval Research Laboratory, in cooperation with Simpleware and SIMULIA have used their combined expertise to convert highly detailed medical image data into complex finite element models for realistic computational analysis using the Simpleware developed head model and Abaqus/Explicit.

The anatomically accurate model was developed from a high-resolution MRI of a healthy 26-year-old male. In Simpleware, different regions of the brain were segmented by tools based on grey-scale levels, region growing algorithms, filters and more. Separation of soft and hard tissue was relatively easy; however, achieving accurate identification of the small, interlinked



High resolution MRIs (left) are used to develop anatomically accurate models (right) for realistic computational analysis.

tissues proved to be more time consuming. In preparation for the finite element analysis, the regions were automatically converted into high-quality, multipart tetrahedral meshes with conforming interfaces between the more than 15 soft and hard tissue structures identified. Mesh parameters were adjusted to give meshes between 1 and 10 million elements.

Within Abaqus FEA, material properties obtained from literature were assigned to different parts of the head model. The brain was viscoelastic, the cerebral fluid hyperelastic (with low shear stress and high bulk modulus) and the skull, vertebrae and generalised tissues were assumed linear elastic. The head model was then subject to a pressure wave simulating the blast generated by an explosive charge at a distance.

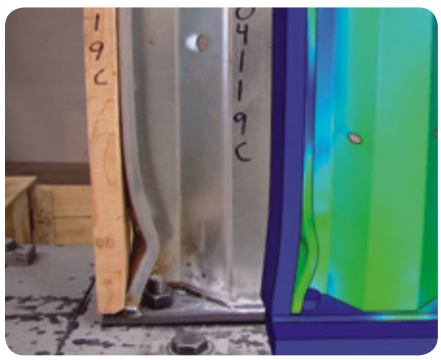
The Abaqus analysis was able to predict the complex stress patterns, oscillating tensile and compressive, across much of the brain surface and to both translational and rotational motion around the brain stem. These findings are significant in understanding the effects on a human brain hit by such a blast. The knowledge acquired from combined computational, experimental and clinical data will be used to further helmet design to reduce the risks faced by front line forces.

For More Information
www.simpleware.com
www.nrl/.navy.mil/content_images/09_FA3.pdf

Strong-Tie Strengthens Product Design with Intel and Abaqus

For more than 50 years, Simpson Strong-Tie has produced structural products for building safer, stronger homes and buildings. While, on the surface, producing structural connectors and fasteners might not appear to require cutting-edge computing technology, the Simpson Strong-Tie team knows the value of running realistic simulation solutions on a high-performance computing (HPC) cluster. "Using HPC for modeling is critical in product design," says Frank Ding, R&D engineer at the company. "To build a physical test specimen with concrete, we would need 28 days just for the concrete to reach design strength. With HPC, we can simulate that environment in two days. Product developers use HPC to design and optimize products for their specific goals in a very short time frame."

As the company's existing cluster neared the end of its life, Ding looked for the processing platform that could deliver the best performance for running Abaqus/Explicit. Benchmark data, provided by



Simpson Strong-Tie's *Steel Strong-Wall* shear wall pushover analysis

SIMULIA, led Ding to select a Certified Intel® Cluster Ready system based on Intel Xeon® processors. "The Abaqus benchmarks clearly showed that the Intel Xeon processors delivered better performance than the competing processing architecture, which we were using in our existing cluster," says Ding. "To be sure, we submitted sample jobs to SIMULIA to see how our workloads would perform, and the results showed significant performance improvement on Intel processors compared with our existing platform. It was an easy decision to move to Intel processors."

As the primary operator of the HPC system, Ding also wanted to work with a system integrator that could deliver a turnkey solution. The benchmarking information led Ding to system integrator Silicon Mechanics. Using the configuration tool on the Silicon Mechanics website and their team's support, Ding was able to design a cluster which uses a Silicon Mechanics Rackform iServ R346 head node and four more R346 servers for the compute nodes. All of the nodes are equipped with the Intel® Xeon® processor 5500 series. The servers run Abaqus/Explicit software on the Red Hat Enterprise Linux operating system.

For More Information
www.simulationclusters.com
software.intel.com/file/29638

A Quantitative Patient-Specific Study of Femoral Fracture Fixation Techniques

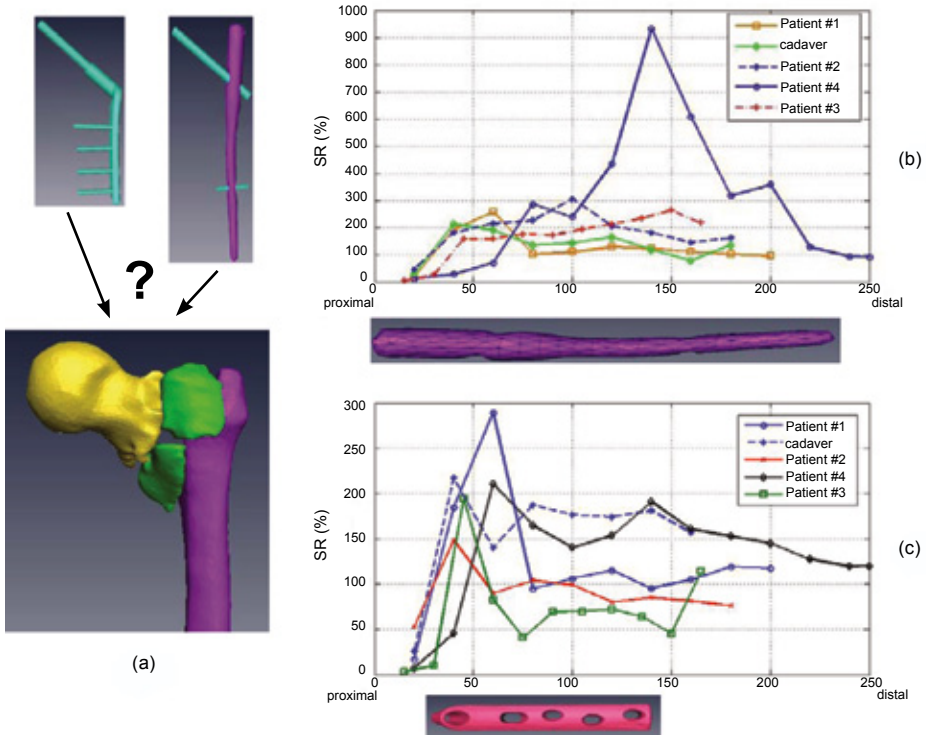
Fractures of the hip are relatively common in adults and often lead to devastating consequences. Fractures of the upper part of the thigh bone (femur) are termed hip or proximal femoral fractures. Roughly half of all hip fractures are extracapsular, in that they lie outside the hip joint capsule. These fractures may be surgically fixed using metal implants. Although the sliding hip screw has become the standard fixation device for extracapsular hip fractures, fixation failure has been reported for up to 10% of patients. This recurrent failure of the sliding hip screw devices led to the development of intramedullary devices, in which the short lever arm creates a mechanical advantage that reduces the risk of fatigue failure of the implant. However, to date, no clinical study has shown that the expected advantages of intramedullary fixation devices have a significant effect on clinical outcome.

The research team at Hadassah University Medical Center believes that the choice of fixation hardware should be assessed on a quantitative, patient-specific basis. Patient-specific factors such as bone quality, bone morphology, bone geometry, and fracture geometry and configuration—and not just type—has significantly different fixation failure risks associated with them.

Analysis method

Four patients with fractured proximal femurs scheduled for surgery and a fresh cadaver were selected for this study. All cases had pertrochanteric fractures with varying instability and had also suffered from various degrees of osteoporosis. Prior to surgery, CT scans were acquired and from this data three Finite Element simulation models were constructed for each case scenario:

1) a virtually intact femur model created by virtual fusion of the bone fragments at fracture surfaces;
2) a fractured bone model fixated with a Richards® Dynamic Hip Screw (DHS);
3) a fractured bone model fixated with a Synthes® intramedullary Proximal Femoral Nail (PFNA) (Figures 1a-b).



(a) Examples of 3D models of the extra-medullary DHS and intra-medullary PFNA fixation devices (top), and multi fragmental fracture (bottom); (b-c) Strain Ratios (SR) computed from FE analysis for all five cases for each fixation device (PFNA top, DHS bottom). The horizontal axis indicates the region in which the SR was measured (femur head, proximal, medial, and distal femur shaft). The vertical axis indicates the SR as a percentage of with respect to the intact bone maximum principal strain. Note the significant differences between patients, regions, and fixation devices.

The intact femur model was used as the reference for the strain ratio computation and for the fixation solution comparisons. Realistic simulation was performed for each of the three models with Abaqus Unified Finite Element Analysis (FEA). Identical boundary and loading conditions were used for all models: a distributed force on the femur head ≈ 2.5 times the patient body weight. The Strain fixation Ratio (SR) for the DHS and PNFA were computed from the intact femur model and from the corresponding fixated models.

Analysis results

The results of the study indicate that the implant and fixation choice has less importance for stable fractures (patients 1 and 4). However, in the case of an unstable fracture type, the choice of fixation type may be of significant clinical importance (patients 2 and 3). The results clearly show patient-specific differences in stress shielding and over stressed patterns. These types of considerations may enable patient-specific preplanning examination and plan modifications to overcome biomechanical pitfalls during surgery, and post-operative recommendations for unstable fractures.

Quantitative patient-specific considerations, including bone morphology, bone quality, and bone and fracture geometry may play a significant role in the choice of inter-tronchanteric fixation type, and reduce fixation failure risk for in the proximal femur. The SR-based comparative approach, which is independent of load amplitude as opposed to regular strain based analysis, indicates that this proposed method is more reliable than existing approaches and may be suitable for clinical use.

Eran Peleg[1], Leo Joskowicz[2], Meir Liebergall[3], Rami Mosheiff[3]

[1] Dept. of Medical Engineering, Hadassah University Medical Center, Jerusalem, Israel.

[2] School of Engineering and Computer Science, The Hebrew University of Jerusalem, Israel.

[3] Dept. of Orthopedic Surgery, Hadassah University Medical Center, Jerusalem, Israel.

For More Information
www.hadassah.org.il/english
www.simulia.com/cust_ref

Professor Plum—in the Library—with a Wrench

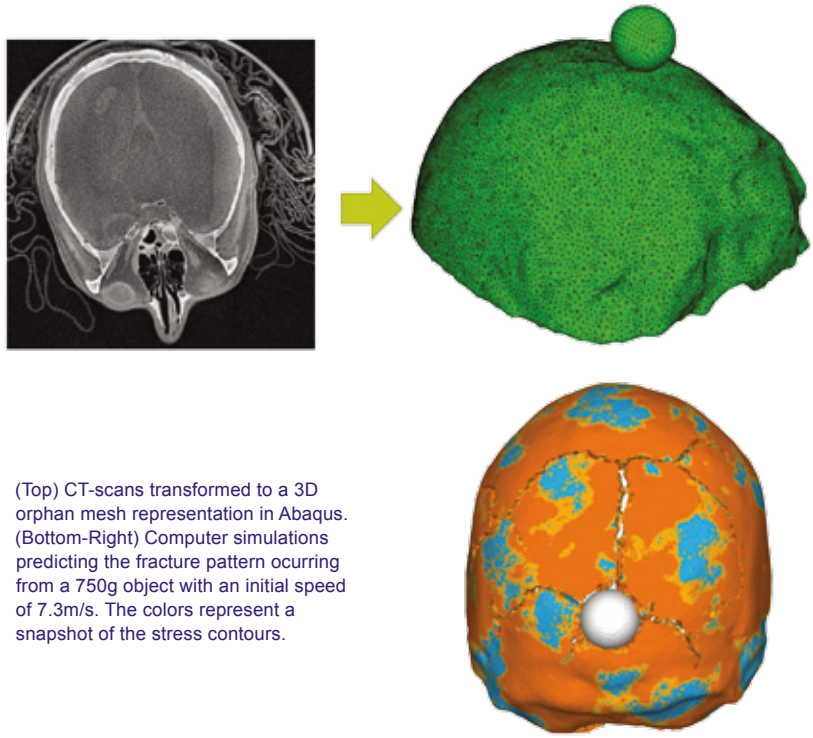
The Forensic Study of Skull Fracture with Realistic Simulation

In cooperation with the Institute of Forensic Medicine at the University of Copenhagen, the Technical University of Denmark (DTU) is performing realistic dynamic skull fracture simulations of specific autopsy cases. The work has, until now, been based on two master student theses from the relatively new Master of Science in Medicine and Technology from University of Copenhagen and the Technical University of Denmark.

An important issue in forensic studies of head injuries is to determine how a specific skull fracture arose. This information can be critical for determining if a criminal offense has taken place or if the injury was caused accidentally. The medical examiner chiefly relies on his experience, since only a few scientific methods are considered sufficiently reliable. At the Institute of Forensic Medicine, University of Copenhagen, all autopsy cases are computer tomography (CT)-scanned, but the obtained CT-scan data is mainly used as a visualization, documentation and filing tool.

During the last decade computers and finite element codes have reached a stage where accurate finite element predictions of complex fracture processes in large 3D structures are possible—for example, a full 3D finite element simulation of a skull fracture. In addition, tools are available to transfer CT-scan data to a 3D finite element model. Thus, research is being conducted to determine the feasibility of using CT-data combined with FEA technology in a more proactive way in forensic medicine.

The first finite element model created by the research group was based on a procedure using the software Amira on a CT-scan from a forensic case. A 3D finite element model of the skull was extracted where the skull material was considered as a homogeneous isotropic linear elastic material. Even though only a rather simple geometrical part of the skull was considered, some manual work to improve the orphan mesh was necessary to create a usable FEA model. In the subsequent finite element analysis, a simple element-removing algorithm describing the fracture process was used, and the skull was simulated using Abaqus/Explicit.



(Top) CT-scans transformed to a 3D orphan mesh representation in Abaqus. (Bottom-Right) Computer simulations predicting the fracture pattern ocurring from a 750g object with an initial speed of 7.3m/s. The colors represent a snapshot of the stress contours.

The skull was exposed to a short dynamic impact from a blunt object. Based on the simulations, a large sensitivity of the fracture modes was found even for small changes in the initial speed of the incoming object. The modeled fracture patterns differ somewhat from the actual fracture pattern of the specific case. This may be due to some of the simplifications used in this first version of the model.

A more realistic simulation should take into account a more correct cranial vault structure, including the spongy bone layer with a low density between the two compact bone layers. In addition, the effects of the sutures between the single cranial bones should be included, as well as a more realistic fracture process simulation. These points are addressed in a new and ongoing master theses project. A work including the use of the implicit dynamic eXtended Finite Element Method (XFEM) introduced in Abaqus 6.10 allows for a much more realistic and mesh independent prediction of arbitrary crack growth.

A more correct representation of the cranial vault structures can be achieved using a 3-part mesh generating code such as Simpleware coupling the gray-level on the CT-scan with varying material properties or by using the capability of the surface reconstruction in CATIA to transform an orphan mesh to a CAD representation of the surface and thereby achieving a higher-fidelity model in Abaqus.

The current research is considered as a preliminary step on the path to developing a general tool for supporting medical examiners with easy, achievable and accurate numerical simulation of a specific forensic case to support their judgment regarding the cause of death. To achieve this goal, additional work and improvement in modeling and analysis process is required and underway.

For More Information

Finite element simulation:
www.pmik.dk/risoe
Dr. Lars P. Mikkelsen
lapm@risoe.dtu.dk

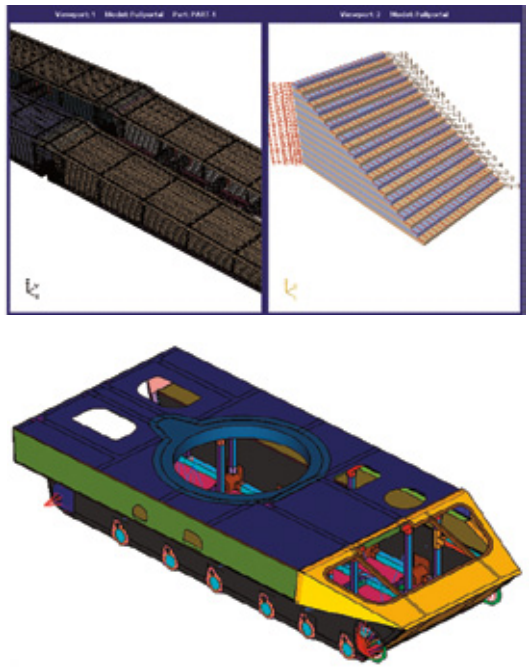
Forensic medicine:
http://retsmedicin.ku.dk/english/
Prof. Niels Lynnerup
nly@sund.ku.dk

Research and Development Establishment (Engineers) Drives Composite Innovation

Composite materials are increasingly utilized by many industries due to their strong yet light-weight properties and the ability of their structural response to be tailored as needed. R&DE (E) has selected Abaqus FEA software for its robust linear and nonlinear structural analysis capabilities, as well as advanced composites modeling and simulation features. Abaqus is being used for performing not only static, but also dynamic load case simulations of impact penetration, vehicle dynamics, and geotechnical interactions. These are essential in determining the right composites material to be used in applications such as bridges, military vehicles, sonar domes, and ship superstructure.

"The bridges and vehicles developed at R&DE (E) need to withstand very harsh environments. Abaqus FEA provides accurate, realistic simulation capabilities for designing safe, reliable products," states Mr. U.R. Gautam, group director, Integrated Management Systems Group, R&DE (E).

>> www.drdo.gov.in/drdo/labs/RDE(E)/English



(Top) Abaqus unified FEA software is also used by R&DE (E) to analyze the structure and durability of composites bridges to meet performance objectives ahead of costly prototype testing. Shown are close-up detail (left) and composite layup analysis (right). (Bottom) RDE (E) can determine the best composite material for each application during the design phase with Abaqus FEA from SIMULIA. Realistic simulation, which includes sophisticated tools for composites fracture and failure prediction, allows engineers to develop safe, reliable products in a shorter design cycle resulting in significant savings.



Senergy Group to Improve Energy Exploration with Realistic Simulation

Senergy is a leading provider of global engineering services for the oil & gas and alternative energy industries. They have selected Abaqus Unified FEA as their primary analysis tool for geomechanics and structural stability assessments within subsurface engineering projects due to its ability to accurately predict behavior of rock formations and to capture rock-structure interaction taking into account nonlinear effects such as plasticity and compaction that are important in subsurface geological environments.

"Using FEA provides us with a more sophisticated method to understand and predict realistic behavior of complex geological environments," says Phil McCurdy, principal geomechanics engineer, Senergy. "We selected Abaqus due to its industry-proven track record and well-documented use cases. Its open architecture also gives us the potential to link reservoir geomechanics models and flow models for more precise co-simulation and multiphysics solutions."

>> www.senergyworld.com



2011 SIMULIA Customer Conference

Hotel Fira Palace • BARCELONA, SPAIN

SAVE THE DATE: May 16–19, 2011

The 2011 SIMULIA Customer Conference (SCC) will be held in beautiful Barcelona, Spain, a city filled with world-famous museums, historical sites, shopping, entertainment, and culinary delights.

For more than two decades, the SIMULIA Customer Conference has provided a valuable forum for learning how engineers and academia are applying the latest simulation technology and methods to accelerate and improve product development. The SCC, made possible by the dedication of our customers worldwide, brings together an international community of realistic simulation users to share their knowledge and experience in advancing methods and technology for finite element analysis, multiphysics, process automation, design optimization, and simulation management.

Conference highlights Include:

- Customer papers featuring industry experts using SIMULIA solutions
- Full-day Advanced Seminars
- Updates on SIMULIA products, including Abaqus, Isight, and SIMULIA SLM
- Industry-focused Special Interest Groups
- Networking with peers to gain professional contacts
- Complementary solutions sessions hosted by our Partner Sponsors

Don't miss out on this tremendous opportunity to attend this year's conference.

Past presenting companies:

- Bausch + Lomb
- BMW Group
- Cordis Corporation
- ExxonMobil
- Honda R&D
- Kimberly-Clark
- Medtronic
- Motorola Inc.
- NASA Glenn Research Center
- Rolls-Royce
- Tetra Pak
- Toyota Motor Corp.

And many more…

The SCC: a valuable professional experience!

"This was my first SCC and it turned out to be a great venue to connect with industry experts and the SIMULIA team to learn about new features in Abaqus."

—Atul Gupta, Medtronic Inc.

"Excellent opportunity to discuss the latest trends in FEA and their efficient implementations in Abaqus."

—Mark Gurvich, Ph.D., Technical Fellow, United Technologies Research Center

"The SIMULIA Customer Conference provided deep and technical presentations of how industry and researchers are using Abaqus to solve difficult simulation problems."

—Dana Coombs, Synthes

For More Information
www.simulia.com/scc2011

SIMULIA Helps Me Breathe.



Simulation for the Real World

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

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

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

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, 2010.