SIMULIA
COMMUNITY NEWS

#04 FEBRUARY 2013



NOVO NORDISK IMPROVES PRODUCT INTEGRITY WITH SIMULATION

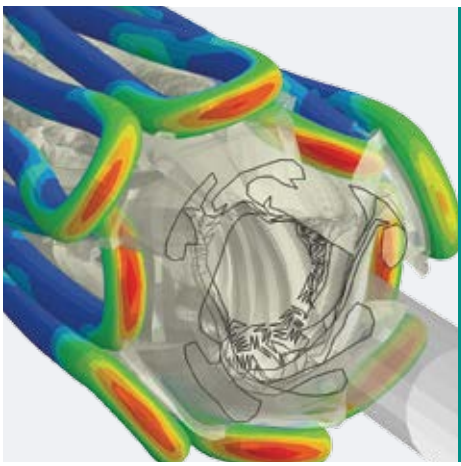
DASSAULT SYSTEMES | The 3DEXPERIENCE Company



FDA 4



NOVO NORDISK 12



MEDTRONIC 16

in this **Issue**

February 2013

3 **Welcome Letter**
Scott Berkey, CEO, SIMULIA

4 **Life Sciences Spotlight**
FDA Recognizes Simulation as Essential to Medical Device Evaluation

6 **Case Study**
Coca-Cola Keeps the 'Pop' in Soft Drinks with Realistic Simulation

9 **Product Update**
Abaqus Knee Simulator

10 **Case Study**
Saint-Gobain Reduces Bottle Weight with Abaqus and Isight

12 **Cover Story**
Novo Nordisk Improves Product Integrity with Simulation

14 **User Community**
2013 SIMULIA Community Conference

16 **Life Sciences Spotlight**
Medtronic Advances Stent Graft Designs

19 **Alliances**
Baker Hughes
Critical Materials

20 **Academic Update**
University of Iowa Optimizes Surgical Positioning in Total Hip Replacement

National University of Singapore Simulates Moisture Distribution in a Microelectronics Package

22 **Tips & Tricks**
How to Perform Wrapping and Unwrapping of Stents in Abaqus/CAE

23 **News & Events**
Download Abaqus Student Edition
New User Testimonial Video
New Technology Brief

Contributors: Torben Strøm Hansen (Novo Nordisk), Atul Gupta (Medtronic), Xavier Brajer (Saint-Gobain), Simon Shi (Coca-Cola), Jacob Elkins (University of Iowa), Andrew A. O. Tay (National University of Singapore), Peter Van Dyke (Baker Hughes), Nuno Faria (Critical Materials), Parker Group, Cheryl Liu, Matt Ladzinski, Samarth Shah, Scott Berkey (SIMULIA)
On the cover: Torben Strøm Hansen and Bo Uldall Kristiansen

SIMULIA Community News 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.3ds.com/simulia
Editor: Karen Curtis **Associate Editors:** Rachel Callery, Tim Webb **Graphic Designer:** Todd Sabelli

The 3DS logo, CATIA, SOLIDWORKS, SIMULIA, DELMIA, ENOVIA, GEOVIA, EXALEAD, NETVIBES, 3DSWYM and 3DVIA are either trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the USA and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners. Copyright Dassault Systèmes, 2013.

SIMULIA

Sharing Experiences

What do engineers simulating medical devices and carbonated beverage containers have in common? They both face complex physics and material modeling problems. The case studies in this issue clearly illustrate how our users in different industries are employing realistic simulation to solve similar engineering challenges.

This is made possible primarily by our users who understand their challenges and apply our technology, but also in part by SIMULIA's ongoing commitment to develop general-purpose FEA, multiphysics, design exploration, and simulation management software. These technologies enable our global user community to not only leverage our software effectively, but to share their knowledge and experience with our community at large to solve common engineering challenges.

An example of this cross-industry fertilization is our Polymer Customer Review Team, mentioned in the Novo Nordisk cover story (page 12). This group includes members from the automotive, high-tech, life sciences, and consumer goods industries who focus on providing input on the requirements for improving the modeling and simulation of polymer materials. Over the past few years, we have initiated several cross-industry customer review teams related to specific technologies or industry workflows. Their feedback drives enhancements and new technologies that are beneficial to everyone in our global user community.

Even if you are not in the medical device industry, I am sure your organization also faces some form of government regulation in your development process. The story about the United States Food & Drug Administration (page 4) highlights how the appropriate use of realistic simulation is now recognized as an integral tool for meeting regulatory requirements.

The consumer packaged goods industry must overcome similar engineering and regulatory challenges. It's amazing to see how Coca-Cola (page 6) uses Abaqus to help them keep carbonation from seeping out of their beverage containers. Our alliance partners, such as Baker Hughes (page 19), are also deeply involved with our community to enhance the methods and technology to understand the complex physical interactions between the earth's subsurface and oil and gas reservoirs during drilling and petroleum extraction.



The international SIMULIA Community Conference (SCC), this May in Vienna, Austria, is a great opportunity to meet in-person and learn how peers across all industries are using realistic simulation to solve challenges that may be comparable to yours. We are expecting 90 end-user presentations covering applications such as composites, contact, design exploration, durability, fracture, plasticity, process automation, material modeling, thermal analysis, topology optimization, simulation management, and much more—across some ten-plus industries.

The 2013 SCC will also feature an inside look at our next generation realistic simulation applications based on Dassault Systèmes **3D**EXPERIENCE platform. At the SCC and over the coming months, you will be learning more about these applications, which leverage proven Abaqus and Isight technologies within an innovative user experience. These applications will provide you with an unmatched toolset to perform realistic simulation, share simulation results, and collaborate on requirements-driven product development.

I look forward to meeting you at the 2013 SCC in Vienna to share more about our common experiences and goals. I hope you can join us!



Scott Berkey
CEO
SIMULIA

Simulation Now Recognized by FDA as Essential to Medical Device Evaluation

Cheryl Liu, Senior Technical Marketing Specialist, Life Sciences Industry, SIMULIA

One of the toughest design engineering challenges is making a medical device that works flawlessly with the human body. The unique anatomy and physiology of every patient create physical complexities, and ever-shifting functional parameters, that must be thoroughly accounted for when producing a therapeutic product that may need to last a lifetime.

Domestic inpatient procedures involving medical devices—stents, heart valves, dental implants, spine and joint implants, surgical tools, blood pumps, endovascular grafts, drug-eluting devices, and more—totaled 46 million in the U.S. alone in 2006, according to the Centers for Disease Control and Prevention (CDC). It's a global market that is growing along with aging populations everywhere.

Computer simulation, already widely accepted in many industries, is increasingly being viewed as an important tool by medical device companies and their designers. It helps them visualize what they cannot see, explore the design space more fully, refine their ideas faster and more accurately—and reduce expensive prototyping and testing.

Solid mechanics simulations can help determine proper implant size, evaluate manufacturing tolerances, compare design geometries, or consider next-gen devices. Fluid dynamics can be employed to identify high-shear stresses on blood vessels, regions of low flow, and potential for blood damage. And simulation-based product development processes can be linked in automated workflows, optimizing huge quantities of design data to provide exquisitely fine-tuned results that are of particular value for creating patient-specific medical devices.

FDA sees increasing numbers of applications that include simulation

As Life Sciences engineers embrace simulation, they are achieving increasingly accurate levels of precision when evaluating device function, including the ability to evaluate aspects of device performance not possible with bench tests alone. As a result, the Food and Drug Administration's (FDA) Center for Devices and Radiological Health (CDRH) is seeing a growing number of submissions for medical devices that include a simulation-data component.

The CDRH is responsible for regulating firms that manufacture, repackage, relabel, and/or import medical devices sold in the U.S. The submissions for these therapeutic devices typically contain data from four types of evaluation models—animal, bench, computational, and human—to demonstrate a reasonable assurance of safety and effectiveness. When a company submits simulation metrics that supplement bench testing, this can help promote approval by demonstrating both the integrity of the proposed device and the required realistic device failure analysis. As the ultimate safety-and-effectiveness regulatory body between medical device manufacturers and patients, the FDA recognizes the value of such advancing technologies—and its own need to stay abreast of them—and has now begun actively encouraging the use of simulation in device evaluation.

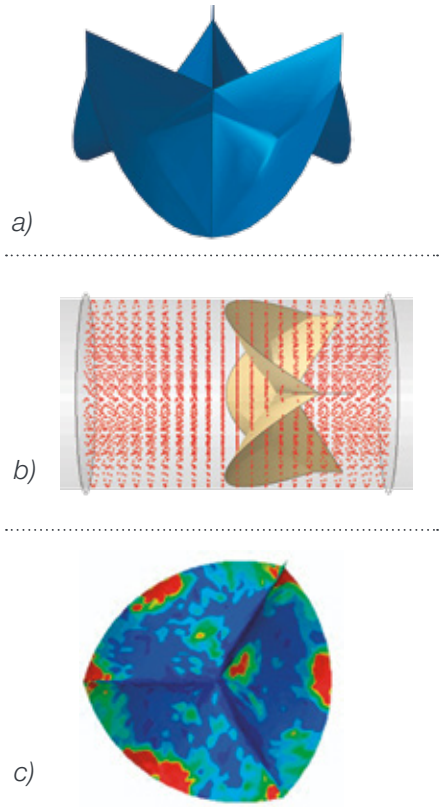


Figure 1. This example of modeling and simulation of a medical device shows an aortic valve geometry (a), a model of the effect of blood flow on the valve in a blood vessel (b), and an Abaqus finite element analysis (FEA) of the stress on the valve leaflets during the diastolic phase (c). This work was performed by SIMULIA in conjunction with the FDA's Center for Devices and Radiological Health (CDRH).

However, the FDA has also put the industry on notice that verification and validation (V&V) must go hand-in-hand with the use of simulation in applications. The CDRH is looking to quantify when a computational model is credible enough, and whether its intended purpose is appropriate for a regulatory submission. Unclear reporting standards, insufficient data about geometries and boundary conditions, lack of validation metrics, incomplete understanding of physiological loads in the body, and variations in patient populations—any and all of these uncertainties can impact the relevance of simulation outputs.

SIMULIA contributes to advancement of knowledge

Noticing that a significant proportion of the applications they have seen in recent years have included simulations with Abaqus finite element analysis (FEA), the CDRH reached out to us in 2010 for support in developing their own internal framework, and in-house expertise, for validating and regulating industry-submitted simulations.

SIMULIA presented at the FDA's 3rd workshop on Computational Modeling of Medical Devices the same year. We continue to deliver on-site training courses to FDA reviewers about best practices in modeling and simulation and to partner with the FDA on aortic valve model development (see Figure 1). The FDA has also presented at our SIMULIA Community Conference and Regional User Meetings. Realizing the importance of model V&V in early 2011, ASME and FDA launched the V&V 40 subcommittee to develop V&V guidelines for the medical device industry specifically; we are actively participating, along with others in the industry and software communities.

As one outcome of these efforts, the FDA will publish a guidance document titled "Reporting Computational Modeling Studies in Medical Device Regulatory Submissions" in 2013. Appendices will cover fluid and mass transport, solid mechanics, electromagnetism, control loops, thermal transport, and ultrasound. Publication date updates can be found on the CDRH website at **www.fda.gov/MedicalDevices/DeviceRegulationandGuidance/GuidanceDocuments/default.htm**.

The 'Virtual Patient' idea is born

As knowledge about the importance of simulation grows, another priority for the FDA is the creation of a publicly available 'Virtual Physiological Patient' of human body computer models in different disease states (see Figure 2). This is not intended to be a single model encompassing every function and disease at once. Rather, the project will comprise a library of verified and validated submodels and data based on the combined expertise of those groups in the relevant disciplines, i.e., cardiology, orthopedics, software, and so forth.

The goal of the Virtual Physiological Patient project is a shared point of reference that will improve understanding of model attributes and limitations, and provide discrete models, data, and simulations validated for regulatory evaluation. Peer review by experts in academia, government, and industry will ensure robust V&V and provide periodic assessment. SIMULIA is contributing expertise to a group that is developing a computational model for the evaluation of a diseased femoral artery for stent evaluation.

Newly launched public-private partnership benefits all parties

Concurrent with the development of the Virtual Physiological Patient concept, the FDA is reaching outward to device manufacturers, software providers, and medical professionals to form a Regulatory Science Public-Private Partnership. Launched in December of 2012, the partnership is called the Medical Device Innovation Consortium (MDIC).

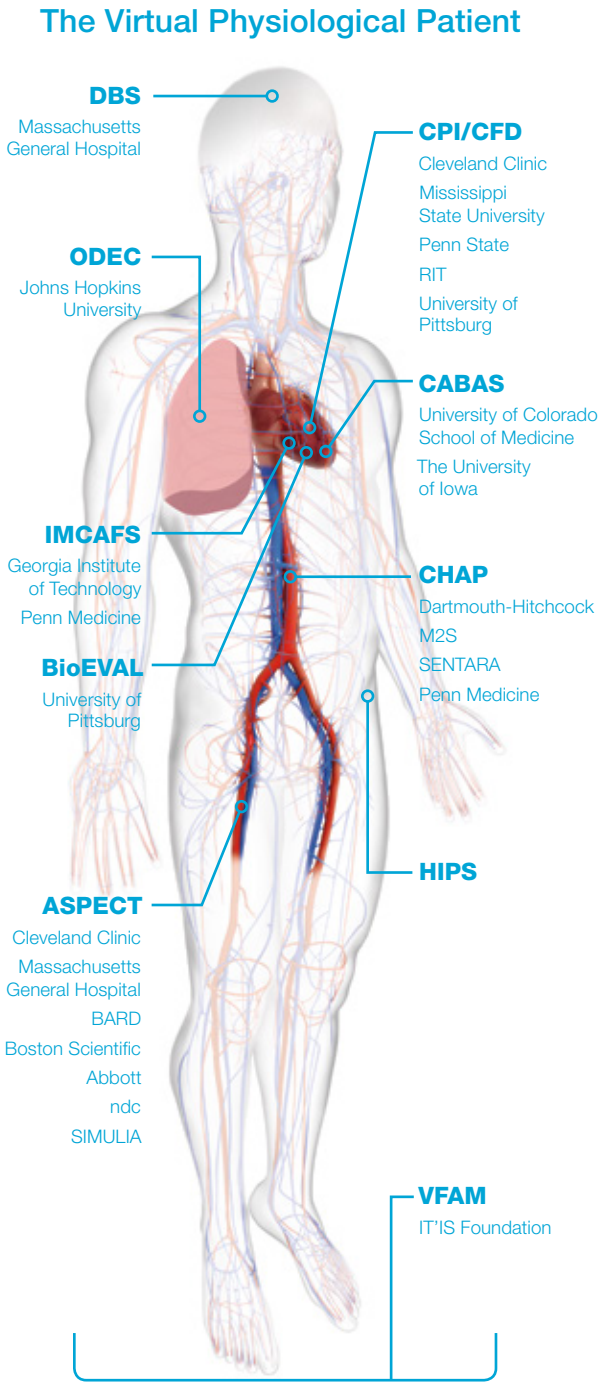


Figure 2. Broad cross-industry collaboration between medical device manufacturers, academia, and software companies is being harnessed for the FDA's Virtual Physiological Patient project.

The idea is to create an opportunity for information gathering in a pre-competitive state, i.e., not device-specific, but disease-specific. For example, if the heart valve community were interested in a comprehensive evaluation of the structure and function of heart valves, costs could be minimized through nonprofit group funding and participation in the development of tools and resources for modeling and simulating of a range of valves. All results would be shared. End-stage renal disease is another area recently identified by the FDA as a priority. Industry forums on this topic are already underway.

The medical device industry can only benefit from such endeavors. Individual device design copyrights certainly need to be protected, but the tradition of publishing evidence-based research results in order to move the entire body of medical knowledge forward has resonated in the life sciences throughout the history of medicine. A deep understanding of the function of the living body is critical to every medical-device developer, and sharing the data that lie at the core of that understanding can be accomplished without infringing on any one company's patents.

The FDA views modeling and simulation as incentives to innovation that can reduce the time and cost of device design, assessment, and manufacturing. It is in all our interests—the medical device industry, the regulatory agency, and software companies—to collaborate to ensure that the power of simulation is increasingly utilized to solve the wide range of challenges in medical device development. We can all agree that the ultimate goal is the safety and effectiveness of medical devices for every physician who uses them, and every patient who needs them.

Special thanks to Dr. Tina Morrison, Dr. Nandini Duraiswamy, and Dr. Donna Lochner of the FDA for their assistance in preparing this article.

Read more about how the FDA is promoting innovation in "High Stakes Balancing Act" in *Compass* magazine—**www.compassmag.3ds.com**.

For More Information

www.fda.gov
www.deviceconsortium.org



Bubbles In, Air Out: Realistic Simulation Helps Keep the 'Pop' in Soft Drinks

Coca-Cola design engineers use Abaqus to analyze carbon dioxide and oxygen flow in plastic beverage bottles

An ice-cold soft drink is one of life's reliable pleasures on a hot day. The whoosh of sound when you twist off the cap of the bottle, the icy tingle of bubbles on your tongue, the crisp taste as you swallow the refreshing liquid. But occasionally this simple joy falls flat: When you twist off the top there's no fizz. When you take your first gulp the drink is dull and tasteless. What exactly has gone wrong?

Engineers who design beverage bottles for Coca-Cola know precisely what has happened because it's their job to ensure that customers don't experience fizz-flop or tepid-taste in any of their soft drinks. After all, as the world's largest beverage company with more than 500 sparkling and still brands, Coca-Cola provides about 1.8 billion drinks daily in more than 200 countries.

Their research and development teams are charged with maintaining the optimum levels of carbon dioxide (CO_2) *inside* bottles to preserve fizz (in those drinks that are supposed to bubble). They also work to keep *out* oxygen (O_2) as long as possible, to avoid compromising taste and freshness in both carbonated and still liquids (such as fruit juice, milk, and tea). And Coca-Cola's distributors around the globe count on such expertise to guide them about storage temperatures and expiration dates.

Advantages of plastic bottles come with new challenges to designers

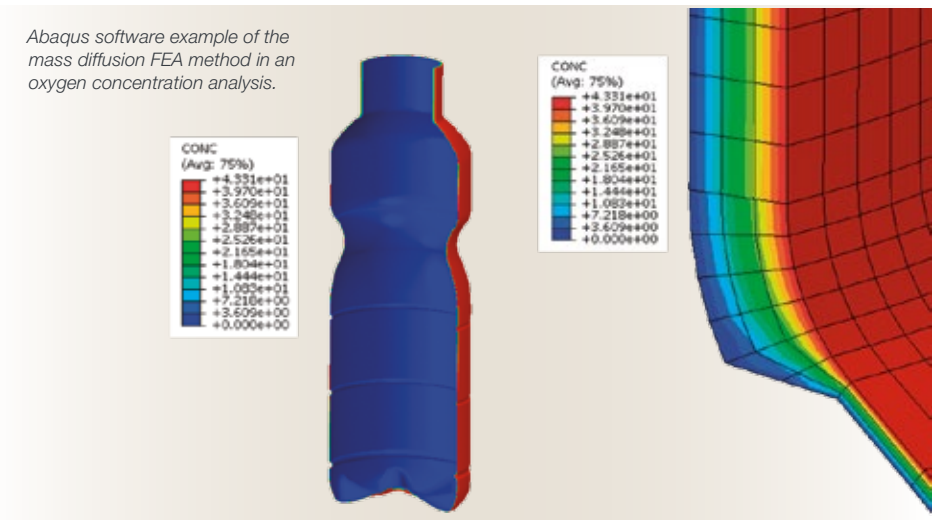
Back when soft drinks were freshly made at pharmacy soda fountains or carried home in capped glass bottles, fizz and taste could be pretty much guaranteed. Then soda fountains gave way to vending machines, aluminum cans replaced heavy glass, and finally plastic bottles came on the scene. Lighter, less-expensive, resealable, recyclable plastics provide many advantages to both consumers and bottlers. Yet maintaining product uniformity in the face of time, climate, and travel has become more of a challenge.

The widely used polyethylene terephthalate (PET) beverage bottle of today is made from one of the more impervious of plastics—40 times more efficient than high-density polyethylene (HDPE), for example—but CO_2 and O_2 nevertheless trickle slowly through its walls over time. PET bottles are manufactured via a melting-and-forming, 'stretch-blow molding' process that orients and crystallizes the plastic on the molecular level. This affects the movement of the two gases through the bottle walls in distinct, complex ways. Although you can see the amount of CO_2 bubbles in a drink for a rough gauge of its fizziness, human eyes are certainly not precise measuring instruments. And taste-robbing oxygenation with O_2 is a largely invisible process. What's an engineer to do?

Finding virtual answers to real-world questions

Realistic simulation—the 3D computer representation of an object, the forces that can impact that object, and the resulting stresses and strains over time—has provided Coca-Cola's engineers with important answers. Their Virtual Packaging Development System is based on Abaqus finite element analysis (FEA).

Using FEA, Coca-Cola engineers have developed a 'virtual bottle model' that they can modify on their computer screens into whatever shape or material they want, depending on the type of beverage it will be



Abaqus software example of the mass diffusion FEA method in an oxygen concentration analysis.

filled with and the manufacturing process that will be used to make it. They can then simulate the effects on the bottle of stacking, crushing, dropping, and sloshing to prove out their designs, quickly and cost-effectively modifying the shapes to make bottles lighter, thinner, stronger, and so on. Having validated their computer models with real-world tests, they now have a 'library' of highly reliable simulations they can use to perfect existing designs and shorten time-to-market for new ones as products, consumer preferences, and industry regulations change.

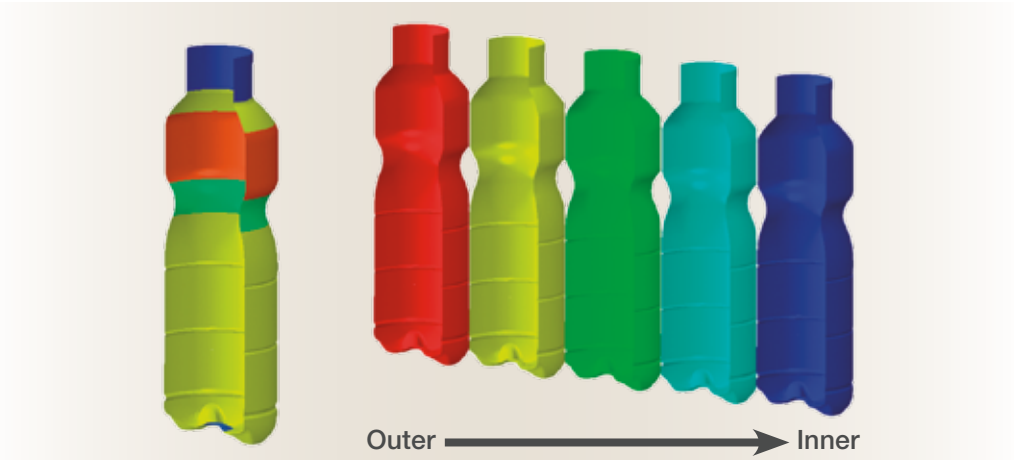
How do you simulate what's invisible?

For a design engineer, it's fairly straightforward to use FEA to simulate a plastic soft drink bottle hitting the floor. But what about predicting how tiny, invisible molecules of CO_2 or O_2 gas migrate through the walls of that bottle? The engineers at Coca-Cola Beverage Co. Ltd's Global Innovation & Technology Centre (GITC) in Shanghai decided to try. And thanks to the unchanging laws of physics, and some pretty cool capabilities in their software tools, they've succeeded.

The team started with the oxygen problem. How much O_2 passes into a beverage bottle through its PET wall per day? A simulation of this O_2 transmission rate (OTR) phenomenon needed to consider the effects of the bottle's geometry, the 'thickness profile' of the PET wall (thicker or thinner in different places depending on the manufacturing process and the curves of the bottle), and the material characteristics of each 'zone' of bottle thickness (where crystallinity, diffusivity, and solubility can vary).

"We investigated two different ways to tackle this problem with Abaqus software," says Dr. Simon Shi, senior packaging engineer at GITC. His team first applied the mass diffusion procedure, which simulates the movement of a fluid through a solid over time. Employed in such diverse industries as electronics and energy, it can be applied to everything from moisture absorption in the electronics chip of a phone to hydrogen embrittlement (gas migration through metals) inside a nuclear reactor. In the case of a PET bottle simulation, the 'fluid' moving through the solid is the oxygen.

The mass diffusion procedure starts with an FEA model of the bottle that incorporates the material properties of PET and the as-manufactured wall thickness profile,



The heat transfer method is an alternative analysis available in Abaqus.

"Realistic simulation gives us confidence that we will always be able to cost-effectively provide product quality to our customers anywhere in the world market."

Dr. Simon Shi, Senior Packaging Engineer, Global Innovation & Technology Centre, Coca-Cola Beverage Co. Ltd

previously determined from physical tests. A pressure gradient is loaded into the model to set the starting oxygen concentration inside (lower) and outside (higher) the bottle. When the diffusion simulation is run, the flow of oxygen passing through the bottle wall (always from higher to lower concentrations) appears on the computer screen as a moving rainbow of changing colors (blue is lower, red is higher). The flow will be higher across the thinnest areas of the bottle where there is less plastic for the oxygen to get through, and it will also be affected by the material properties, particularly the cystallinity. "The mass of O_2 that builds up on the inside wall of the bottle over time is what diffuses into the liquid to affect taste and freshness," says Dr. Shi.

Another FEA technique the engineers considered was Abaqus' heat transfer method. This is commonly used in the automotive industry to study thermal performance of powertrain assemblies, and in the electronics industry to analyze heat and power cycling of components. "The governing equations used to solve the O_2 question are the same for heat transfer as for mass diffusion," says Dr. Shi, "but in this case we were looking at the conduction of temperature (instead of the change in concentration of gas) from one side of the bottle to the other."

With the heat transfer method, the team could use a different type of 3D element as the building block for their models. Elements, used by the hundreds of thousands in an FEA model, are the mathematical units that describe the object being analyzed. "We found that the 'shell' elements available in the heat transfer method were more efficient," says Dr. Shi. "They use less computational time than the solid 'hex' ones used in the mass diffusion procedure and gave us slightly more accuracy in this particular case." As they ran their models, the engineers monitored the heat flux on the inner surface of the bottle, which varies over time due to PET wall thickness and material properties.

The engineers did some post-processing of their results from the heat and the mass simulations and found that predicted O_2 flow rates from both methods came very close to real-world measurements of about 0.04 milliliters of oxygen passing from outside to inside a bottle per day. "This doesn't sound like much on a daily basis," says Dr. Shi, "but if that bottle sits on the shelf for too long, those milliliter fractions of oxygen will mount up and impact beverage quality." In a bottle of juice, for example, each 1 mg of O_2 that gets into the solution can consume 11 mg of Vitamin C, depleting the nutritional value of the drink

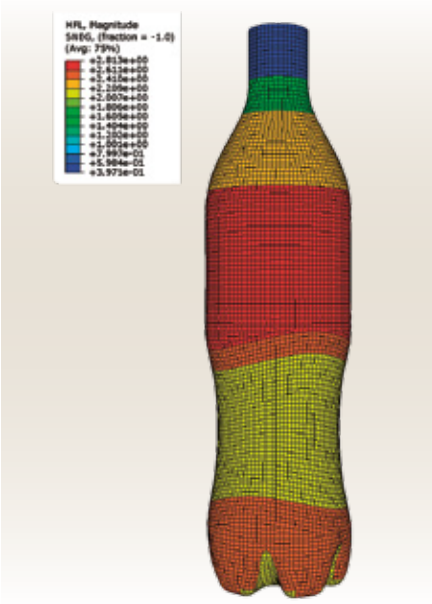
Continued

over time. Knowing now what the oxygen rise rate is—and how to simulate it in their virtual bottle models—Coca-Cola engineers can optimize PET bottle shape, wall thickness, and manufacturing processes to minimize it.

That bottle sitting on the shelf is actually moving

The second challenge, simulating CO_2 loss, required more steps to solve since the physics of pressure change are more complex than those of oxygen concentration. Freshly bottled Coca-Cola has an inside CO_2 pressure of 60 psi (about the same inflation as a compact spare tire). This provides that satisfying whoosh sound when opened as well as those tingles on the tongue and down your throat as you drink. When the pressure drops to 78.6% of the initial value the liquid goes "flat." The question is, how long does it take for the drop to occur, and therefore, what is the shelf-life of the product?

To complicate the picture, pressure loss is not due solely to CO_2 leaking out as oxygen leaks in. The bottle itself is also expanding slightly, or 'creeping,' over time due to the internal pressure. While creep is a natural characteristic of many materials, in a plastic soda bottle it causes the total volume inside the bottle to gradually get bigger, resulting in a secondary drop in pressure. Lower pressure inside the bottle allows more CO_2 to escape from the beverage into the empty space between liquid and cap. The result: fewer bubbles in the drink itself.



An example of a CO_2 loss rate analysis in Abaqus. A greater proportion of the gas passes through the thinner areas of the bottle.

To make their simulations of creep as realistic as possible, the engineers began with a laboratory test of a bottle containing some dry ice (frozen CO_2), which would evaporate to create the target pressure of 60 psi. Then they measured how much the bottle dimensions crept over one week, while at the same time tracking the CO_2 loss rate. "Starting from these real-world tests gave us highly accurate values for crept-bottle shape, volume, and pressure that we could build into our virtual bottle model," says Dr. Shi.

"Simulating these processes takes about two hours—much less time than with physical measurements recorded over a week in the laboratory."

Dr. Simon Shi, Senior Packaging Engineer, Global Innovation & Technology Centre, Coca-Cola Beverage Co. Ltd

Now the team was ready to run a heat-transfer method CO_2 analysis on their virtual bottle, similar to what was done with the oxygen simulations. Of course this time the pressure gradient was set up in reverse from the oxygen flow model, with higher levels set inside instead of outside, to reflect the CO_2-charged state of the bottle. Plugging the crept-bottle measurements into their model, the engineers could then run their analyses to predict both pressure and volume changes over time.

"Simulating these processes takes about two hours—much less time than with physical measurements recorded over a week in the laboratory," says Dr. Shi. "And O_2 flow simulations only take a couple of minutes. Our FEA models now predict the flow of both carbon dioxide and oxygen gases in PET bottles very reliably. This gives us a more complete picture of what will happen to our products in the real world and provides an efficient way to evaluate bottle performance at the early stages of design."

Proving out new concepts and delivering beverage quality

The Coca-Cola engineers see continuing value in their new methodologies going forward. "Bottles are getting lighter and thinner in response to economic and environmental realities," says Dr. Shi. "We now have a validated methodology in Abaqus that we can use to prove out new concepts in shape and material thickness.

"We plan to explore the behavior of other materials besides PET, such as plant-based plastics, in the future. Realistic simulation gives us confidence that we will always be able to cost-effectively provide product quality to our customers anywhere in the world market."

The history of 'soda pop'

The inspiration for 'soda pop' can be traced back to ancient times, when people thought bathing in naturally carbonated mineral springs could promote health and cure diseases. If dipping in such water was good for you, then shouldn't drinking it be even better?

That's what Joseph Priestley was thinking when he created the first man-made, drinkable glass of carbonated water in 1767. Experimenting with gases in his local brewery, he discovered that distilled water suspended over a fermentation vat became infused with carbon dioxide, taking on a tangy, bubbly taste. Others learned how to directly add CO_2 to water, as either dry ice or a high-pressure liquid, and then began mass manufacturing the increasingly popular beverage.

Flavored syrups were refined by chemists in the late 19th century, and the soda fountain blossomed as a community center where people went to socialize and enjoy a fizzy beverage. Some pharmacists' syrup formulas (like Coca-Cola's secret one) developed into brands still known today. The advent of bottling factories made soft drinks portable and storable and by 1920 over five thousand bottling companies were registered with the U.S. Census Bureau. Consolidation means that fewer major brands survive today—but soft drinks remain as popular as ever all over the world.

Getting to Zero...with the Abaqus Knee Simulator

Accelerating the design of knee implants



The Abaqus Knee Simulator provides orthopedic device manufacturers with the tools necessary to deliver innovative, safe, and effective knee implant designs to patients in a fraction of the time.

IMPROVE
design confidence and reliability

It is estimated that nearly 1.5 million knee arthroplasty (replacement) surgeries will take place globally in 2013. As this number continues to rise, orthopedic device companies involved with designing, testing, and manufacturing knee implants will be expected to accelerate design cycles and patient care to meet the increasingly demanding market needs.

The Abaqus Knee Simulator vs. physical testing

In the past, orthopedic device manufacturers have relied heavily on numerous in vitro tests to assess the performance of a device based on new design parameters. However, the evaluation of orthopedic implants, such as total knee replacement (TKR), under physiological loading conditions has been difficult to achieve using these traditional testing methods. Physical simulators, such as the Kansas Knee Simulator, have required a cadaver knee joint and a complex loading apparatus to mimic the in vivo conditions. This type of testing is too expensive and time consuming to be a practical device evaluation tool.

The Abaqus Knee Simulator was developed to address this issue and provide knee implant designers with five easy-to-use workflows to explore a variety of different designs in less time and at a fraction of the cost. More importantly, the Abaqus Knee Simulator is based on validated model data, so the designs will be safer and more effective for patients.

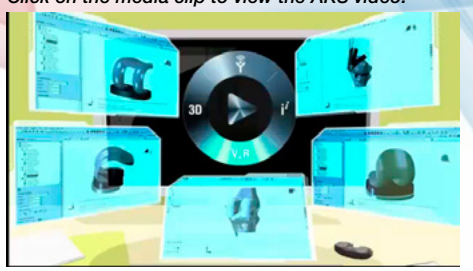
Workflows

The Abaqus Knee Simulator suite of five semi-automated tools perform a series of knee simulations of varying complexity, from simulating basic contact to general activities of daily living, all which address:

1. **Contact Mechanics:** Evaluate tibiofemoral contact mechanics at static positions throughout flexion
2. **Implant Constraint:** Measure laxity between femoral and tibial components in the absence of soft tissue structures
3. **Tibiofemoral Constraint:** Measure laxity of the tibiofemoral joint with soft tissue
4. **Wear Simulator:** Predict wear on the tibial insert over a number of gait cycles
5. **Basic Total Knee Replacement Loading:** Evaluate whole joint mechanics during activities of daily living under basic muscle loaded conditions

The five simulation workflows were developed with a custom interface to allow easy conversion from rigid to deformable structures, adjustment of mechanical properties and component alignment, editing of material properties and boundary conditions, and presentation of simulation results.

Click on the media clip to view the AKS video.



Designer and analyst applications guide users from model creation to results interpolation.

For More Information

www.3ds.com/aks

Keeping Carbonation Bottled Up with FEA

Verallia, Saint-Gobain's Packaging Sector, uses Abaqus simulation and Isight automated optimization to help reduce bottle weight while preventing breakage



When Dom Pérignon—French Benedictine monk, winemaker, and namesake for fine champagne—was asked to look into why sparkling wine bottles were unexpectedly bursting in the cellars of his abbey in northern France three hundred years ago, he decided to investigate the bubbles. Fermentation, it turns out, continued after the beverage was bottled, producing additional carbonation and increasing pressure. By experimenting, so the story goes, Pérignon was able to control the refermentation process and reduce breakage and loss of the bottles' precious contents.

Today, research engineers working for Verallia at Saint-Gobain's Research Center, near Paris, take a different tack to ensure there is no breakage: They analyze the bottles, not the bubbly. Xavier Brajer, a mechanical engineer responsible for the 15-person Mechanics of Materials Group in the company's R&D department, leads these efforts. Central to the group's analyses are initiatives—from the government, wine and beverage industry, and Saint-Gobain itself—to reduce the quantity of raw materials used in the bottle-making process. "In order to reduce our impact on the environment, we want to minimize the materials and energy used," says Brajer, "and at the same time guarantee that the bottles have mechanical properties that will maximize their lifetime."

Glass bottles are made from four readily available sources: silica or sand, soda ash, limestone, and cullet (recycled glass). Coloring agents are also added, the specific color dependent on the beverage being bottled and the customer's wishes. An average empty 750 milliliter wine bottle weighs about 500 grams (some are as light as 300 grams), accounting for about 30 to 40 percent of the bottle's weight when full. A champagne bottle weighs about twice that amount. The sparkling beverage's heavier, thicker-walled container is required because of the pressures produced by its signature carbonation: reported to be as high as 90 pounds per square inch or approximately three times that of a typical car tire. While champagne produces the highest gas pressure of any beverage, every carbonated drink—including hard cider, soda, and other sparkling wines—creates internal stress on the glass that needs to be considered when designing lighter bottles.

"In the past, it might have taken us a week to run the 100 simulations needed for this optimization. But now with an iterative-looped process, it only takes about an hour."

Xavier Brajer, Mechanical Engineer, Verallia R&D, Saint-Gobain

Exploring bottle shape and strength with simulation and optimization

Despite the fact that beverage containers come in an almost unending variety of shapes, sizes, and colors, the profile of a champagne bottle is easily recognizable from across the room. Even the heft of the container is part of its perceived quality. Other bottle shapes are also closely associated with specific beverages or brands. So changing a bottle's shape to reduce weight and materials treads into sensitive territory. Given that, when it comes to making these changes, Brajer and his R&D team consider the subjective aesthetic factors. But they do so while relying on objective engineering tools.

"We start with a CAD model of the bottle shape that has been drawn by Verallia's design department, in agreement with the customer," says Brajer. "Then as we try and reduce the weight of the bottle, we use Abaqus finite element analysis (FEA) to simulate stresses and couple that with Isight software to optimize the geometry so that the container will withstand those stresses without breaking." Saint-Gobain has been using Abaqus for about 15 years, and started using Isight for automated and integrated simulation process flows about two years ago. "These two software packages link together easily and allow us to run a series of calculations that save time while leading us to the optimized bottle geometry," Brajer adds.

In a recent optimization analysis, the Saint-Gobain research team tested a lightweight design for a hard-cider bottle. Like champagne, this carbonated alcoholic drink creates internal pressure loading that is most likely to rupture the bottle at its weakest point—the bottom. So while trying to reduce the overall amount of material, it is on this region that Brajer and his team focused their engineering analysis, looking to maintain its mechanical resistance and strength.

The team started the pressure analysis by first creating a model of a reference bottle in Abaqus: A 2D model utilizing glass' basic material characteristics was used to take advantage of the bottle's axisymmetric geometry and save computing time (see Figure 1). They then meshed the model and applied boundary conditions and loads. To optimize the geometric parameters (in this case nine, but sometimes more) that were used to describe the bottle's bottom—such as internal and external shape, curvature, and a number of different radii—Abaqus was coupled

with Isight. This allowed the team to automate the simulation workflow and systematically make changes in the parameters, calculate stresses for each profile, and ultimately determine minimal stress and optimal bottle shape (see Figure 2). "In the case of the hard-cider bottle, we were able to reduce weight by 10 percent and stress on the critical bottom region by about 17 percent," says Brajer. "In the past, it might have taken us a week to run the 100 simulations needed for this optimization. But now with an iterative-looped process, it only takes about an hour."

With the help of Verallia's technical teams, physical testing was also utilized to complement simulation. "How we use testing may change in the future," he adds, "because once we have calculations that validate the tests, design engineers will trust the virtual optimization process."

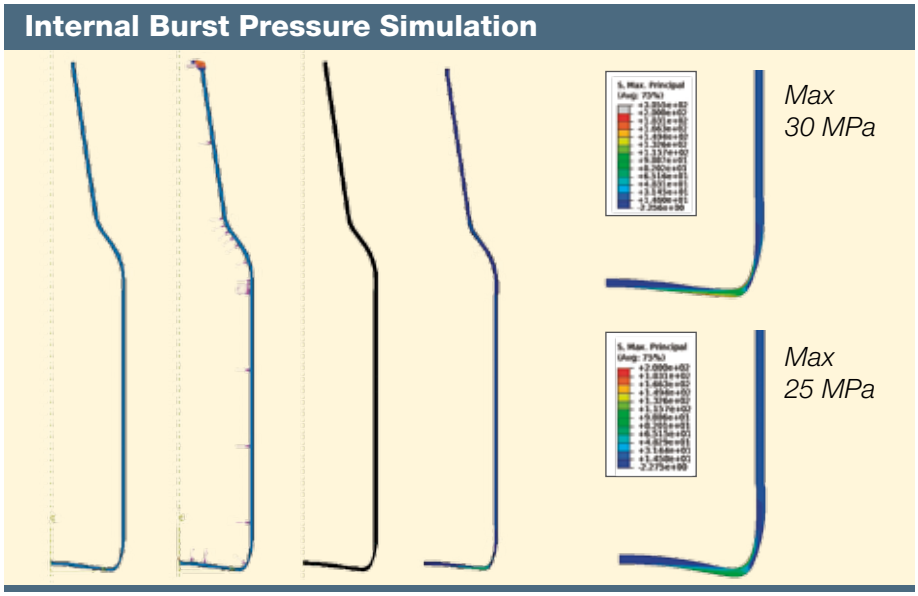


Figure 1. This simulation of internal burst pressure for a hard-cider bottle using Abaqus FEA shows (left to right for half-bottle models): geometry generation of the axisymmetric model; application of carbonation pressure loading and boundary conditions; meshing of the model; and analysis of stress results. In the detailed view of the bottle-bottom profile, internal pressure loading has been reduced from Max 30 MPa (top detail) in the original bottle to Max 25 MPa in the modified/optimized version (bottom detail).

Simulation helps reduce bottle weight

Internal loading pressure is one of the most common causes of breakage for carbonated beverage containers. But it is far from the only one. As bottles move from the glass-making mold to the bottling assembly line and then finally to the store shelf—whatever the beverage or bottle—they are filled, capped, stacked, and transported. This subjects them to a variety of loading scenarios including thermal stress (from hot liquids), impact, squeezing, and compression. Basically, the life of a glass bottle can be a precarious one.

Brajer and his team are working to change that fact. So while the pressure analysis was the first optimization study that the engineering team conducted, they intend to use a similar process—and the same software tools—to investigate the other ways that bottles break. In future designs, using fewer raw materials will remain of primary importance to the team. That's one of the goals of Verallia R&D.

This design strategy is critically important today given the need to reduce CO_2 emissions from manufacturing processes. "Our goal is to use less energy to produce each bottle," Brajer says. "If we reduce raw materials, we reduce the energy needed to process those materials and to melt and form the glass. The amount of CO_2 created in the process, in turn, is directly linked to the energy and raw materials usage."

Given the quantities of bottled beverages and food consumed worldwide, the carbon footprint for bottle-making is worth addressing. Verallia, Saint-Gobain's Packaging Sector, is the second largest glass bottle and jar manufacturer in the world—with a yearly production of roughly 25 billion containers. Given that, it's easy to see how their design strategy to optimize weight could provide significant environmental benefits—plus champagne bottles that would have made Dom Pérignon proud. We can all toast to that.

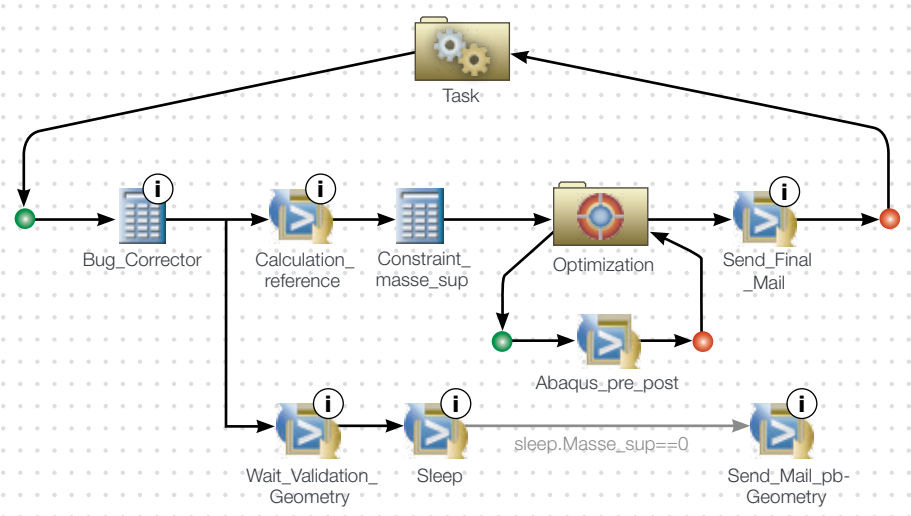


Figure 2. This Isight workflow, with an embedded Abaqus FEA loop, allowed the Verallia design team to systematically optimize bottle shape, making slight changes in the bottle's geometry parameters in each iteration. The automated process shortened analysis time from one week to one hour.

For More Information

www.saint-gobain.com

Novo Nordisk Makes Designing Injection Pens a Snap (Fit)

Leading innovators in diabetes care use realistic simulation to improve product integrity from design to manufacture



Many medical conditions can be treated with tablets, but others require injections under the skin in order for therapeutic drugs to reach the bloodstream. In the case of insulin administration for diabetes treatment, patients need to self-inject the drug daily.

Making those injections easy and safe is of prime importance for Novo Nordisk, the Danish company that has been a world leader in the production of insulin ever since it was discovered by Canadian scientists in the 1920s. The company innovated beyond standard syringe technology to produce the world's first patient-friendly self-injection system, the NovoPen, some 25 years ago.

With more than 350 million diabetics worldwide—8.3 percent of the global population and growing, according to the International Diabetes Federation—demand for insulin pens will likely remain strong into the foreseeable future. Since effective control of the disease is dependent on consistent use of the drug, these delivery systems need to be portable, easy-to-use, reliable, and even resistant to minor misuse by patients.

Small medical device, big design task

An insulin pen may be small, but it is a precision instrument with a number of complex parts that must work in perfect tandem. Some pens are durable, containing a replaceable drug cartridge, while other disposable ones come pre-filled with the drug.

"Having material models incorporating time-dependent viscous behavior is very important for our work and we're now able to simulate both creep and relaxation with Abaqus."

Torben Strøm Hansen, Principal Scientist, Device R&D division, Novo Nordisk

Injection typically involves twisting a short needle onto the pen, turning a dial to the required dose, and pushing a button to deliver the medication under the skin. After a given number of doses are injected, the cartridge is exchanged for a new one (with a durable device) or discarded (with disposable pens).

Audible clicks that occur at key stages of this procedure reassure the patient that they are engaging the device correctly at each step. It looks pretty easy. A one-minute video of a woman checking her blood sugar and then using a NovoPen to inject insulin is available here: **http://www.novonordisk.com/press/broadcastroom/default.asp**. But every one of those reassuring clicks represents a challenge that has been overcome by the engineers who created the pens. So do the clicks the patient never hears: those that occur as the pen parts are assembled in the factory before use.

"Parts that click into place with 'snap fit' instead of screw connectors are very efficient to assemble within mass production," says Torben Strøm Hansen, principal scientist in the Device R&D division of Novo Nordisk, near Copenhagen, Denmark. "Snap fit is the commonly used way to connect parts in our device mechanisms, and it also signals reliability when the internal components have optimal connections that don't rattle. It's very efficient when designed correctly."

Getting those designs correct from the start is the task that Hansen and his Mechanical Analysis team focus on in close collaboration with Novo Nordisk's mechanical designers. "Even though an injection pen is not that big, there are a lot of fine details in its design," he says. Whatever the configuration of device, the plastic-polymer components must withstand the rigors of both manufacturing and patient use, performing as required at different temperatures and loads.

Teaming up to model polymer behavior

To ensure the integrity of their designs, Hansen and his team in the Device Simulation department rely on computer simulation with Abaqus finite element analysis (FEA).

"More than a decade ago, my colleagues and I explored a number of commercial software codes," he says. "We chose Abaqus because it was a well-integrated solution that provided both implicit and explicit capabilities, and could model the nonlinear behavior of the fine details in our designs correctly, including the high number of interfaces in contact."

Over time, the group's device models have become more refined, sophisticated, and computationally demanding. The SIMULIA Polymer Customer Review Team (PCRT) has worked closely with Novo Nordisk all along to provide updated enhancements in Abaqus that enable the company to model

and predict the complexities of polymer behavior with increasing accuracy and efficiency. (The PCRT includes members from the automotive, high-tech, life sciences, and consumer goods industries.)

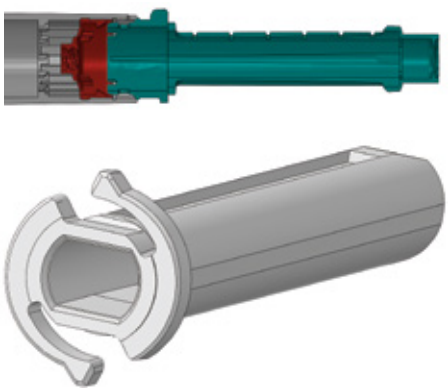
The snap-fit challenge

A recent focus on snap fits in insulin pens demonstrates the challenges the team has faced when modeling polymers. "We concentrated on snap fits because they demonstrate almost ideal cyclic loading, with parts repeatedly loading and unloading from single to multiple cycles," says Hansen.

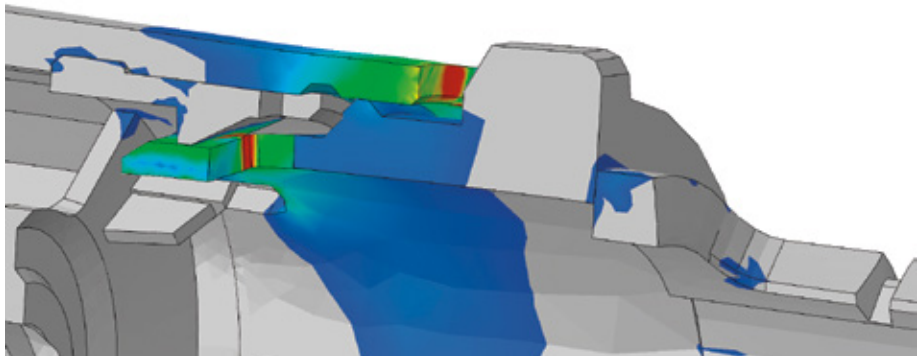
During such cycles, the viscous nature of the thermoplastic material determines how the bouncing back to 'normal' occurs. Prediction through analysis of such time-dependent behavior is key to the device development process.

Since devices can be subjected to different environments, including elevated temperatures, the function of the device must be as unaffected as possible by such changes and always comply with the specification even though the material properties of the components vary. Even just sitting on a pharmacy shelf or in the medicine cabinet, polymer materials are prone to creep and relaxation over time at rates that can vary with the temperature. Some polymers are also more complex than others: those used in durable devices may contain carbon or glass fillers that show anisotrophic behavior, which can be hard to predict.

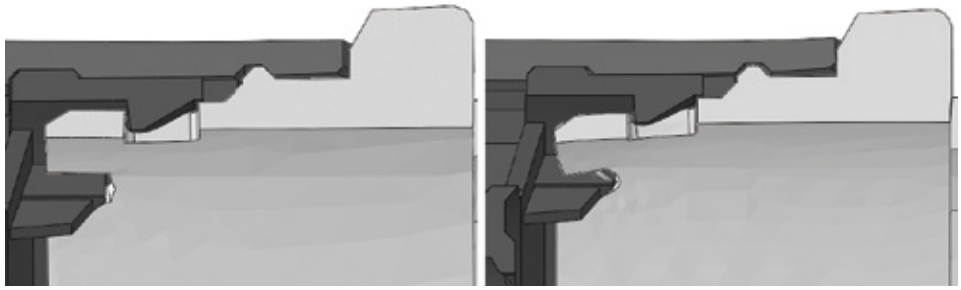
"Modeling these diverse material characteristics as well as the behavior of the polymer as the load induces larger strains closer to yield is difficult," says Hansen. "In order to predict such viscoelasticity precisely, we needed a more refined model that goes beyond a mainstream elastic-plastic approach."



(Top) CAD image of diabetes pen components. Grey and red cylindrical parts are snapped onto the green. (Bottom) An injection-molded ratchet component from a medical device used by Novo Nordisk for a benchmark study.



FEA analysis demonstrates the ultimate straining of the snap fit on the device components during assembly.



Final snap deformation model (left) in Abaqus FEA (using a parallel framework to capture the changing behavior of the polymer material under cyclic loading) shows a more accurate plastic deformation of 0.12mm. The earlier elastic-plastic model (right) over-predicted that deformation would be 0.66mm.

Modeling material that's constantly changing

The team is now using the 'parallel rheological framework' methodology available in Abaqus to model polymer nonlinear viscoelasticity with greater accuracy than ever before. The framework makes use of an arbitrary number of viscoelastic networks and an elastic equilibrium network to create a specific nonlinear viscoelastic model that is used to predict and track changes in the internal structural networks of a polymer as the material responds to repeated cyclic loads during snap fit. The material parameters in the FEA model are updated at each time step to reflect the new, altered state of the polymer. Since every type of polymer shows a different response to temperature, load, etc., the team continues to explore ways to identify the material characteristics of different polymer networks.

Not only are such advanced models useful to designers fine-tuning the latest pen configuration, the data can help inform manufacturing processes in the factory.

"We have a process-simulating capability, through Moldflow, for which Abaqus has an interface. This allows us to input the stress fields that result from the molding process right into our models," says Hansen. "As a result, we have greater insight into our manufacturing process and are more able to design parts that have a very low level of residual stresses in critical regions.

"SIMULIA is working closely with us to provide capabilities we need," says Hansen. "Having material models incorporating time-dependent viscous behavior is very important for our work and we're now able to simulate both creep and relaxation with Abaqus. We are investigating how well the model will adapt to different kinds of thermoplastics, which may require different networks. Calibration will be key going forward."

For More Information

www.novonordisk.com
www.3ds.com/SCN-February2013

CONNECT. SHARE. INNOVATE.



SIMULIA COMMUNITY CONFERENCE

May 21–24, 2013

Vienna, Austria

Social sites, blogs, tweets, webinars, podcasts—the list of resources to gain knowledge is ever-growing. While online resources provide easy access to information on current technology or industry news, there is no substitute for the significant value of meeting with colleagues and peers face-to-face. The annual SIMULIA Community Conference (SCC) provides this opportunity, bringing users together from all over the world to put faces to names, make personal friendships, and create a shared experience.

Our users' commitment to presenting their strategies and applications is the main reason for the ongoing success of the SCC. This year's 100+ abstracts from the aerospace, automotive, electronics, and energy industries, as well as others, have been reviewed and selected to uphold the SCC's reputation for outstanding quality. The user presentations provide real-world examples of what can be accomplished using SIMULIA solutions, and we hope they will offer ideas and inspiration to attendees to further their maturity level of realistic simulation in their organization. The sidebar has a sampling of some of the abstracts received. Visit our website to see the full list along with the abstracts.

We are also pleased to announce that our keynote presenters this year will be **BMW Group** and **Ethicon Surgical Care, Inc., a Johnson & Johnson company**:



Rudolf Blaim
Department Manager Requirement Management and Verification in Process IT Idea to Offer
BMW Group



David Smith
Principal Design Engineer
Ethicon Surgical Care, Inc., a Johnson & Johnson company

The conference will offer plenty of opportunities to network with SIMULIA executives, industry and technology experts, developers, Alliance Partners, and fellow members of our user community. Prior to the event, we will offer an online community for conference attendees to see who is registered, get an early look at the conference proceedings, and make plans to connect with other users. Onsite, attendees can participate in evening networking receptions, special interest groups, and have discussions during breaks and lunch. We hope you can join us in Vienna, Austria, this May to share ideas, keep up with emerging technologies and trends, network with peers, and be your own "voice of the customer" to all of us at SIMULIA.



A sampling of papers submitted to the 2013 SCC:

ABB Corporate Research Center Germany
Absolute and Relative Phases in Twin-tube Structures and Performance Criteria for Coriolis Meters

Airbus
Tire Debris Impact Modeling on a Composite Wing Structure

Ansaldo Energia
DOE Analysis for Internal Cooling Configuration of Gas Turbine Blade

Baker Hughes
Validation of Abaqus Virtual Simulation Model for an Expandable Liner Hanger

BorgWarner Turbo Systems Engineering GmbH
Damage Assessment of Casting Materials in High-temperature Applications Influenced by Varying Plasticity Models

E.ON Anlagenservice GmbH
Enhancing Operational Flexibility of Fossil Power Generating Assets Based on Nonlinear Numerical Thermomechanical Damage and Fracture Analysis

Fokker Landing Gear B.V.
Prediction of Damage Evolution in Composites Using Abaqus

GM Powertrain
The Use of Optimization Software in a Standard Flexplate Design Process

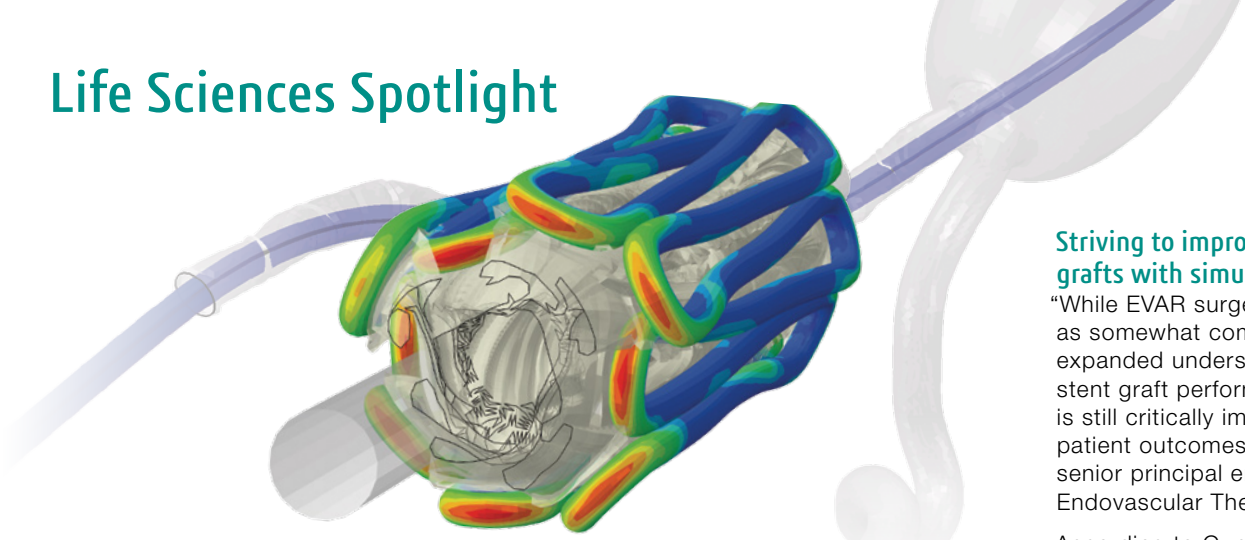
Pirelli Tyre S.p.A.
Pirelli Formula 1 Tire Modeling Application with Abaqus

Rolls-Royce Deutschland Gmbh
Collaborative Robust Engine Optimization

SKODA - JS a.s.
Identification of Ductile Damage Parameters

Tetra Pak
Simulating the Pouch Forming Using a Detailed Fluid-structure Interaction

For More Information
www.3ds.com/simulia



Improving Stent Graft Designs—Without Missing a Beat

Medtronic explores design space for aneurysm treatment devices using Abaqus FEA and Isight

The heart pumps blood through the 60,000-mile-long human circulatory system 60 to 80 times each minute. The body's core blood vessel, the aorta, must carry oxygenated blood from the left ventricle of the heart all the way down to the abdomen. As a result, aortic walls are under pressure from the blood's constant pulsing. Over time, age stiffens the vessel's walls and disease can create additional stress. Apply these cumulative loads over the years and it's no surprise that the body's major artery can be prone to problems.

Damage to the aorta can result from atherosclerosis (clogged vessels), smoking, diet, or heredity. When vessel walls weaken, an aneurysm (or bulge) can form that allows the blood to pool, impairing normal circulation. If left untreated, the aneurysm can eventually rupture with potentially life-threatening consequences.

Originally, the only way to repair an aortic aneurysm was through open surgery. But with the first successful implant of a stent graft in the early 1990s, doctors and patients had an effective and less invasive treatment option available.

Stent grafts are tubular wire-mesh structures with a membrane-like fabric covering. They function like an alternative piece of piping, channeling blood flow so that it bypasses the weakened section of the artery. (Stents, on the other hand, are wire mesh tubes without a covering that are used to physically expand a constricted vessel.) Stent grafts are now frequently used in both thoracic (above the diaphragm) and abdominal aortas in a procedure called an endovascular aneurysm repair (EVAR). It's reported that as many as 45,000 abdominal aortic aneurysm surgeries are performed each year (see Figure 1).

Striving to improve stent grafts with simulation

"While EVAR surgeries are now seen as somewhat commonplace, an expanded understanding of in-situ stent graft performance and delivery is still critically important for improving patient outcomes," says Atul Gupta, senior principal engineer on Medtronic Endovascular Therapies' R&D team.

According to Gupta, as a surgeon inserts the stent graft in its catheter, they can encounter difficult anatomy, calcified vessels, or an aneurysm site that's angulated. Once the device is in place, it must provide a good seal with no endo-leaks, and it can't migrate away from the site as a result of the constant sheer stress due to blood flow. It also needs to withstand the structural fatigue originating from pulsatile loading for a minimum of ten years—or about 400 million cycles total—to satisfy regulatory and ISO standards requirements.

To fully understand these complex structural behaviors and fine-tune stent grafts to accommodate them, Gupta's California-based group employs computational modeling with Abaqus finite element analysis (FEA). "The Endovascular Therapies group at Medtronic has been using Abaqus in-house for more than six years now," says Gupta. Of particular importance to his team are its advanced contact capabilities, scripting interface for customizations and the fact that it's an excellent modeling tool for Nitinol, the metallic alloy from which these stents are made.

"The medical space is highly regulated, so the quality assurance and ISO certification of the software is also

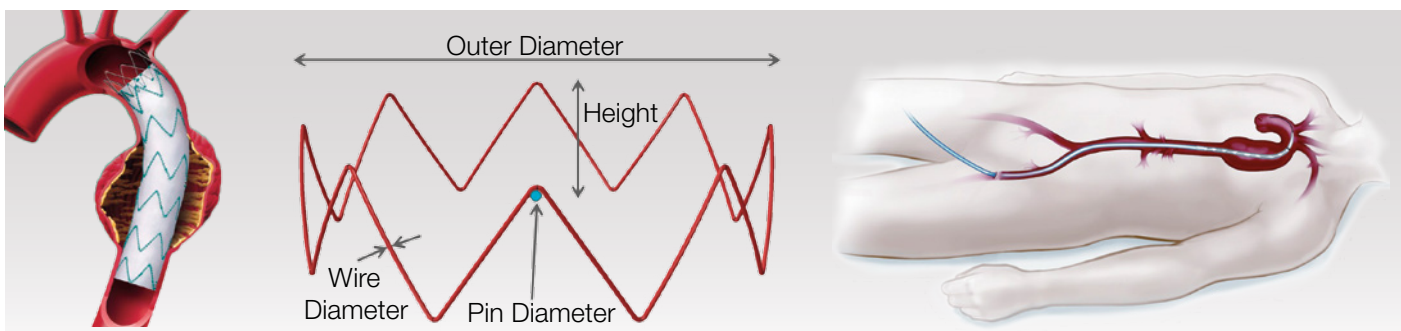


Figure 1. This cutaway view of a thoracic aorta (left) shows a stent graft that has been deployed at an aneurysm site so that blood can bypass the bulge in the vessel walls. The detail view of the stent rings (center) identifies the key stent graft design parameters. The device is inserted using a minimally invasive catheter-based procedure called an Endovascular Aneurysm Repair (EVAR) (right). Medtronic utilizes Abaqus FEA and Isight workflow automation to simulate stent graft and surgical delivery-system behaviors.

"Now the product development cycle is much more efficient because we run numerous simulations first and then choose the design that is optimal—before we make anything."

Atul Gupta, Senior Principal Engineer, Medtronic Endovascular Therapies' R&D team

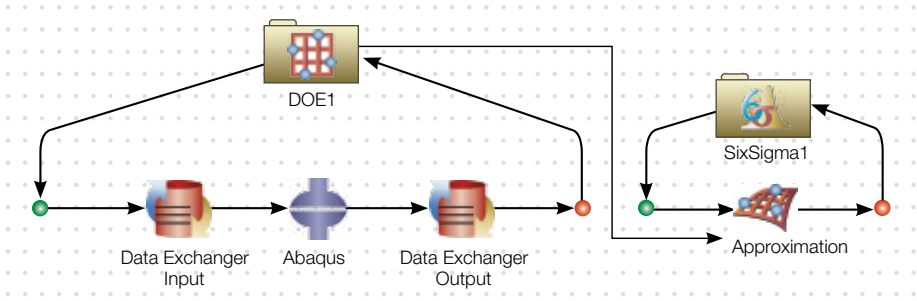


Figure 2. This simulation process-automation workflow in Isight shows (on the left) the iterative loop for a Design of Experiment (DOE) that included an Abaqus FEA analysis and (on the right) the probabilistic evaluation that helped in evaluating the reliability of the stent design when parameters were varied within tolerances. The workflows are created in Isight using a simple drag-and-drop process.

very important," says Gupta. Verified and validated simulation models can provide medical device designers, as well as regulatory agencies, with highly accurate assessments of performance, reliability, and durability. When modeling stent grafts, for example, FEA has allowed Gupta's team to look closely at radial strength, peak stresses and strains, and fatigue safety factors of the metallic stent rings.

Earlier exercises in stent modeling typically included some necessary simplifications to keep the analyses manageable. "In the past, we created models with nominal dimensions and material properties to save computational time and analyst effort," says Gupta. Manufacturing variances were often not included in analyses, though fabrication relies on considerable hand labor, and both material and dimensions could vary within tolerances. Additional variability—resulting from the often dramatically different in vivo boundary conditions seen in actual patient-specific anatomy—was also not part of standard methods. "In order to fully understand the variation of all factors and their influence on device performance, we knew that further investigation was required," says Gupta.

A single FEA model can provide a snapshot of one set of stent graft variables for one specific design at a time. An entire series of FEA results provides the researcher with a kind of 'animation' in which each subsequent image captures the behavior of a slightly different design with a slightly different set of design variables. Viewing the results as a sequence of snapshots allows engineers to more completely understand and explore the vast range of design-space options and solutions, and leads to a more robust design.

With this goal in mind, the R&D group decided to examine their Abaqus results through statistical analysis techniques that could be linked into an automatic workflow to save time. They chose Isight simulation process automation and design optimization software. "Isight is well integrated with Abaqus," says Gupta. "Its easy-to-use, drag-and-drop graphical interface provides access to a number of advanced probabilistic modeling routines. It also accommodates our internal tools."

Testing new automated workflows

To put their new methodology to the test, the R&D team started with a thoracic aortic stent ring and created a model for it in Abaqus/CAE. They used a rigid representation of the vessel and assigned superelastic material properties to Nitinol, determined from experimental data. To encompass as much design variability as possible, they then added real-world manufacturing tolerances for key stent parameters, including stent height (±10%), wire diameter (±10%), crown radius/pin diameter (±25%), outer diameter (-1% to +2%), diameter of the vessel (±2.5%), and material plateaus (±10%).

The analysis incorporated an automated Design of Experiment (DOE), which ran 200 iterations in Isight with Abaqus serving as the FEA solver (see Figure 2). For the FEA analysis, the engineering team subjected the model to a sequence of quasi-static loading steps. The outputs of most interest, according to Gupta, were crimp strains inside the delivery catheter during loading (important for avoiding permanent plastic deformation), the radial forces at deployment (which ensure proper contact of the stent graft to the vessel wall), and the safety factor of the stent under pulsatile fatigue loading due to vessel dilation. The team created a response surface approximation from these outputs (with errors less than one percent for all results), which were then combined with the expected manufacturing variability added earlier.

Continued

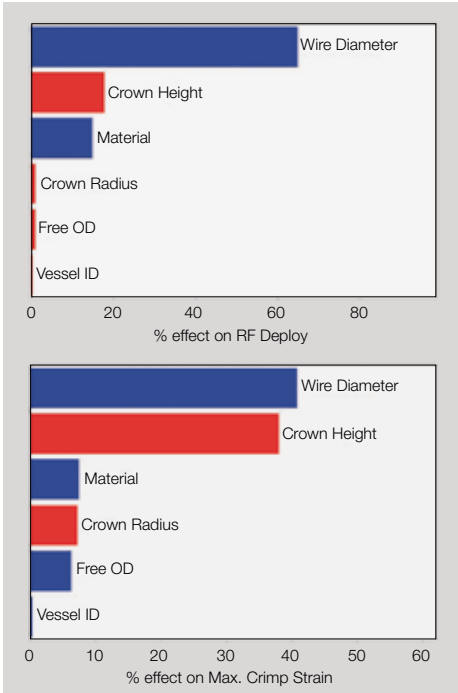


Figure 3. Computational modeling techniques in Isight software calculate sensitivity plots (Pareto distributions) for stent graft radial force (top) and crimp strains (bottom). In both outputs, the key parameters were determined to be wire diameter and crown height. Blue bars indicate a positive correlation and red bars indicate a negative correlation.

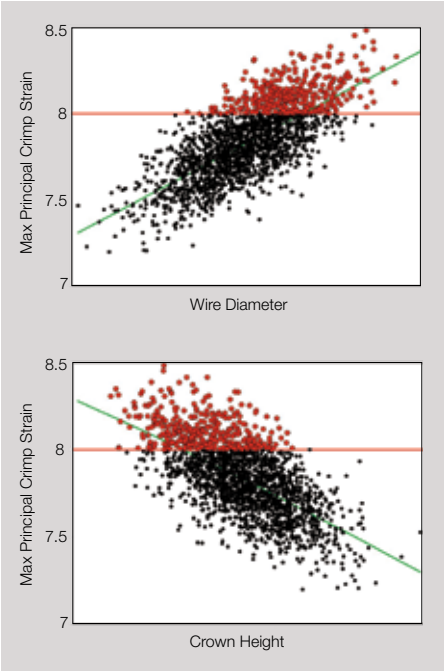


Figure 4. These scatter plots, produced as part of an Isight workflow, resulted from a probabilistic technique used in this case to show how crimp strain on the stent is affected by changes in wire diameter (top) and crown height (bottom). The green trend line indicates a positive correlation between crimp strain and wire diameter and a negative correlation of strain with crown height. Red data points indicate the values for which crimp strain surpassed the predetermined limit indicated by the red line.

"Next we ran probabilistic sampling analyses that included thousands of simulations and allowed us to change variables for all the key parameters to assess design reliability," says Gupta. The result was a statistical distribution of crimp strains, radial force, and fatigue safety factors. Tools in Isight, like Pareto charts and scatter plots, helped the team identify wire diameter and stent height as the most critical design parameters when trying to achieve target crimp strains (see Figures 3 and 4).

Isight includes a variety of Six Sigma probabilistic techniques—such as Monte Carlo, importance, and mean value sampling methods, among others. As part of the analysis, the engineering team was able to compare these methods to determine which could most accurately predict the probability of success and the tightest confidence interval.

To validate the various findings, the R&D team compared analysis calculations with bench test results derived from radial force testing. The fatigue performance of the stent was also confirmed using an acoustic-fluid testing protocol, which applied radial pulsatile loading on the stent graft. The experimental data, collected over three to six months, mirrored the ISO and regulatory requirement of bench testing for 10-year load cycle. "We found that our simulations and probabilistic methods accurately captured the inherent variability in stent performance and helped in coming up with a design that could survive in the in vivo loading environment," says Gupta.

The future of stent graft improvement

"Before we had these computational modeling tools, we practiced a make-and-break type of approach: We would fabricate some stent prototypes, do some quick testing, and then repeat the process," says Gupta. "Now the product development cycle is much more efficient because we run numerous simulations first and then choose the design that is optimal—before we make anything." The efficiencies of new workflow routines have allowed the team to cut the stent design cycle by an order of magnitude, Gupta adds.

Acceleration of time to market and identification of key design parameters are both major improvements in the Medtronic team's product development cycle. But Gupta feels there is much more that simulation has to offer. "So far we have only looked at the stent rings and some parts of the delivery system. All of the device's components and its delivery system can be independently optimized," says Gupta. "And the assembly, which is very complex, can also be analyzed as a whole. So we'll take it one step at a time."



Figure 5. This cutaway view shows a bifurcated abdominal stent graft used in abdominal EVAR interventions. Computational modeling is being used to predict the performance and delivery of these devices and has been instrumental in significantly shortening the design cycle.

"Ultimately, what we do is all about patient safety. Simulation helps us be much more certain of our designs."

Atul Gupta, Senior Principal Engineer, Medtronic Endovascular Therapies' R&D team

For next-gen devices, first steps already include using simulation to explore improved abdominal stent graft designs (see Figure 5). The R&D team is also conducting a study that examines multiple materials used to make a catheter and is looking at lowering its profile so surgical delivery is even less invasive. Future steps include exploring how best to model the unstable behavior of the polyester fabric membrane and stent-graft interactions with blood flow inside the vessel. For this, engineers are developing models that can capture instabilities in the material and a fluid-structure interaction (FSI) framework.

Currently the FDA accepts FEA results as a way to present the 'worst-case' model in a device design application. Results are always validated with physical testing prior to reporting. But the regulatory emphasis is evolving to include improved definitions about the in vivo loading environment and applicability of computational models to capture device behavior throughout their life cycle in-situ. And Medtronic is collaborating with the federal agency on refining these regulations.

"Ultimately, what we do is all about patient safety. Simulation helps us be much more certain of our designs," says Gupta. "Given the predictive power of available software tools and techniques, computational modeling is sure to become increasingly important for our industry in the future."

For More Information

www.medtronic.com
www.3ds.com/SCN-February2013

Predict Full-Field 3D Geomechanical Behavior of Petroleum Reservoirs

Safely and profitably extracting and processing oil and gas resources from deeper and more complex geological environments poses significant engineering challenges. In particular, predicting how the subsurface will behave and understanding the effect on the volume of oil and gas that can be extracted has become increasingly important. As a result, realistic simulation of the geomechanical behavior of petroleum reservoirs has become a fundamental part of the development life cycle. For this purpose, it is critical to integrate advanced simulation techniques properly with available field observations and laboratory tests.

As a new Abaqus Integration Partner, Baker Hughes (a leading supplier of oilfield high-performance drilling, evaluation, completions and production technology and services, integrated operations and reservoir consulting) has achieved significant advances in developing technology capable of creating full-field 3D geomechanical models. Their JewelSuite™ 3D GeoMechanics software provides

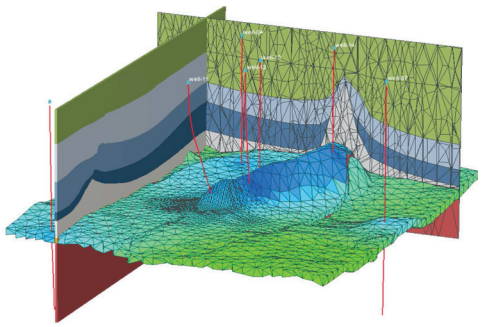


Figure 1. Wellbore planning optimization for a salt dome field

capabilities to integrate seismic, structural, geologic, fluid flow, and geomechanical models into a single, multidisciplinary workflow.

It is important to more accurately quantify complicated stress fields, effects from production, fault activation, wellbore stability, and surface subsidence. Consequently, Baker Hughes has integrated Abaqus with JewelSuite to provide a single modeling and simulation environment that simplifies and accelerates the analysis of reservoir geomechanics. Figure 1 illustrates a JewelSuite solution developed to optimize wellbore planning in a salt dome field to enable drilling that minimizes wellbore failure.

The JewelSuite workflow utilizes all available subsurface data to produce run-ready Abaqus simulation models seamlessly. This is a powerful alternative to a conventional time-consuming model development process. The integrated solution enables rapid and consistent computation of stresses and deformations within and around hydrocarbon-bearing reservoirs.

Operators are able to use JewelSuite with Abaqus to predict and explain current and future full-field geomechanics behavior under drilling, stimulation, and production scenarios. This helps guide field development strategies, reduce costs, and improve oil and gas recovery.

For More Information

www.bakerhughes.com/JS3D

Embedded Signal Processing Analysis Solution for Abaqus

The current analysis trend in advanced mechanical systems is the inclusion of better dynamic insight into real-world problems. This industrial trend requires intensive signal processing operations, and intensive correlation between simulated and experimental results.

FE analysts use third-party (external) software to execute dynamic signal processing operations compromising the interactivity with the FE model, increasing time for post-processing operations, and creating a higher probability of errors due to the amount of uploaded data and lack of interactivity between the analyst and the FE model.

Critical Materials' solution, SPA - Signal Processing for Abaqus®, is a fully embedded module in Abaqus capable of executing signal processing and mathematical and statistical operations for a dynamic analysis. A fully interactive experience is guaranteed—making use of the conventional functionalities made available by Abaqus and also through

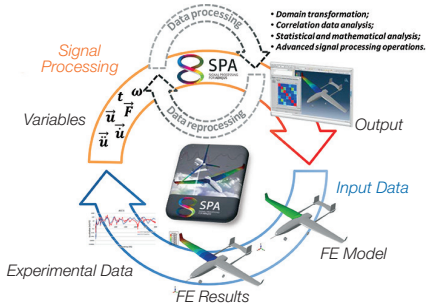


Figure 1. Workflow of SPA highlighting its interaction with both experimental and FE data.

new user-driven functionalities specially developed for SPA. Additionally, SPA allows the importation of external data that can be further used to perform correlation studies between experimental and simulation results.

To highlight some of these features, SPA was used to process dynamic data retrieved from an UAV (Unmanned Aerial Vehicle) FE model. In this context, statistical (correlation model/FE model) and domain transformation capacities are highlighted (see Figure 1). The composite wing of the UAV was analyzed through a steady-state modal dynamic procedure in order to understand its response when excited near the wing root. Initially, different mesh sizes were tested and MAC (Modal Assurance Criteria) was considered as criteria for mesh convergence studies, selecting 10 nodes to act as displacement control points. SPA outputs the MAC matrix by comparing Eigenmodes similarity between two different mesh sizes. Based on MAC output it is possible to choose the mesh size that is more convenient for a specific analysis frequency range.

Also, temporal behavior was retrieved, directly from the modal analysis procedure, by choosing a node at the wing tip and applying an IFFT (Inverse Fast Fourier Transform). This procedure enabled the visualization of the displacement profile for a specific time period without the need of conducting a simulation in the time-domain.

For More Information

www.critical-materials.com/products/spa

Optimization of Surgical Positioning in Total Hip Replacement

When joint pain and loss of mobility occur as a result of end-stage osteoarthritis or other severe hip pathologies, over 250,000 people choose to have total hip replacement (THR) surgery. Even though THRs are one of the most successful surgical inventions in medical history, they do fail. THR failures are often grouped as "early" or "late," with early failure usually due to dislocation of the head from the cup, and late failure frequently due to adverse biologic reaction to wear debris generated at the bearing surface. Despite nearly six decades of investigation, the ideal surgical orientation of THR components remains unclear. Positioning of total hip bearings involves significant tradeoffs, as cup orientations most favorable in terms of stability are not necessarily ideal in terms of reduction of contact stress and wear potential. Previous studies and models have not addressed these potentially competing considerations for optimal THA function. Additionally, it is currently unknown whether the ideal orientation varies on implant parameters, such as variations in femoral head size. We, therefore, investigated optimal surgical cup orientation with a previously generated and physically validated finite element (FE) model of metal-on-metal THR.

Method

The FE model consisted of bony anatomy and the hip soft tissues (see Figure 1). Five dislocation-prone motions as well as gait were considered, as were permutations of femoral anteversion (0° to 30°), femoral head diameter (32 mm to 48 mm), cup inclination (25° to 75°), and cup anteversion (0° to 50°), resulting in 4,320 distinct FE simulations. A novel metric ("Performance Score") was developed to delineate optimized cup orientation by considering both surface wear and component stability (see Figure 2 A-D).

All FE simulations were performed using Abaqus/Explicit.

Results

Ideal cup position was substantially more sensitive to cup anteversion than to inclination. Regressions demonstrated strong correlations between optimal cup inclination vs. head diameter (Pearson's r = -0.88), between optimal cup inclination vs. femoral anteversion (r = 0.96), between optimal cup anteversion vs. head diameter (r = 0.99) and between cup anteversion and femoral anteversion (r = -0.98) (see Figure 2 E-H).

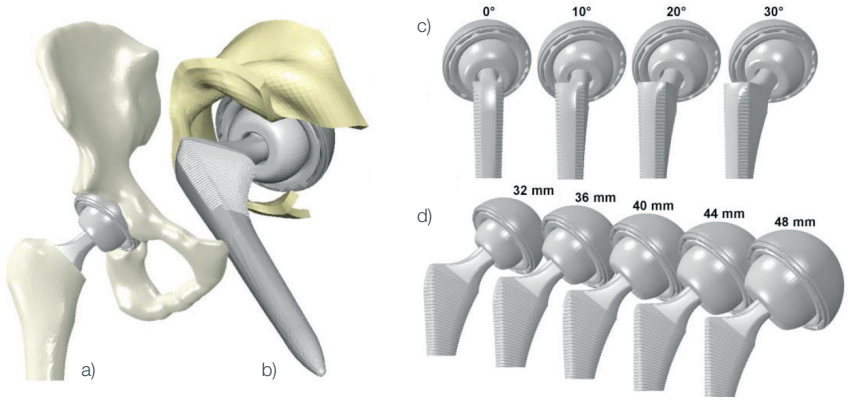


Figure 1. The FE model consisted of bony anatomy (a) and the hip soft tissues (b, anterior region of capsule rendered transparent for clarity). Four values of femoral anteversion were considered (c) as were five distinct femoral head sizes (d).

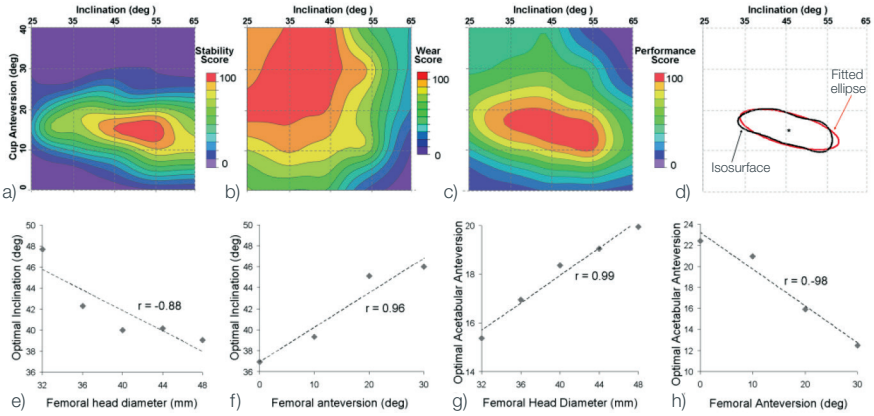


Figure 2. For every combination of femoral head size and femoral anteversion (20 such combinations total), the Stability Score (a) and Wear Score (b) are combined to determine the Performance Score (c). The optimal orientation is determined as the center of an ellipse fitted to an isosurface of scores > 90 (d). When considering all 20 combinations, regressions could be performed demonstrating optimal surgical orientation (e-h).

Discussion

The "landing zone" of ideal cup orientation did not increase with increased head size, challenging the presumption that larger heads are more forgiving in terms of stability and durability. Additionally, ideal cup positioning was considerably more sensitive to cup anteversion than to inclination. Finally, the current investigation is the first to quantitatively suggest that ideal cup positioning varies with both femoral anteversion and femoral head size.

Positioning THR bearings involves significant tradeoffs with regard to stability and long-term bearing wear. The computational analysis identified optimal orientations to balance these considerations. These tradeoffs help explain the alarming rates of adverse local tissue response reported for large head metal-on-metal THR devices that have demonstrated an improvement in joint stability. The conclusions from this study can readily be translated to other hard bearing surfaces—including ceramics and highly cross-linked polyethylene—suggesting careful consideration of the choices and compromises in THA design are required for all bearing couples.

Jacob Elkins, John Callaghan, Douglas Pedersen, and Thomas Brown, Department of Orthopaedics and Rehabilitation, University of Iowa, Iowa City, Iowa

For More Information

www.uiortho.com

Simulating Moisture Distribution and its Impact on Delamination in a Microelectronics Package

A semiconductor package is a metal, plastic, glass, or ceramic casing containing one or more semiconductor electronic components. This casing provides protection for the semiconductor device and mechanical strength to support the leads and handling of the package. For economic reasons, many devices are encapsulated in an epoxy plastic, even though this material is hydrophilic and so absorbs moisture in a humid environment. The impact of moisture on interfacial delamination is two-fold. On one hand, it produces hygrostress, which increases the crack driving force, and on the other hand, it decreases the interface resistance to delamination propagation.

Obtaining accurate values of the distribution of moisture concentration in an Integrated Circuit (IC) package can be important. Hygrostresses are induced since polymers will expand with the absorption of moisture while other materials, such as metallic alloys, will neither absorb moisture nor expand. These stresses are in addition to the thermal stresses caused by the mis-match in thermal expansion between the materials. Moisture at a metal-polymer interface reduces its interfacial adhesion, and if a defect, a delamination, or a void is present, water vapor pressure can also add to the interfacial stress. With the knowledge of the moisture distribution within the package, the design of packages can be improved and the possibility of delamination can be reduced.

To achieve this objective, finite element simulation is very useful since analytical solutions are often limited to simple geometries and it is difficult to measure moisture concentration during physical tests. Many numerical schemes, such as the temperature-moisture analogy, normalization approach, and direct concentration approach, have been proposed to model moisture absorption and desorption. Some of the numerical schemes proposed might be capable of modeling physical observations more accurately but can be more involved than others.

In this numerical study, a plastic package (Plastic Quad Flat Pack, PQFP) with a small crack at the pad-encapsulant interface, is first subjected to moisture preconditioning

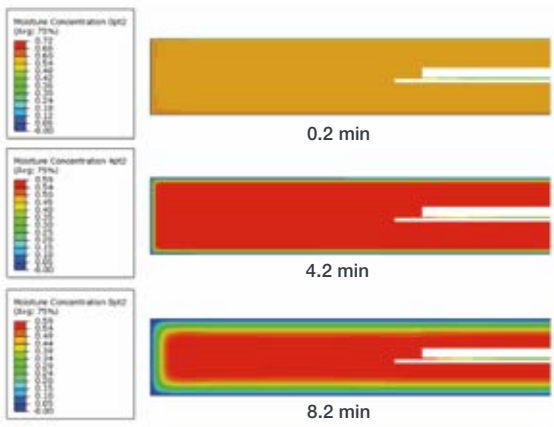


Figure 1. Moisture distribution during solder reflow.

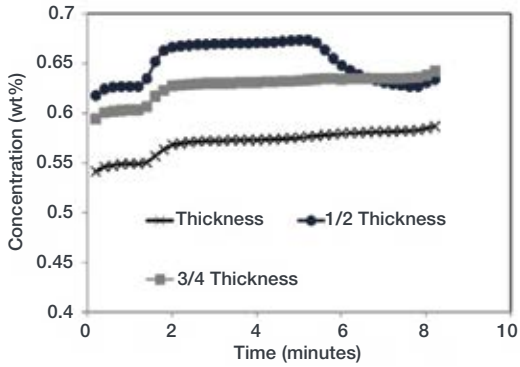


Figure 2. Effect of package thickness on moisture concentration on the crack tip.

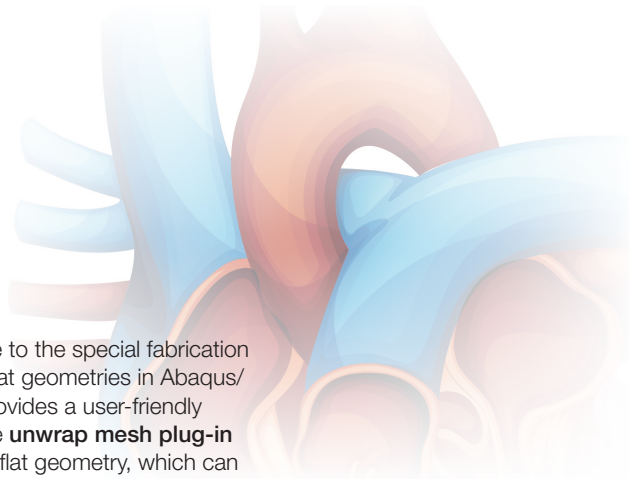
(85°C/85%RH) and subsequently exposed to the solder reflow process, following the industry standard JEDEC J-STD-020C. Local moisture concentration and fracture mechanics parameters at the interface crack—energy release rate (ERR) and mode mixity—are determined using Abaqus. Figure 1 shows the moisture concentration in the package obtained from the simulations. The thickness of the package and the assumptions made in the numerical model greatly influenced the local moisture concentration during solder reflow. Dependence of C_{sat} on temperature is commonly observed in materials at elevated temperatures. In such cases, although the moisture content in the atmosphere is low, the local moisture concentration can increase during solder reflow. Figure 2 illustrates examples of the described phenomenon. When a package is thick, for instance in the case of "Thickness" and "3/4 Thickness," moisture concentration at the crack tip continues to increase during the entire solder reflow process, whereas for a thinner package, moisture concentration reaches a peak at around five minutes and decreases thereafter. ERR and phase angle arising from a combination of moisture, vapor pressure, and temperature factors are then calculated.

It is observed that a thicker package leads to higher ERR and for all the cases, ERR reaches a maximum in the vicinity of peak solder reflow (around 5-6 minutes) primarily attributed to the increase in mismatch of thermal expansion with elevated temperature. Higher moisture concentration at the interface will lead to increased degradation of the interface. With the combined effects of reduction in interfacial toughness and increase of thermal ERR due to elevated temperature, the risk of delamination during solder reflow increases significantly especially at peak solder reflow temperature.

Ho Siow Ling and Andrew A. O. Tay, Department of Mechanical Engineering, National University of Singapore

For More Information

mpetayao@nus.edu.sg



Abaqus/CAE Plug-in Utility

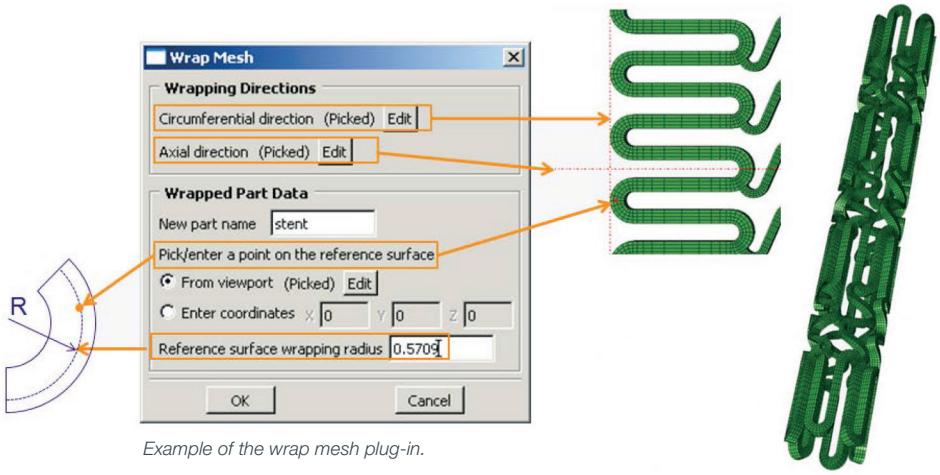
How to perform wrapping and unwrapping of stents in Abaqus/CAE

Stent models are usually created from a flat (unrolled) geometry due to the special fabrication techniques used. It is very convenient to create and mesh these flat geometries in Abaqus/CAE. Once the flat part has been meshed, the **wrap mesh plug-in** provides a user-friendly interface to wrap the mesh into the final tubular shape. Conversely, the **unwrap mesh plug-in** can unwrap a deformed mesh of a stent from tubular shape into a 2D flat geometry, which can be used to validate the FE solution by comparing it to a 2D image taken for the actual stent; or it can be used by a laser cutting tool to create a stent based on the deformed shape.

Here's What You Do:

Steps to create a **wrapped** part using the plug-in:

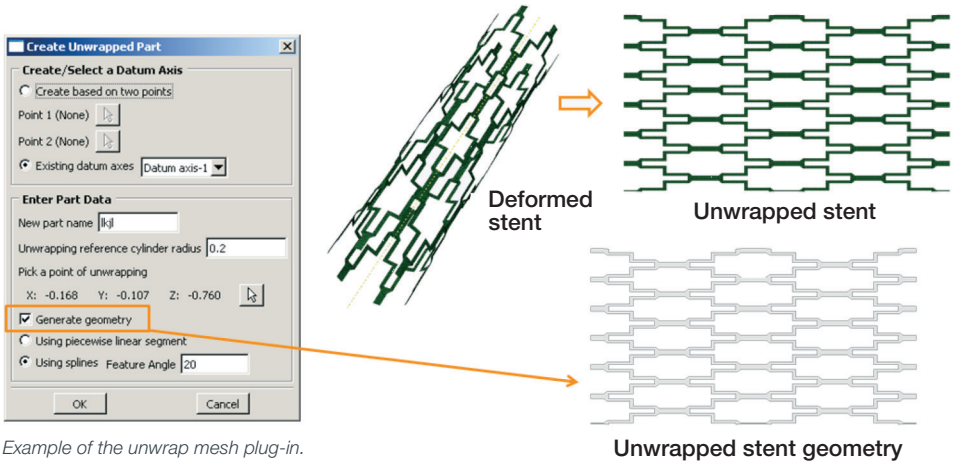
1. Select circumferential direction from the current viewport
2. Select axial direction from the current viewport
3. Enter the name of the wrapped part
4. Either pick point on the reference surface or enter coordinates of the point
5. Enter wrapping radius of the reference surface
6. Click OK



Example of the wrap mesh plug-in.

Steps to create an **unwrapped** part using the plug-in:

1. Create a datum axis by picking two points or use an existing datum axis
2. Enter the name of the unwrapped part
3. Enter the unwrapping reference radius
4. Pick a point from where the unwrapping has to occur
5. Select a check box if a 2D geometry part is to be created
6. If the 2D geometry part is to be created, select either piecewise linear segment or the splines option with a feature angle
7. Click OK.



Example of the unwrap mesh plug-in.

The plug-in applications along with the installation and the usage guidelines can be accessed using the following Knowledge Base articles: QA00000008170 (wrap mesh plug-in) and QA00000021479 (unwrap mesh plug-in).

For More Information

https://www.3ds.com/support/knowledge-base/

Inside the SIMULIA Learning Community

Download the Abaqus Student Edition—for Free!



The 6.12 version of the Abaqus Student Edition is now available for download for all SIMULIA Learning Community members. It is available for both 32- and 64-bit versions of Windows Vista and Windows 7 Operating Systems and contains the following functionality:

- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit model sizes up to 1000 nodes
- Abaqus/CFD model sizes up to 10,000 nodes
- Full Abaqus Documentation Collection in HTML and PDF format including the Getting Started and Example Problems Manuals.

The Abaqus Student Edition may be installed on personal laptops and is governed by a personal use license that does not expire. This Edition is an excellent tool for evaluating Abaqus or sharpening your skills while away from your office or campus license.

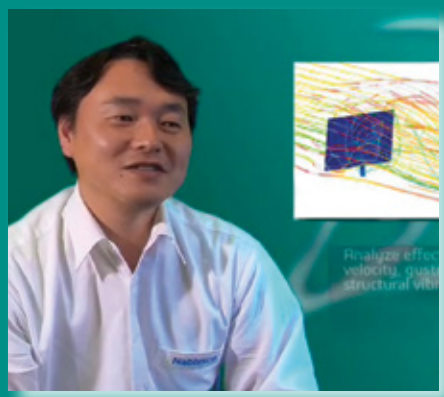
Learn More:
swym.3ds.com/#community:73

New User Testimonial Video

Nabtesco Ensures Gear Strength and Endurance in Gusty Wind Conditions

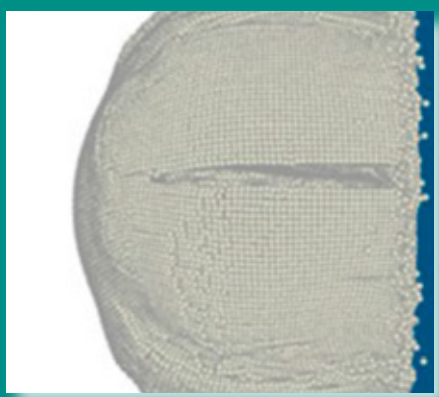
When Nabtesco needed to optimize their pinion gears, the part that transmits power from the drive to the nacelle, they turned to SIMULIA's solutions. Their engineers utilized Abaqus to analyze the stress and contact area on the gear teeth, and developed their own subroutine to model the history of both the stress and contact area on the model. With the help of Isight, Nabtesco was also able to significantly reduce design time, perform design optimization, and process automation. The result was an optimized gear design that is significantly more robust than the previous one.

Watch this video:
www.3ds.com/SCN-February2013



New Technology Brief

Simulation of Hail Impact Using the Smoothed Particle Hydrodynamic Method in Abaqus/Explicit



Ice or hail impact is an important design consideration in the aviation, energy, and astronautics industries. Physical testing of impact scenarios can be expensive and impractical. Numerical simulations provide cost effective and convenient tools for including hail impact scenarios in the design process.

Numerically simulating high-speed ice or hail impact is challenging because of the extreme deformation, fragmentation, contact, and material model complexity. Learn how the Smoothed Particle Hydrodynamics functionality in Abaqus/Explicit can be used to analyze ice impact.

Download the Tech Brief:
www.3ds.com/SCN-February2013


IF WE
want to avoid surgery, can we swallow a robot?
Nanobot medicine
– a dream our software could bring to life.
3D
i
V+R
It takes a special kind of compass to explore the world's future possibilities. Innovative companies use our 3DEXPERIENCE software platform to understand the present and navigate the future.
Find out more: 3DS.COM/LIFE-SCIENCES
DASSAULT SYSTEMES
IF WE ask the right questions we can change the world.