SIMULIA Community News

May/June 2012
www.3ds.com/simulia

Eni Drills Deeper into Oil and Gas Reservoir Sustainability

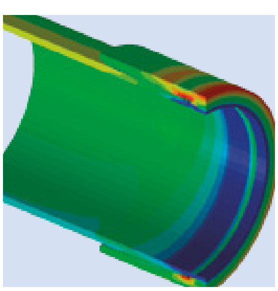
DASSAULT SYSTEMES

SIMULIA
Community News

May/June 2012
www.3ds.com/simulia

Energy Spotlights

On the cover (left to right): Silvia Monaco, Roberto Vitali (SIMULIA), Stefano Mantica, Gaia Capasso



Also In This Issue

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

Contributors
Silvia Monaco and Stefano Mantica (Eni), Kazuhiko Yokoji (Nabtesco), Jueren Xie (C-FER Technologies), Soo Hyun Park (Samsung), Cosimo Chiarelli (Thales Alenia Space Italia), Leonardo Chica (University of Houston), Mehrdad Zangeneh (University College London), Angie Schrader (Firehole Composites), Srinivas Kodiyalam (NVIDIA), Parker Group, Ken Short, Dale Berry, Matt Ladzinski, Jill DaPonte, Bhushan Thakar (SIMULIA)

Graphic Designer
Todd Sabelli

The 3DS logo, CATIA, SOLIDWORKS, SIMULIA, DELMIA, ENOVIA, 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, 2012.

JUNE_SCN_412_VOL 2

SIMULIA

Below my letter, you'll see some of the results from our SIMULIA User Community survey, conducted in April of this year. We highly value your input and appreciate the time you took to complete the survey. I can assure you that we do our very best to take steps to make improvements based on your feedback.

A Better Approach—Thanks to Our Users

Like most of us, I'm often asked about what I do for a living. In the past, I would explain about the finite element method, parallel computing, and partial differential equations. The person I was talking to would respond with a bewildered expression and the conversation would end rather abruptly.

Today, I have a better approach. I tell them that I'm in the coolest software business imaginable—that I am privileged to work with companies from all industries who use our technology to help improve the lives of hundreds of millions of people. I explain that we help the users of our cool simulation software to improve the performance and safety of the aircraft we fly in and the cars that we drive, engineer running shoes to be more biomechanically efficient, and develop more reliable medical devices and implants. We even play a part in bringing engineering innovation to everyday consumer products like razors, tablet computers, phones, cameras, and cleaning products—including the packaging!

This edition of *SIMULIA Community News* (SCN) highlights the energy industry, where the expanded use of our tools is having a positive impact. Simulation is enabling increased production of the energy resources that power our economy and personal lives—while at the same time helping safeguard our environment. Our cover story by Eni illustrates the use of Abaqus in running extremely large and detailed geomechanical models. This is a great example of how multiphysics is improving efficiency and safety, leading to lower costs for oil extraction.

Another user story describes how C-FER is using Abaqus to access oil reserves that were previously not economically viable. The Nabtesco case study focuses on the role Abaqus and Isight play in improving the reliability and endurance of wind turbine components.

Speaking of reliability and robustness, note the way Isight is being used by Thales Alenia Space Italia to optimize hypersonic vehicle design! This case study and many others illustrate impressive time savings and significant decision guidance that can be achieved using Isight with Abaqus and other simulation tools. If you have not yet tried Isight, I strongly urge you to read some of our recent user papers on our website at www.3ds.com/scc-papers and arrange for a trial license at your company.

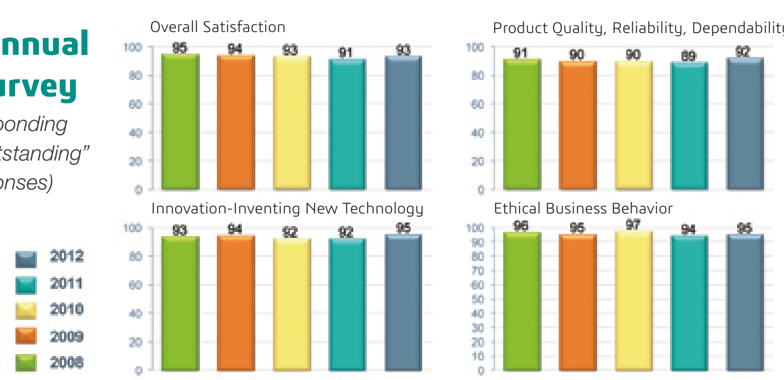
As you know, we highly value your input on ways we can improve our software and services. We have heard from our users that a SIMULIA-focused online user community would be a great way to allow users to interact and learn from each other. So we recently launched the SIMULIA Learning Community to help bring all our users together—whether they are involved in product development, research, or teaching—in a way that allows them to learn, ask questions, and connect to other users in the community. I invite you to get actively involved in the SIMULIA Learning Community to help make it a vibrant and valued user resource.

I hope that I'll be able to meet many of you at one of our Regional User Meetings that will be held at more than 20 locations worldwide. I plan to attend as many as possible so I can renew my vision of what we do—through learning about what you do with our software. I'm constantly impressed by the applications and results of what our users can achieve with our software. I look forward to seeing more examples of this and receiving your feedback in person.

Ken Short
Vice President, Strategy & Marketing

2012 Annual User Survey

Percent responding "Good to Outstanding" (2072 Responses)



Nabtesco Powers Up Wind Turbine Gear Performance with Realistic Simulation

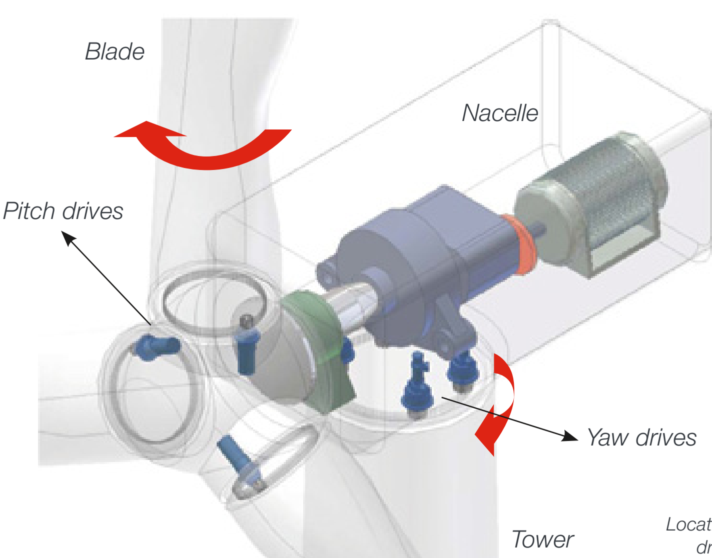
Abaqus FEA helps improve strength and endurance of nacelle yaw and blade pitch drives

Wind turbines have become an iconic symbol of alternative energy with their tall, upright towers and graceful, spinning blades. Choosing a proper site for the towers is certainly the starting point for maximizing the output of a large wind farm. But for the most efficient conversion of the kinetic energy of wind into mechanical energy, or 'wind power,' additional control over the position of each turbine's nacelle and blades is essential. This is the task of the yaw and pitch drives, which adjust the physical orientation of those components in response to fluctuations in the velocity and direction of prevailing breezes.

Located at the base of the nacelle, a yaw drive changes the direction which the nacelle faces. Where each blade meets the nacelle, a pitch drive changes the angle of the blades. Working in tandem, these computer-driven gear mechanisms automatically optimize the orientation of the turbine relative to the wind so power can be generated in the most efficient manner possible.

Gearing up for large wind turbines

Precision yaw and pitch drives for robotic reduction gears are a specialty of Nabtesco Corporation of Japan, known worldwide for unique technology that allows those components to be smaller and lighter than many conventional robotic drives. But when the company expanded into energy harvesting equipment, such as wind turbines and solar energy collector trackers, they realized that 'smaller, and lighter' have their limits in large industrial structures.



Location of yaw and pitch drives on wind turbine.

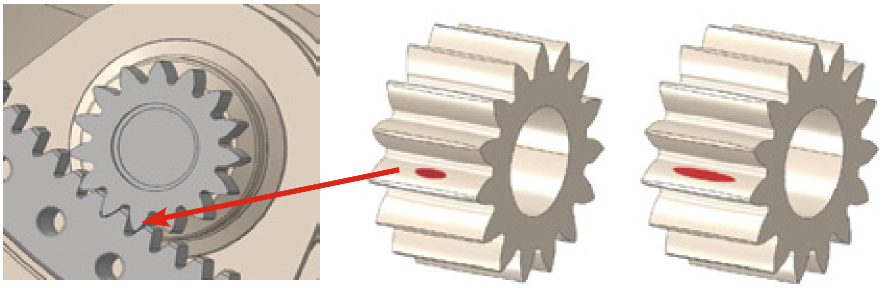
"Particularly in the wind industry, our biggest challenges are ensuring sufficient gear strength and long-term endurance in gusty conditions," says Kazuhiko Yokoji, CAE manager, CAE & material department, whose team provides computer-aided engineering services for the entire Nabtesco group. "Our reduction gears are made up of a lot of very complicated assemblies, with many parts that come in contact with one another. For each wind turbine configuration, we have to provide our customers with the best possible design that minimizes overall stress while maintaining durability."

Every yaw and pitch drive in a wind turbine engages with a pinion gear, made from specialized treated steel, that transmits power from the drive to the nacelle or blade. In a huge wind turbine, the rotation angle between drive and pinion gear teeth is particularly small, so repeated contact over time—particularly under the 'routine' stresses of high winds and tower vibration—has the potential to damage tooth surfaces and cause assembly breakdowns.

Balancing contact for best performance

To minimize such hazards, the engineers strive to balance the contact between ring and pinion teeth so that the 'normal' stress of rotation and the sheer stress of the teeth against each other are reduced. Decreasing the stress on teeth improves their durability—and that of the entire assembly as well.

Since the thinner, outer edges of a tooth are most susceptible to damage, teeth are manufactured with a curved surface ("crowning") so that the edge dips away to either side of the center of the tooth and the contact between teeth occurs somewhere in the middle. Too steep crowning decreases the contact area too much, intensifying tooth stress. Too shallow crowning allows stress to extend too close to tooth edges. Damaging edge contact can also increase if the pitch drive shaft bends. "Since crowning has such a significant effect on contact area and



(Left) Smaller pinion gear engages with outer ring gear. Note that rotation angle where teeth intersect is very small; this can contribute to gear tooth damage over time. A small contact area (red, center) between teeth can be optimized with CAE analysis (red, right) to decrease average stress and prolong gear life.

maximum stress, pinion tooth shape is a major focus in our design process," says Yokoji.

Finding that optimum tooth shape is a time-consuming challenge when done manually. The contact area between drive and pinion teeth can be visually identified in a test rig by coating the teeth with a special paint that rubs off where they engage. "But using this method means a lot of pinions have to be manufactured and tested in order to identify which shape is the most desirable," says Yokoji. "And this method doesn't give us the overall stress data that helps us evaluate durability. We felt we could get a more complete picture using an analytical model that could simultaneously analyze tooth contact area and maximum stress."

SIMULIA solutions speed up analysis

Yokoji's team uses Abaqus unified finite element analysis (FEA) from SIMULIA to solve a number of wind turbine-related design challenges. "Abaqus' capabilities for reproducing motion, and also investigating fluid-structure interaction (wind is modeled as a fluid) are particularly important to Nabtesco," says Yokoji.

To create an analytical simulation of the pinion test rig, the engineers started from a global Abaqus CAE model based on imported CAD geometries of rig, drive, and pinion. The bottom surface of the test apparatus was modeled as fixed. Resistance (this data, including the effect of wind velocity, was provided by a Nabtesco customer) was applied to the reduction gear shaft, and the pinion was rotated at a prescribed angle. This allowed the team to see when and where the opposing teeth engaged as the gears rotated, and what the resulting stresses were.

For a deeper understanding of what happens when the teeth interact, the engineers needed to mesh the relatively small area of tooth-face contact with particularly fine C3D8R elements. To reduce computation time for this portion of the model, they created a submodel that contained only those areas of acute interest. When compared against the painted gear rig tests, the FEA results showed good agreement.

"We still wanted to get a complete picture of how the total contact area and stresses fluctuated over the course of the entire engagement processes," says Yokoji. "This would provide us with the 'big picture' of contact history that we needed for evaluating tooth designs for durability." So his team developed a proprietary post-processing technique that employed an Abaqus user subroutine to show the history of how stress developed from the start of engagement, through changes in the rotation angle, to the end of engagement. Now they had a complete toolset to start fine-tuning individual pinion tooth shape.

Optimizing the analysis with Isight

At this point the group turned to Isight process automation and optimization software from SIMULIA, coupling their own in-house program for creating contact stress distribution history into the workflow. This allowed them to quickly generate clear results from huge result files and evaluate both the immediate (stress) and the long-term (strength/durability) consequences of every design change. They then plotted the results into three-dimensional stress distribution graphs that tracked all phases of gear engagement and let them see how modifying gear crowning contours affected overall performance.

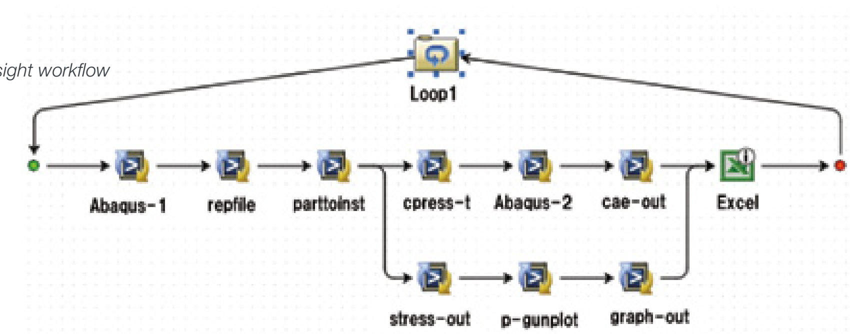
"Our CAE workflow now enables us to optimize crowning of pinion gear teeth designs accurately and with far less manpower than before," says Yokoji. "This method reduces design times dramatically."

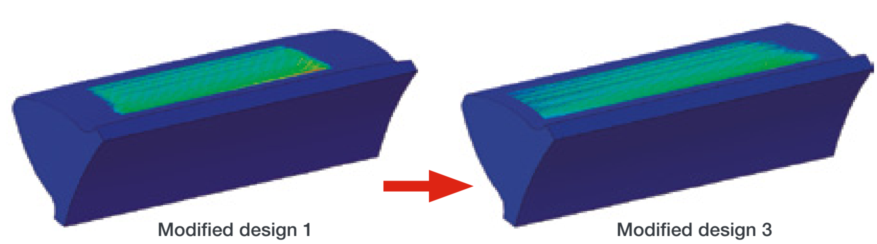
Nabtesco's newly designed gears are just being introduced into the field and the company is gathering data to generate accurate comparisons with previous models. "We believe that there will be a significant improvement in durability," says Yokoji. "Abaqus and Isight give us confidence that we can design durable gears that can stand up to the rigors of wind power generation."

For More Information

www.nabtesco.com/en
www.3ds.com/SCN-June2012

Isight workflow





Optimizing gears with Isight resulted in a design that lowered contact stress across the entire surface.

Fat Finger Syndrome Solution Found with Finite Element Analysis

Samsung employs realistic simulation to design mobile device keypads for fewer typos

Anyone who's ever typed on a computer keyboard or mobile device keypad has experienced this: While aiming for one letter, you hit a different one on a nearby key. The result? Poor spelling, mangled messaging, an email you never should have sent. As electronic devices and instrumentation become increasingly compact, the search for a cure for "fat finger syndrome," as it is known in the industry, is becoming ever more urgent.

Flat touch screens operated by pressure sensors may be taking over tablet computers and smart phones, but keypads and keyboards are still widely used in many electronic devices. Desktop computers, laptops, some cell phones, remote controls, and appliances, such as washing machines and dryers, all still rely on the touch of a finger on a spring-loaded key.

At the Global Production Technology Center of Samsung Co., Ltd. in Suwon, Korea, engineers strive to stay ahead of the trends toward tinier keys and denser key layouts with each new model. "We are working to make products both smaller and easier to use," says Soo Hyun Park, Manufacturing Core Technology Team, Global Production Technology Center at Samsung, "so we want to reduce the amount of mistyping that can occur on the more compact keypads."

Samsung engineers decided to delve deeper into the fat finger phenomenon by examining the physics behind keystrokes, finger pressure, and strike angle to determine what can go wrong and how to make it happen less often. "Since keyboards will remain widely in use for the foreseeable future, we will continue to study the physical user interface to better understand the ergonomics of human-device interaction," says Park. Using Abaqus finite element analysis (FEA), they were able to cut mistyping errors from 35% to 7% with an intermediate prototype model of a QWERTY keypad, so named for the sequence of letters that run left to right on a standard type-key layout (see Figure 1).

Realistic simulation of the interaction between human fingertips and device keys enabled Park's team to identify the variables that lead to mistyping. "By systematically modifying the relevant design parameters, we could see which keypad configurations led to the least number of typing errors," he says.

Where the finger meets the keypad

When they first decided to tackle fat-finger physics, the engineers realized that they needed two different finite element models to realistically simulate the problem: one of a human fingertip and the other of a device key.

Nature has, of course, already designed the 'perfect' human finger configuration; Samsung needed to come up with an FEA model that could mirror it. "It was important to define the separate material properties of skin, subcutaneous tissue, bone, and nail in order to model the overall biodynamic response of the finger," says Park. Since most small-device QWERTY keypad users type with both thumbs, the engineers started from the thumb bone structure of a 178 cm (about 5 feet 9 ½ inches) tall male combined with exterior skin surface data from a 3D laser scan.

Basing their finger-parts' definitions on previous studies of human tissues, they queried the available material models in Abaqus for the properties and element types they needed to build their FEA model (see Figure 2, left). The nail and bone were modeled as linearly elastic, while the skin (epidermis and dermis) was assumed to be hyperelastic and linearly viscoelastic. The deeper subcutaneous tissue of the finger was represented by a biphasic

Figure 1. Prototype of an intermediate mobile personal computer (now discontinued) that was used in the Samsung keypad optimization study.

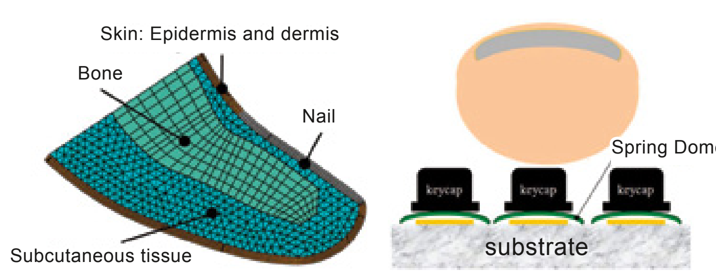


Figure 2. Abaqus FEA model of side view of a human thumb tip (left) and CAD model of finger contacting keys showing spring dome (right).

material composed of a fluid phase and a hyperelastic solid phase (essentially a sponge-like porous material representing muscle, fully saturated with fluid to represent plasma).

Next the team needed to create a virtual keypad where the interaction of fingertip and key could be simulated. When a key is pushed, it compresses a spring dome (see Figure 2, right) that completes an electronic circuit to register whatever symbol (letter, number, or punctuation mark) is on that key. As they meshed their FEA models, the engineers built in a nonlinear spring element beneath the key to simulate this action of the spring dome.

To capture the 'snap ratio', which is the tactile feeling that the user experiences when pushing down on a key, they lab-tested an actual spring dome with a load cell underneath it to record the pressure as the key was pushed with different degrees of force. They could then use this real-world spring stiffness data to characterize the response of the nonlinear spring element in their model.

Striking a balance between a host of variables

Now it was time to put thumb and key models together. By surveying users' real-world gripping and pushing behavior as they typed (they photographed 16 men and five women as they struck the 'K' key with their thumb), the engineers had pinpointed the average angle they wanted to use where the finger model hit the key model. This turned out to be 16.6 degrees from the front of the keypad and 16.4 degrees from the side. When the 'K' key was pressed, the force on the two neighboring keys on either side was calculated; the 'mistyping ratio' could then be defined as the force on the neighboring key buttons divided by the force on the target key.

The contact pressures predicted by the thumb/keyboard FEA model (see Figure 3) were compared with test measurements obtained through the use of an I-SCAN instrument (from Tekscan), which contained an extremely thin (0.1 mm) flexible tactile load cell sensor. There was good agreement between models and tests.

With a working FEA model of fingers striking keys now in hand, the engineers next turned to design optimization of the key layout. What side-by-side configuration of keys would produce the least mistyping? The team identified seven design variables:

- Width of key
- Height of key
- Vertical angle of key
- Pitch of key
- Inclined angle between center points of adjacent keys
- Reference face level of key
- Slope of key.

Each of these seven variables needed to be applied to the three keys being studied. "The Python script in Abaqus was very useful here because it enabled us to automatically carry out multiple, repetitive FEA-model tasks," says Park. The mistyping ratio of each analysis case was determined and then a response surface method was used to identify the optimum key position. The analysis revealed that the first five design variables all had pretty similar effects on mistyping, while the last two (reference face level and slope) showed stronger, yet contradictory, tendencies. Running through 27 sets of analyses, the engineers determined that the pitch of the keys on either side of the center key had to

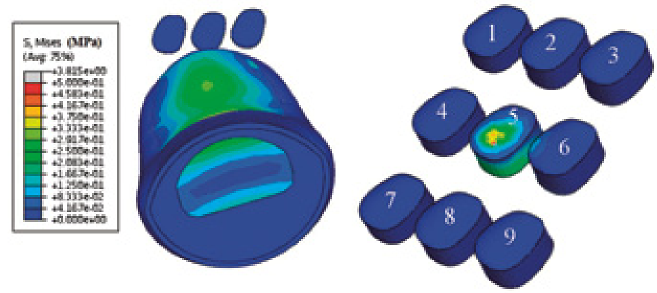


Figure 3. Analysis of mistyping ratio.

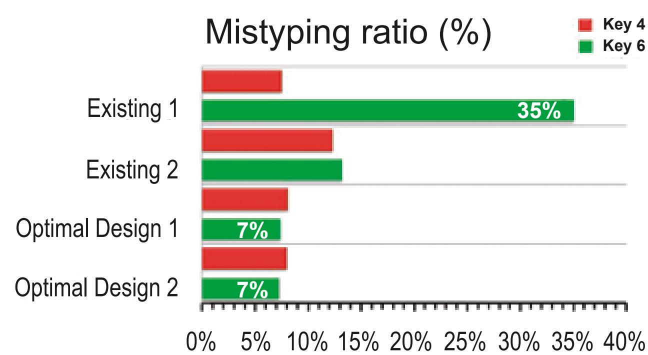


Figure 4. Previous prototype device keypad designs (top two sets of bar graphs) produced too much mistyping (an error rate as high as 35%). Optimal design 2 (bottom bar graph) showed the lowest overall mistyping ratio (7%).

be made greater in order to decrease the mistyping ratio. (Isight could now be used to automate the running and post-processing of these jobs.)

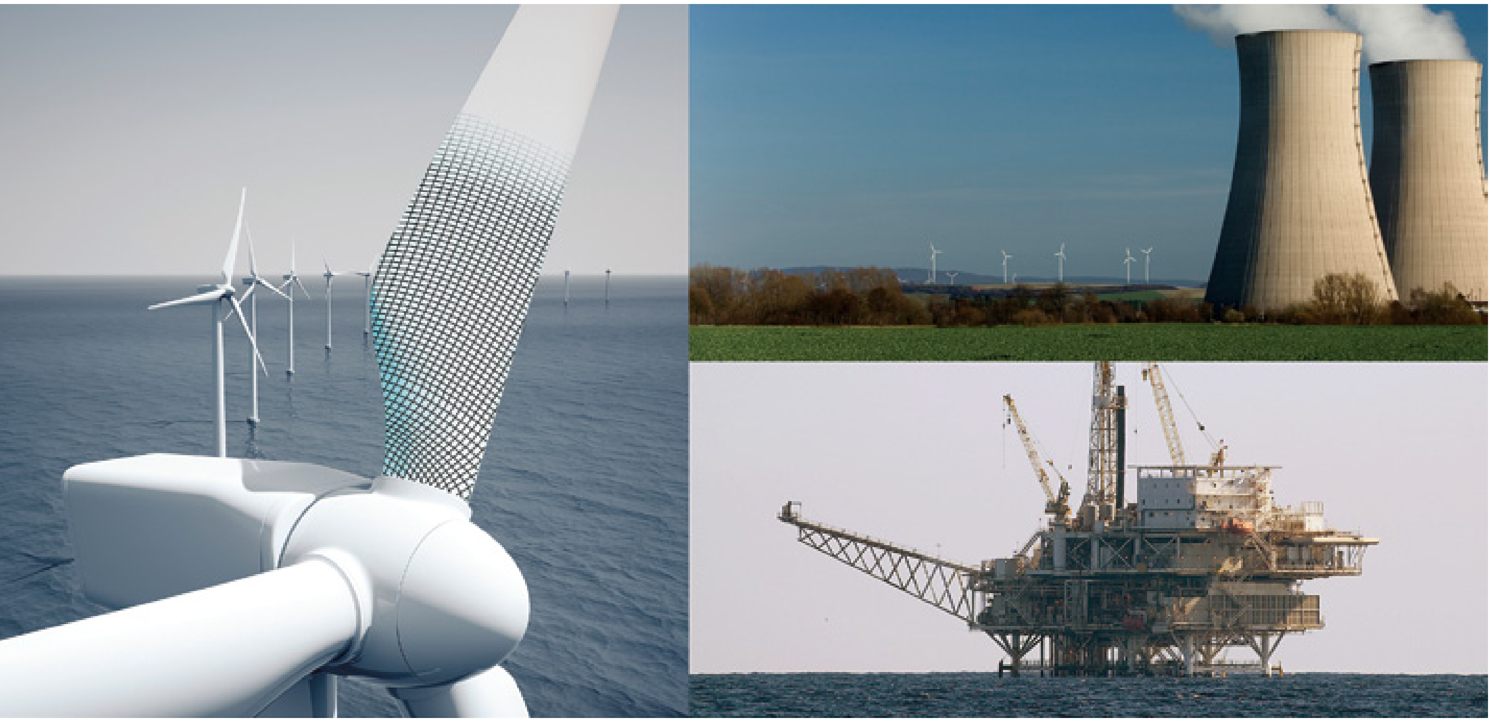
Using the results of their virtual finger/keypad optimization exercise, Park's team was able to systematically alter their key design variables and identify a design for which the mistyping ratio improved from a 35% error rate to 7% (see Figure 4). Again, the FEA results showed good agreement with subsequent experimental tests.

Going forward, Samsung engineers see value in developing hand, arm, and whole body models so that all aspects of the user's motion can be incorporated into device design. Says Park, "As human/device interfaces continue to advance, the use of FEA to model these interactions will contribute to reduced trial and error with the design process and improve the 'emotional' quality of our products."

About Soo Hyun Park

Soo Hyun Park is an engineer of Mechatronics & Manufacturing Technology Center for Samsung Electronics Co., Ltd. Since joining Samsung in 2006, he has developed new structural analysis techniques for the company using Abaqus, and he has managed many projects for electronic products. Park graduated with a M.S. in Mechanical Engineering from Seoul National University in 2006.

For More Information
www.samsung.com



Supplying Realistic Simulation to Meet Global Demands

Dale Berry, Senior Director of Technical Marketing, SIMULIA

Energy demand throughout the world is increasing. Yet the economic, financial, and environmental challenges facing society and the energy industry are more demanding than ever. Looking back to our prior energy review in 2009 shows how far we've come, but how far we still have to go to achieve a safe, clean, abundant supply of energy that meets worldwide demand. Realistic simulation is an increasingly important tool along the path to a sustainable future.

The January 2009 issue of SCN (formerly called *INSIGHTS*) contained a cover story on evaluating nuclear plant safety by TÜV, a case study on realistic simulation in mining by Beck Arndt Engineering, and other updates including a review of new features in Abaqus version 6.8. Mahesh Kailasam, the energy industry lead for SIMULIA at that time, wrote an overview of our strategy focused on providing key functionality to enable usage of advanced simulation and optimization to aid extraction and usage, and reduce environmental impact in an energy hungry society. (For reference, the January 2009 issue of *INSIGHTS* and a recent interview with Stephan Arndt on the latest in mining simulation can be found at the url at the end of this article.)

In just a few years, much has changed.

Since 2008, the energy industry has been through events both predictable and unimaginable. The Deepwater Horizon accident in the Gulf of Mexico in 2010 illustrated the equipment, environmental, and operational challenges facing offshore oil production. In 2011, the earthquake and tsunami that struck Japan caused a terrible nuclear crisis at Fukushima. And no matter your opinion on global warming, the increase in global CO_2 levels appears to be accelerating.

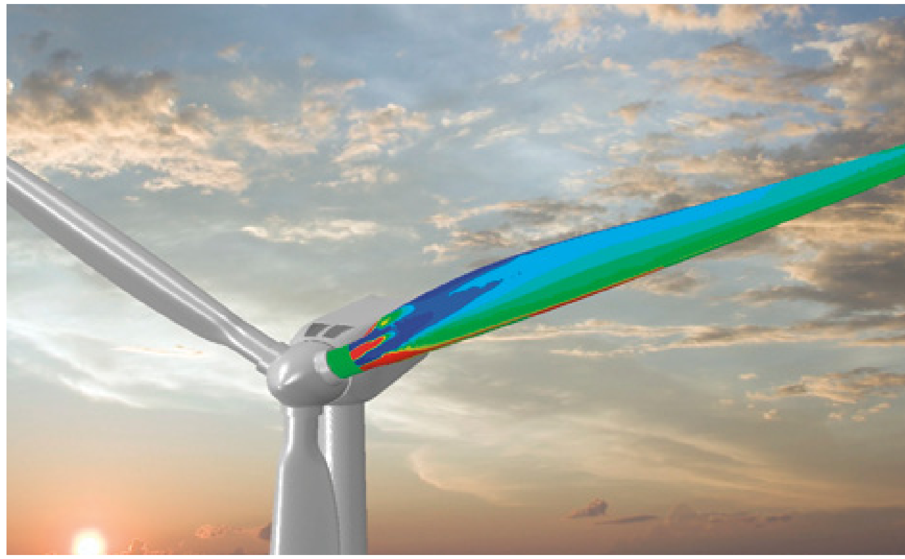
On the positive side, global automobile fuel economy is increasing, and the introduction of hybrid and all-electric cars is accelerating. Here in the United States, huge natural gas deposits are now beginning to be exploited driving the cost of natural gas down, and contributing to a switch to gas for electricity production. Consequently, CO_2 emissions and imported energy in the U.S.A. are both decreasing, while concerns about the environmental impact of “fracking” (the production process to extract the gas) is on the rise. Similar challenges and opportunities are being confronted by other world regions as well.

All of these global and industry changes highlight the fact that no matter what your personal or national politics are, the world has a need to more fully exploit our limited energy resources of all kinds (whether petroleum, natural gas, wind, solar, geothermal, or other) and to make sure those energy resources are extracted carefully, with environmental preservation in mind, and subsequently used to their fullest.

Dassault Systèmes (3DS) and the SIMULIA brand are in full support of these trends. First, the energy industry is at the heart of the Dassault Systèmes **3D**EXPERIENCE strategy. The **3D**EXPERIENCE Platform transforms the way "innovators will innovate with consumers" by connecting designers, engineers, marketing managers, and even consumers, in a new 'social enterprise'. To support this strategy within 3DS, a multidisciplinary team focusing on the energy industry has been formed. I'm proud to say that after 13 years in SIMULIA, Mahesh Kailasam has joined this team as an Experience Director. His new role as a thought-leader behind the value of the entire 3DS portfolio in energy will allow SIMULIA to continue to broaden the value of realistic simulation for the energy industry in alignment with the other 3DS brand applications. The first focus of this team is the wind energy sector where simulation plays a large role in both blades and wind turbines and gearing systems as illustrated by the Nabtesco story (p. 4). I'm sure you will see and hear more from Mahesh on the 3DS energy strategy in the coming years.

Since 2008, usage of our products to help achieve corporate and societal goals is increasing. In the energy sector in particular, usage of realistic simulation to help ensure an adequate, yet safe, energy supply is growing even faster. We have seen a marked increase in the number of papers presented from the energy industry at our SIMULIA Community Conference as well as in the number of energy industry attendees, which doubled between the 2008 and 2012 events.

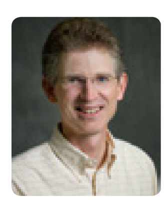
This trend is illustrated both by the Eni (p. 12) and C-FER (p. 16) articles. These are not just cool one-of-a-kind applications of simulation of arcane subjects on the fringes of analysis. Rather, they are stories that illustrate what I think will come to be the required use of simulation to reduce the economic and environmental risk of deep-earth and deep-water extraction. As "easy oil" comes to an end, we must use all of our intellectual capabilities to meet our needs without spoiling our planet. The recent 3DS acquisition of Gemcom illustrates just how important modeling (and hence simulation) of the earth and its resources are to achieving this vision.



Of course, what users want to know is: What's new in our software? Since the release of Abaqus 6.8, our technical progress and delivery of functionality of value to the energy industry has also accelerated. We have added eXtended Finite Element Method (XFEM), coupled pore pressure-temperature-displacement elements, Aqua wave modeling in Abaqus/Explicit for offshore use, as well as expanded solver technology for large geotechnical problems. In the Abaqus 6.12 release, there is new functionality in smooth particle hydrodynamics (SPH) and continued improvements in co-simulation and Abaqus/CFD, just to name a few. As Abaqus technology is the foundation for FEA and multiphysics in SIMULIA V6, this robust simulation technology will—in future releases—be added to the new SIMULIA ExSight product suite as well.

In addition to Abaqus, there has been substantial uptake in the energy industry of the Isight product line used for process capture and design space exploration. Last year in our January 2011 issue, for example, we highlighted a story from Baker Hughes, who used Isight to discover an entirely new and superior geometry for an oil-field tool called a "downhole tubular expander." At this year's SCC, there were 14 total papers on Isight usage to manage, control, and derive insight into a product's design space. You, our users, are beginning to "get it" on how Isight provides an entirely new level of understanding of design beyond what you normally get from individual analysis jobs. Such understanding is essential given the challenges facing the energy industry today.

All-in-all, due to uncertainty worldwide in relation to the economy, energy sustainability, and environmental safety, the growth in the use of realistic simulation within the energy industry is certain. At SIMULIA, we plan to continue to respond aggressively to the technology requirements and the growing expectations of the energy community. I encourage you to join and participate in the online SIMULIA Learning Community to learn more and share your industry requirements. Plus, to stay informed be sure to attend your Regional User Meetings later this year and the SIMULIA Community Conference in 2014. Stay connected!



About Dale Berry

As the Senior Director of SIMULIA Technical Marketing, Dale has 24 years experience with SIMULIA products including 13 years with AC Engineering as an Abaqus representative and consultant. In 2001, Dale became responsible for technical marketing and product planning to broaden the applicability of SIMULIA tools for workflows in all major industries.

For More Information
www.3ds.com/simulia
www.3ds.com/SCN-June2012

Designing-in Certainty in an Uncertain World

Thales Alenia Space Italia uses Isight to optimize and virtually test hypersonic reentry vehicle designs

As computer-aided engineering (CAE) tools become increasingly sophisticated, engineers can now refine their designs to nearly final form virtually. They can also push physical prototype testing later and later in the product development cycle. While prototyping remains essential for most types of products, in some industries it is impossible to create a test environment that produces real-world conditions. Vacuum chambers and wind tunnels can assess certain aspects of space travel, but it is impossible to test for all conditions at the same time.

Such is the case for Thales Alenia Space Italia (TAS-I), an European space system solution provider, in its ongoing design of hypersonic reentry vehicles for the European Space Agency (ESA) and other commercial and government entities. As a result, TAS-I engineers involved in these efforts rely mainly on CAE from start to finish of the design/virtual test/build process.

Design exploration and process automation help optimize complex designs

Cosimo Chiarelli, head of the aeromechanics and propulsion unit at TAS-I, is part of a team charged with designing and testing every aspect of the company's space reentry vehicles on the digital drawing board. Given the complex physics of atmospheric reentry, this is a daunting analysis challenge requiring a multidisciplinary optimization (MDO) approach. The project's feasibility was demonstrated in the AeroThermodynamics Configuration for Space Transportation (CAST) program financed by the Italian Space Agency (ASI) and conducted by the Italian Aerospace Research Center (CIRA) in early 2007.

First there is the spacecraft's structure to consider: this includes its geometry (length and shape) and the dimensions and material characteristics of the shell and thermal protection system (TPS). Then there is the trajectory, comprised of the vehicle's speed, altitude, and angle of attack (a steep angle increases friction and heat, while a shallow angle does the opposite). Thermal conditions for the vehicle's windward, leeward, and nose zones also have to be taken into account, along with the aerothermodynamic loads—such as pressure distributions and aerothermal fluxes—that it encounters. The final design makes allowance for all of these variables, with a focus on a time window of approximately 150 seconds: the most critical portion of reentry.

ESA's hypersonic reentry vehicle demonstrator under development at Thales Alenia Space Italia Courtesy of ESA - J. Huart

To optimize their designs, the team conducts a separate simulation for each of the physics disciplines. They use a collection of software packages—including six commercial, five proprietary, and two aerodynamic codes. And they divide the analysis into seven major computational "tasks" and 40 "sub-tasks," many with their own input and output file types (see Figure 1).

"We did a trade-off study of several commercial software tools and selected Isight to run our optimization," says Chiarelli. "It has features and add-ons that allow us to integrate all of the different codes and interfaces used for our physics sub-routines." Isight facilitates the creation of flexible simulation workflows and automates design exploration. With such a vast design matrix, TAS-I engineers have used the software as a way to systematically evaluate the countless possible solutions.

To conduct a feasibility study of their new optimization methodology, the engineering group—working in close association with Exemplar, a value-added reseller for SIMULIA in Italy—chose a theoretical hypersonic reentry vehicle and applied simplified assumptions. Further streamlining the process, they decided to optimize globally for all variables combined, rather than locally for each individual variable. Identifying minimized cost as the primary design target, they applied Isight's adaptive simulated annealing algorithm, an exploratory technique that helped them search the envelope of design solutions; this approach is one of many statistical methods available in Isight, including Design of Experiments and Design for Six Sigma. The SIMULIA Execution Engine was used to implement Isight process flows across company compute resources and enabled the sharing of results throughout the enterprise.

Performing two-hundred iteration cycles in only a day, the analysis identified strong correlations between variables and arrived at several designs that satisfied the requirements for the spacecraft. More importantly, it demonstrated that the various simulations could be integrated and that optimization could be successfully applied to the resulting workflow.

"Isight played a key role. It helped us unify our process and saved a considerable amount of time," says Chiarelli.

Design verification also benefits from automated workflows

Having established a methodology to optimize their designs, the engineering team now turned its attention to verifying that the vehicle would actually function as designed during the harsh conditions of reentry. As with their first analysis, the virtual testing process involved consideration of a large number of related variables—more than 25 parameters associated with flight conditions needed to be examined while computing aerothermal loads during reentry—to evaluate the design and performance of the vehicle's all-important guidance, navigation, and control (GNC) and thermal protection systems

(TPS). Again, TAS-I engineers worked in collaboration with Exemplar and used Isight to manage the workflow.

In past design projects, once a reference trajectory had been defined, the identification of extreme aerothermodynamic (ATD) loads on the proposed design was divided into a series of separate steps. But this approach only allowed engineers to look at specific points on the vehicle (not at all points, or even at all zones) and only at the worst-case scenarios (worst attitude, angle of attack, flap deflections, and trajectory, for example). For these reasons, design assumptions were typically conservative and vehicles might suffer performance-degrading effects from overdesign.

To apply detailed real-world conditions, the team based the new Isight-driven methodology on a strong association between GNC and ATD simulations and plugged in an extensive proprietary ATD database for all 25 flight condition measures. The goal of the analysis was to assess each time step….for every possible trajectory….for any zone on the vehicle.

TAS-I engineers handled this huge computational challenge by dividing the analysis into three discreet tasks and relying on Isight's capability to combine the separate steps and run the entire process flow automatically (see Figure 2).

- The first task was a Monte Carlo simulation experiment (a powerful random sampling technique available in Isight) in which the GNC model's performance was evaluated for the 25 flight parameters during 1,000 sample trajectories.
- In the second part of the analysis, the Monte Carlo results were post-processed for three key variable time histories, at 100 different locations on the vehicle, for all 1,000 trajectories (a total of 300,000 time histories).
- In the third step, all of the time histories were analyzed to identify maximum heat loads and fluxes for every point on the vehicle during each trajectory.

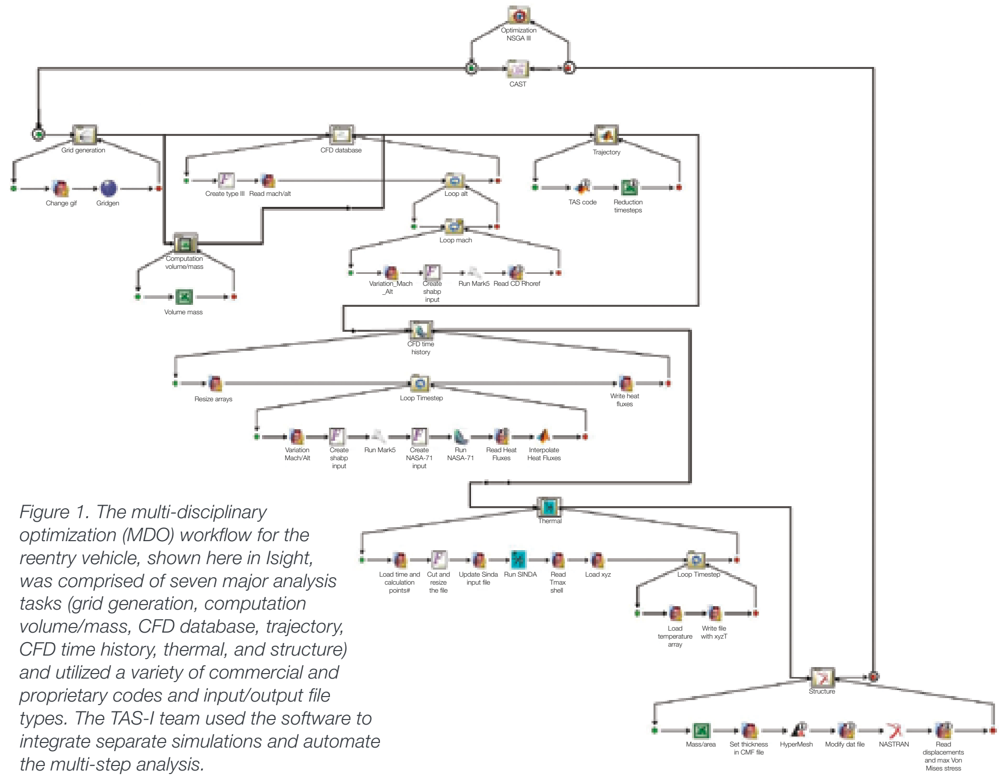


Figure 1. The multi-disciplinary optimization (MDO) workflow for the reentry vehicle, shown here in Isight, was comprised of seven major analysis tasks (grid generation, computation volume/mass, CFD database, trajectory, CFD time history, thermal, and structure) and utilized a variety of commercial and proprietary codes and input/output file types. The TAS-I team used the software to integrate separate simulations and automate the multi-step analysis.

As hoped, the new method enabled the TAS-I team to handle the data-rich aerothermal characterization of the vehicle, calculating the loads on the entire vehicle surface while taking into account all the inaccuracies affecting the trajectory itself. The traditional sequential process of design, analysis, and optimization, which used to take two weeks, now takes only about 48 hours in the automated Isight environment.

"Our analysis is more robust now because we can process huge amounts of data," says Chiarelli. "Using the Isight environment, we have been able to reduce the use of conservative assumptions for our designs."

Probe payloads are the payoff

Because of the success of their optimization and verification studies, TAS-I engineers are now incorporating higher fidelity codes and applying the new methodologies to more complex reentry vehicle models. They are able to process larger data sets in shorter times: TAS-I estimates that man-hours for new simulation iteration could be reduced by about 80 percent once the automated process has been established and proved.

Design flexibility has been enhanced, as well, because a change in any parameter can be recalculated simply by running a new analysis loop within the established process flow. Moreover, the new workflow has decreased manual errors from data transcription, increased efficiency, and stimulated deeper collaboration between engineering disciplines and departments.

According to Chiarelli, integrating and automating analysis workflows are helping TAS-I engineers address the many design uncertainties of hypersonic reentry. The ultimate benefit to the scientific community will be the safe return of probes with valuable payloads from deep space.

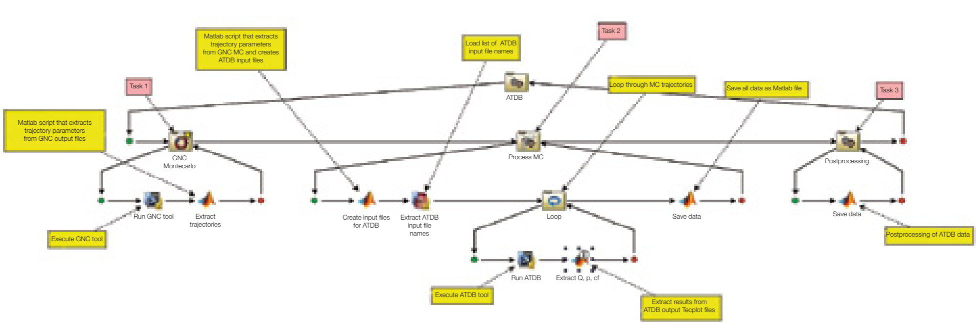
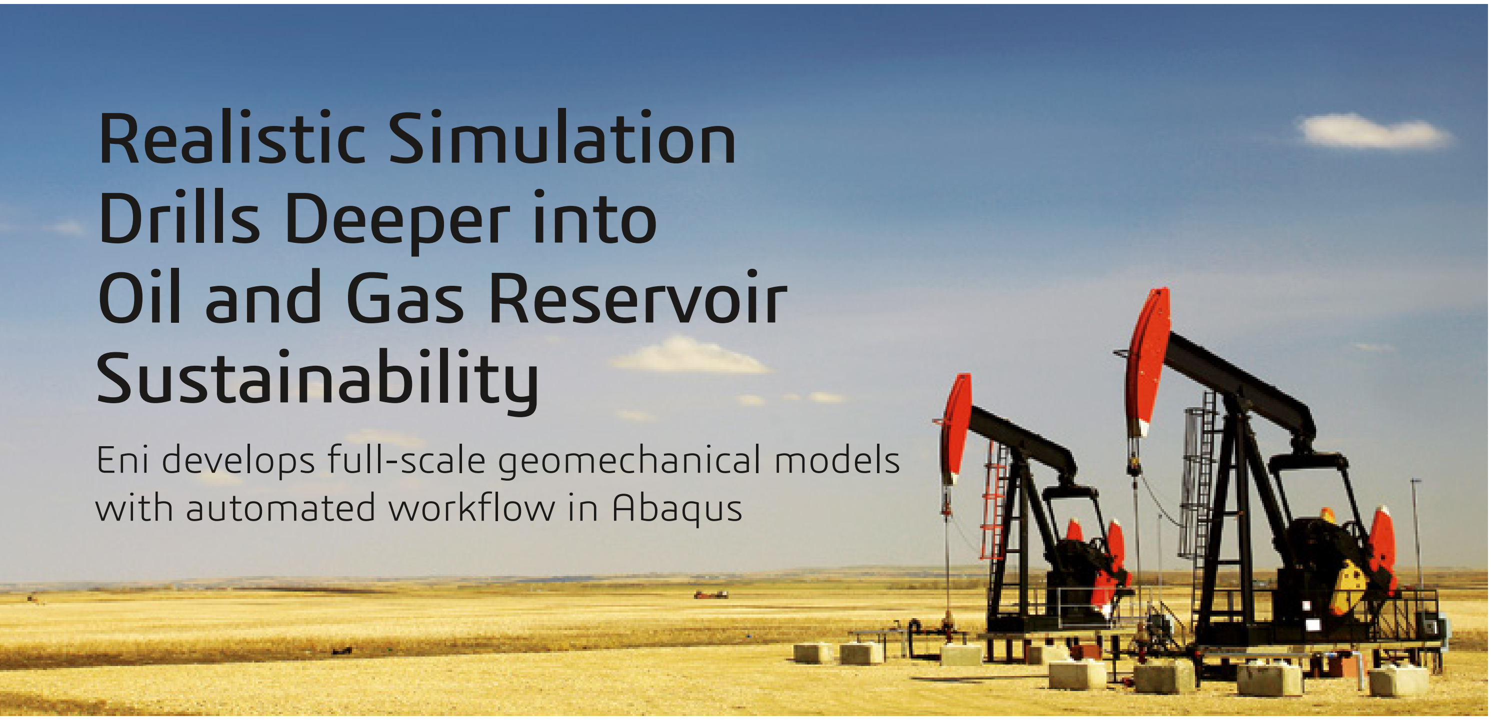


Figure 2. The workflow for verification of a theoretical reentry vehicle includes three major sub-routines. Isight's open integration technology allowed TAS-I engineers to include proprietary scripts, applications, and databases. The unified process flow has decreased time investment by an estimated 80 percent and eliminated data transcription errors.

For More Information
www.thalesgroup.com
www.3ds.com/SCN-June2012



Realistic Simulation Drills Deeper into Oil and Gas Reservoir Sustainability

Eni develops full-scale geomechanical models with automated workflow in Abaqus

Managing the lifespan of an oil or gas field is an ongoing, big-picture concern for energy companies. With huge investments needed just to start the flow of hydrocarbons from a well, keeping production levels at optimum rates for as long as possible is a necessity: the world still relies heavily on petroleum.

The challenge of such reservoir "sustainability" has been partially met with flow-predicting software and on-site monitoring tools. When flow rates drop, the injection of fluids can boost production higher again. But there is more to the puzzle than how fast the oil or gas will come out, and for how long. As petroleum is pumped from its original bed, subsidence and compaction of the soils surrounding the reservoir can affect rock permeability, the integrity of boreholes, equipment function, and even the geology of the land around the production sites.

This happens because the extraction of petroleum from underground reservoirs leads to a reduction in pore fluid pressure within the reservoir, which results in a redistribution of stress in the rock formation. Since rock deformations are often plastic, this produces subsidence of the ground around the reservoir that expands over time as extraction continues. As the rock deforms, the permeability of the rock itself changes, which then affects the flow of fluid within the reservoir. The phenomena of fluid flow and mechanical deformations are thus inexorably coupled to each other (see Figure 1).

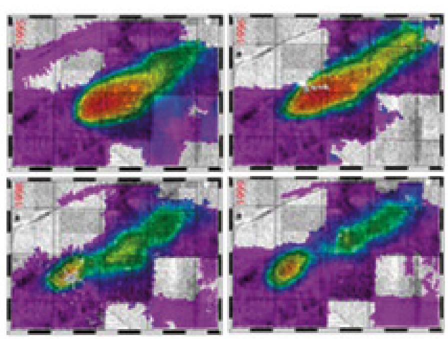


Figure 1. The NASA images above show the rapid rate of subsidence (in red) of over 3 cm/month during active production in the Lost Hills area of California. Note that production occurs over several years and so easily results in several feet of subsidence.

Subsidence challenges petroleum industry both on and offshore

Reservoir compaction has been extensively investigated to determine its impact on both hydrocarbon field production and environmental stability, onshore or offshore. The effects can be cumulative. For example, in the Netherlands, subsidence at the large Groningen gas field, though only on the order of tens of centimeters to date, poses significant long-term challenges since large portions of the Netherlands are below sea level and protected by dikes. Some important, much-documented lessons from the past clearly demonstrate the negative impact of the phenomenon over time. The city of Long Beach, California, experienced subsidence of some 20 square miles of land, with a surface dip of 29 feet near the center, due to extraction from the huge Wilmington oilfield. Subsidence from the Goose Creek oilfield in Texas affected over four square miles, with up to five feet of surface drop. Remediation in both cases cost millions of dollars. Offshore, the Ekofisk field in the North Sea suffered seafloor subsidence that required highly expensive interventions to re-establish the safety of the producing platforms.

While the majority of oil and gas projects don't encounter challenges at such a large scale, petroleum engineers now clearly understand the value of starting with deeper knowledge of the terrain at the earliest stages of reservoir development.

A more realistic view of what lies beneath

As an integrated energy company operating in engineering, construction, and drilling both off- and onshore for customers around the world, Italy's Eni S.p.A. devotes considerable manpower and resources to research into reservoir management. Their work helps clients close to home as well: Gas fields in the Adriatic Sea have become a major source of energy for the country. Due to the particular morphology of the shoreline in that area, it is of paramount importance for Eni to be able to correctly predict the land subsidence that may be induced by hydrocarbon production in order to guarantee the sustainable development of the offshore fields.

Eni has for some time been developing advanced methodologies for studying the problem of reservoir subsidence and compaction with the help of Abaqus finite element analysis (FEA). "Abaqus is our main stress/strain simulator for studying the geomechanical behavior of reservoirs at both field and well scale," says Silvia Monaco, geomechanical engineer in the petroleum engineering department of Eni E&P headquarters in San Donato, Milan, Italy.

The ability of Abaqus Unified FEA to realistically simulate complex structural and material behavior makes it well suited to the task. Although the study of subsidence in petroleum fields has been slowly advancing since the 1950s, earlier approaches were based on an assumption of homogeneity of the whole system, i.e. they described the side-, over-, and under-burdens of rock and soil with mechanical properties identical to those of the reservoir. But soil and rock are in fact very non-homogeneous and show highly nonlinear behavior that is strongly influenced by previous stress paths. Incorporating FEA into a computer model of a reservoir provides a much more realistic simulation of this truth. Different types of finite elements, a large variety of material properties, coarser or finer element meshes, and data-based boundary conditions can all be woven into a prediction that much more accurately reflects the full effects of the geomechanical complexities unfolding beneath the surface.

Coupling Abaqus with the leading flow simulator

Of course it's the start of oil or gas flow out of the reservoir that gives rise to the effects that FEA models anticipate. So the Eni group links their Abaqus FEA models to the leading flow simulator ECLIPSE (from Schlumberger). "Fluid-flow analysis is essential in order to forecast production and manage field development," says Monaco. "But the geomechanical processes at work in the rock and the fluid contained in its pore space are also of primary interest since they can affect the behavior of the reservoir itself. By transferring pore pressure depletion data from ECLIPSE into Abaqus, we can more fully understand the mechanisms involved in surface subsidence in order to forecast and prevent well failures and adverse environmental impact." (see Figure 2)

Running a computer model of the large-scale dynamics of an entire oilfield is becoming much more efficient these days, thanks to huge leaps in parallel processing and high-performance computing that can handle FEA models with millions of degrees of freedom (DOF). And for Eni, creating those kind of models in the first place has recently become much easier.

When the Eni team first began coupling Abaqus with their ECLIPSE models several years ago, there was still considerable effort involved in creating the complex workflow needed to produce simulations that behaved realistically and correlated well with real-world measurements. "Previously, we had a number of non-automated procedures as well as simplifications related to the geometry description, such as the smearing of faults and simplified treatment of collapsing layers," says Monaco. "It used to take almost two months to complete a single model suitable for running."

Continued

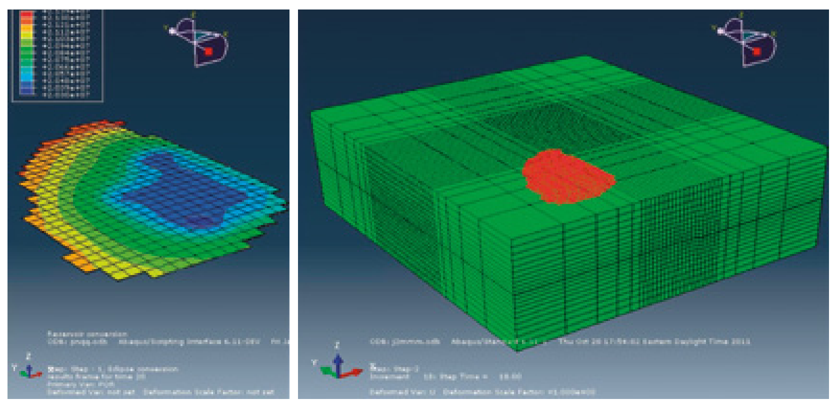


Figure 2. (Left) Active region generated from the flow simulation solution. (Right) Abaqus mesh showing the active region within a reservoir. Linking ECLIPSE with Abaqus incorporates the geomechanical effects of extraction for a more realistic simulation of full-site development over time.

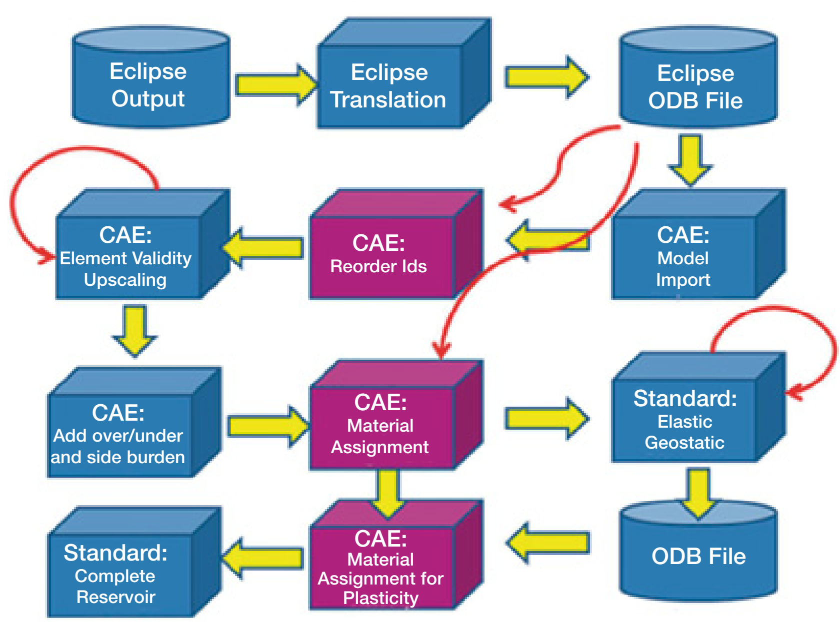


Figure 3. Reservoir geomechanics workflow. An output database file (ODB) is created from ECLIPSE and imported into Abaqus/CAE for creation of an FEA geomechanical model from which the stress distribution over a reservoir can be derived. A plastic analysis then predicts the geomechanical deformations (subsidence) in the surrounding terrain that will result from this stress.

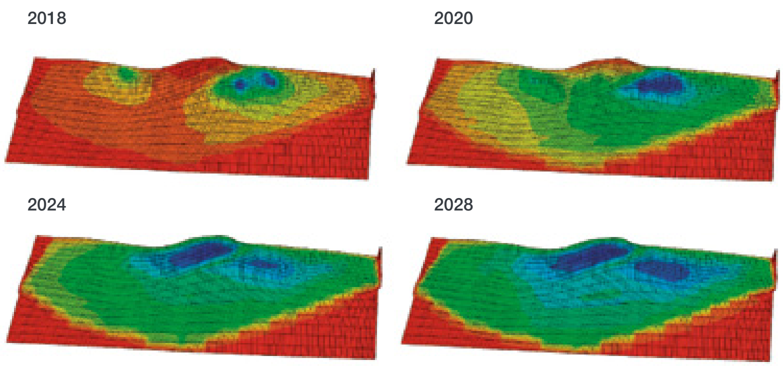


Figure 4. Four increments in an Abaqus FEA simulation of subsidence in a hypothetical oilfield, displayed over ten years. Blue areas denote greatest downward displacement of the surface. This particular example from Eni contains just 300,000 degrees of freedom; enhancements in model setup and automation now allow the running of huge full-scale models with millions of DOF in just a few hours. Rock faults (not pictured here) can be included in simulations.

Geomechanics group at Eni: (left to right) Caterina Topini, Gaia Capasso, Anna Corradi, Stefano Mantica, Marco Brignoli, Giorgio Volontè, Silvia Monaco, and Francesca Bottazzi.

With the goal of streamlining this process, Eni teamed with SIMULIA in a two-year R&D collaboration, the results of which were presented at the 2011 SIMULIA Customer Conference in Barcelona, Spain. "SIMULIA worked closely with us to develop new features in Abaqus that definitely change the approach to geomechanical reservoir simulation by allowing a completely automated workflow," says Monaco. "Now we can build a geomechanical model in only four weeks: We obtained an improved efficiency compared to the previous process in terms of elapsed time needed to set up an analysis. Moreover, the new iterative solver implementation provides a strong reduction in computational times and memory usage that further speeds up the execution of the study."

The new workflow (see Figure 3) automates the transfer of data from ECLIPSE into Abaqus and speeds the subsequent FEA model set-up, expanding the flow-centric view of a field-scale reservoir into a much richer 3D profile of flow-plus-subsidence over time. This involves the following steps:

- A translator establishes a link between ECLIPSE and Abaqus. All the information from the reservoir model (grid, properties, and pressure) is automatically populated into the FEA model in the form of data that can be used for the geomechanical analysis. For example, ECLIPSE cells are designated either as gas, oil, or water according to the percentage of fluid saturation they hold; in the Abaqus model the elements that are automatically derived from these cells can be assigned as many as 300 different material property definitions. ECLIPSE pressure history descriptions are also translated into Abaqus pore pressure values. These values are essential for calculating the change in the effective stress in the reservoir. Abaqus meshing tools automatically adjust the elements and nodes as needed and perform upscaling, a process that condenses the size of the FEA model by merging horizontal rows of elements while maintaining the vertical zones (where drill data has already been collected), which are more relevant to subsidence prediction.
- Burden regions over, under, and to the sides of the oil reservoir are created in Abaqus to extend the analysis to the terrain beyond the reservoir as the petroleum is pumped out.
- Once the model is set up, results from an initial elastic run are used to update the plasticity values (since rock behavior is elasto-plastic) to make the models more realistic. The simulations are then run over time increments so predictions can extend over many years (from the year 2018 to 2020 to 2024 to 2028, as seen in Figure 4).

New geomechanical models provide greater predictability

"We now have a logical scheme for easily and automatically executing all the steps required for creating and running our geomechanical models," says Monaco. "This significantly improves our efficiency in terms of user time in the preprocessing stage. Our analyses are now measurably more precise."

Such precision is helping Eni better serve their energy customers in developing strategies for ensuring sustainable oil and gas production for the long term.

"The increased quality of the results we've obtained with the new Abaqus implementations allows for a highly accurate and predictive environmental analysis," says Monaco. "This is a key point for a sustainable development of Italy's hydrocarbon reservoirs. Moreover, as a result of the cutback in computational times, a larger number of studies can also be performed internally, thus strengthening the link between geomechanical engineers and the team in charge of the geological and reservoir model construction."

In the near future, the Eni team plans to turn its attention to a comprehensive integration of the huge quantities of deformation measurements they've acquired at different scales and through different methodologies. "The automatic calibration of the rock properties of a geomechanical model will allow for this," says Monaco. "Isight process automation and optimization software from SIMULIA could be a proper tool for obtaining results."

For More Information

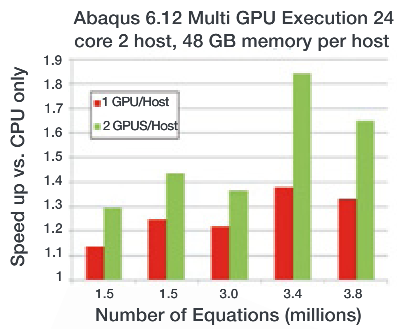
www.eni.com
www.3ds.com/SCN-June2012

Multi-GPU Support in Abaqus 6.12

With the ever-increasing demand for faster computing performance, the high-performance computing (HPC) industry is moving towards a hybrid model, where GPUs and CPUs work together to perform general-purpose computing tasks. In this hybrid computing model, the GPU serves as a co-processor to the CPU. Co-processing refers to the use of an accelerator, a GPU, to offload the CPU of the compute-intensive tasks and to increase computational efficiency. Today's GPUs can provide memory bandwidth and floating-point performance that are several factors faster than the latest CPUs.

In order to exploit this hybrid model and the massively parallel GPU architecture, application software needs to be redesigned. Engineers at SIMULIA and NVIDIA, Dassault Systèmes Simulation partner, have been working together on the use of GPUs and CUDA parallel programming architecture to accelerate the sparse direct solver in Abaqus. SIMULIA officially released support of single GPU computing with Abaqus 6.11, and the most recent release, Abaqus 6.12, works with DMP (distributed memory parallel) and multiple GPUs.

In the energy industry, well completions are going deeper and deeper into the earth to recover hydrocarbons. The market demands that oilfield tools and equipment be operated under much heavier loads and more severe environmental conditions. Abaqus has proven to be an excellent tool for evaluating equipment conceptual designs and structural capabilities prior to prototyping and testing. Parallel efficiency and turnaround times continue to be important factors behind engineering decisions to develop higher-fidelity finite element analysis models. Engineers at Baker Hughes, one of the world's largest oilfield services company, use NVIDIA Tesla GPUs for accelerating their production of Abaqus/CAE simulations, enabling them to nearly double the number of simulations in a given period of time. More details can be obtained from their 2012 SIMULIA Community Conference technical paper found at the link below.



Increased levels of parallel processing in Abaqus with GPU computing enable larger and more complex simulations to be addressed in product development workflows. The performance speed-ups allow Abaqus users to add more realism to their models, thus improving the quality of the simulations. A rapid CAE simulation capability from GPUs has the potential to transform current practices in engineering analysis and design optimization procedures.

For More Information
www.nvidia.com
www.3ds.com/SCN-June2012

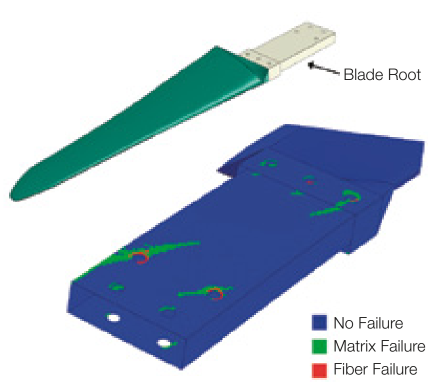
Composite Wind Blade Optimization with Helius:MCT for Abaqus/Standard

The evolution of the wind turbine blade is advancing at a rapid pace. Modern blades must be lightweight, efficient, and relatively easy to manufacture. Beyond these requirements, Z4 Energy Systems, LLC is developing a residential scale wind blade with a shape that is self-regulating. This is achieved using a unique composite layup that couples the bending and twisting behavior of the blade. Allowing the wind speed to "shape the blade" means that the efficiency of the blade can be increased in typical winds. In high winds, the blade protects itself by twisting into a feather configuration.

Z4 knew that their novel design would require an iterative process, but time and financial constraints precluded the common "build-and-break" design process. So the company turned to Dassault Systèmes Simulation partner Firehole Composites to simulate the response of their blade with Helius:MCT™, an advanced composites analysis plug-in for Abaqus/Standard.

Going Beyond First Ply Failure

Most composite failure methods employ a first-ply-failure (FPF) approach. In multi-axial laminates, the FPF is typically a matrix failure. This neglects the remaining capacity of the laminate resulting in an under prediction of the laminate strength. A progressive failure analysis, found in Helius:MCT, allows for simulation of not only failure initiation but ultimate structural failure.

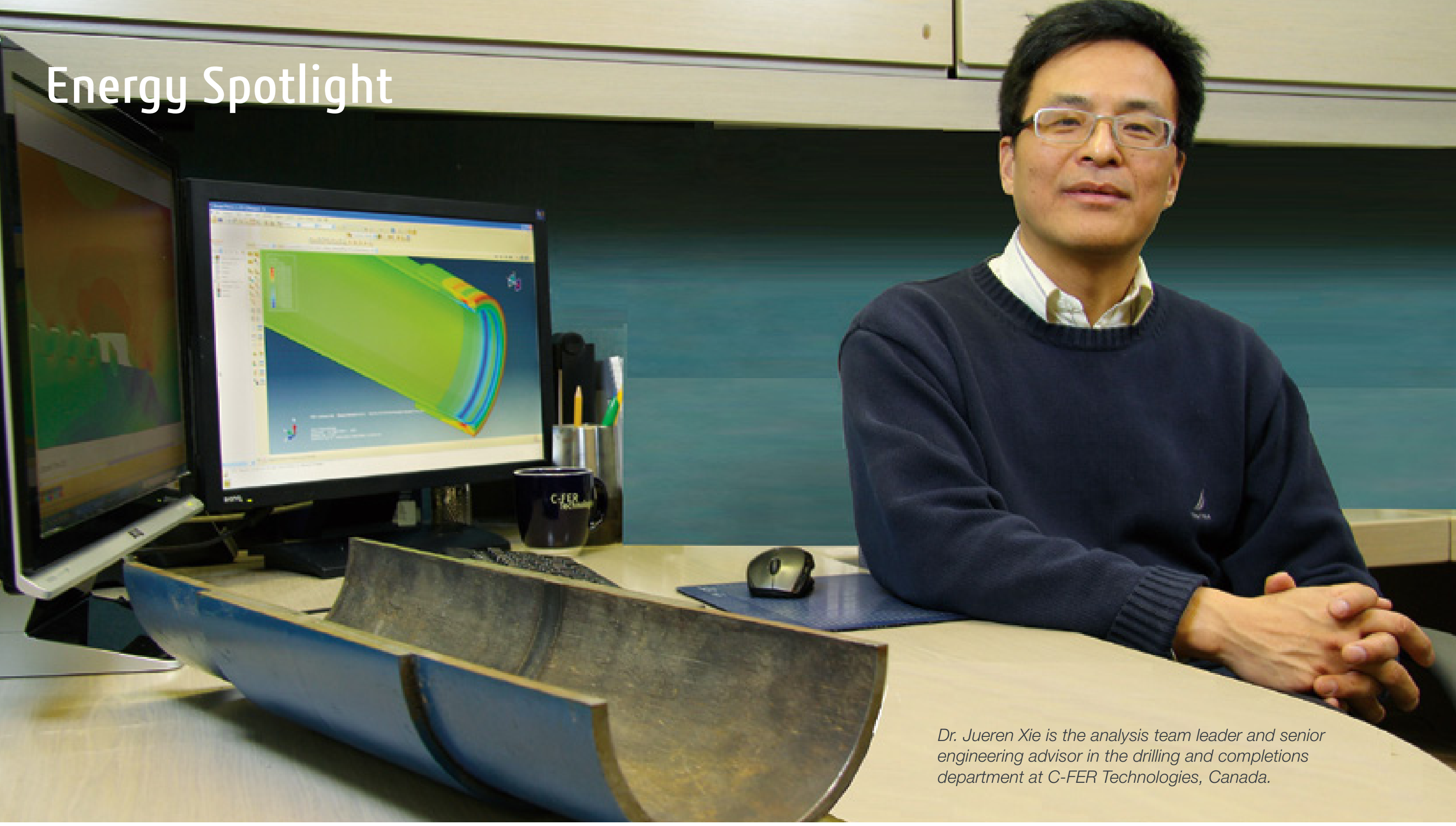


Composite wind blade optimization shows fiber and matrix failure.

One of Z4's load cases required the determination of the load capability of the root (the region of the blade that connects to the rotor hub). Using Helius:MCT, slight matrix cracking was first detected at a load ratio of 0.20, while ultimate failure of the root did not occur until a load ratio of 0.69. The ultimate load is a factor of 3.45 times larger than the FPF load! Had Z4 designed to FPF, their blade would have been overly conservative resulting in a heavier, more costly blade.

The added insight enabled the designers to further optimize the blade, and the ability to virtually test allowed them to test several load cases for multiple blade configurations. This substantially impacted the design cycle as actual building and testing each configuration was simply not feasible.

For More Information
www.firehole.com



Dr. Jueren Xie is the analysis team leader and senior engineering advisor in the drilling and completions department at C-FER Technologies, Canada.

Simulating the Physical Challenges of "Unconventional Oil"

Abaqus accurately predicts realistic performance of well designs in challenging oilfield applications

These days, oil wells are deeper, well locations less accessible, and the oil heavier or otherwise more difficult to extract. All of this has led to the development and testing of a host of technologies needed to reach, and recover, what is now called "unconventional oil."

Many of these technologies have been explored and improved upon by C-FER Technologies Inc. (C-FER). Originally the Centre for Frontier Engineering Research, it was founded in the 1980s as an industry-directed engineering, testing, and applied research center to solve problems posed in constructing and maintaining offshore and Arctic structures in Canada. In the 1990s, C-FER formed a wholly owned subsidiary, C-FER Technologies Inc., to bring its advanced engineering and testing services to the marketplace on a competitive, commercial basis.

Computer-aided engineering is key to oil well performance analyses

Today C-FER's structural, mechanical, petroleum, and reliability engineers conduct applied research and development, perform full-scale testing, and provide engineering consulting for the upstream oil and gas, and pipeline transmission industries, as well as other industries. "We have a longstanding tradition of rigorous physical testing for products," says Dr. Jueren Xie, senior engineering advisor for C-FER, "but we have nearly as long a history with computer-aided engineering (CAE) tools."

From the beginning, finite element analysis (FEA) software has been an important tool at C-FER. In 1994, as the company's FEA services met with increasing demand, they adopted Abaqus FEA. Projects conducted by C-FER using the software include design of full-scale tests, failure investigation, and design optimization.

"Proving out technologies for unconventional oil wells with FEA has long been one of C-FER's most important tasks," says Xie. Unconventional oil and gas typically refers to resources such as oil sands, heavy oil, oil and tight gas shales, deep and deep-sea reservoirs, and Arctic reservoirs located below thick layers of permafrost. "These wells frequently involve ancillary extraction technologies that place considerable loads on the wellbore equipment," Xie notes.

For example, thermal well technologies, such as Cyclic Steam Stimulation (CSS), with peak temperatures higher than 330°C, and Steam Assisted Gravity Drainage (SAGD), with peak temperatures higher than 220°C, have been widely used to produce viscous heavy oil and bitumen. Many of these applications use large diameter wells with complex three-dimensional trajectories to reach the target reservoirs.

The high temperature and/or high pressure often cause significant formation loading from the interaction of the wellbore equipment with the surrounding formation. This can potentially induce large deformations and changes in material properties, causing well mechanical failures, such as buckling, shear, and collapse; casing connection failures from parting or fluid-leaking; cement functional failures (cracking and the formation of fluid flow paths); horizontal well failures due to structural damage; serviceability failures (wellbore access and sand control); and in rare instances, wellbore leakage and blowout events.

Since the traditional stress-based design criteria used for conventional wells no longer apply, C-FER developed a strain-based design concept for designing unconventional wells. Incorporating initial well-completion designs and operational and field conditions, C-FER evaluates projects using both full-scale physical tests and numerical simulations. "Because such physical testing is generally costly, especially for qualification testing under a wide range of load scenarios, it is more economical and efficient to evaluate early-stage designs with FEA," says Xie.

Two recent examples where C-FER has used FEA to optimize wellbore designs in unconventional applications include coupled thermal-mechanical analysis of wellbore production and optimization of casing and slotted liner designs. "Both of these applications involve highly nonlinear response of wellbore equipments to production loads," Xie notes, "and in both cases, Abaqus helped to determine the realistic load response and to evaluate whether or not the field equipment would perform properly for the life of the well."

Case 1: Coupled thermal-mechanical analysis of wellbore production

The coupled thermal-mechanical analysis demonstrated that at a typical heating rate of 0.6°C/min on casing internal surface (i.e. about six hours to reach a peak temperature of 220°C), the cement sheath over the entire well depth would experience significant cracking, potentially compromising the primary function of the cement sheath hydraulic (see Figure 1).

However, the analysis also showed that heating rates slower than 0.6°C/min appeared to mitigate this cracking potential because slower heating rates allow sufficient heat to be transmitted to the formation, so that both the outer surface of the cement and near wellbore formation expanded, resulting in more confinement to the wellbore. The coupled analysis showed that slow heating rates would lead to less potential damage to the cement.

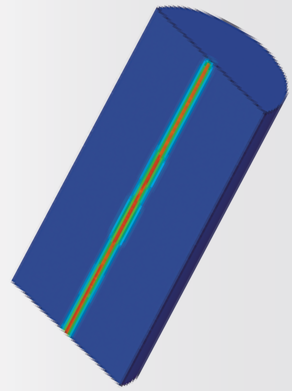


Figure 1. Simulation of a thermal well production showed significant temperature gradient from well center, suggesting high potential for cement cracking during heating (red for the high temperature, and blue for the ambient temperature).

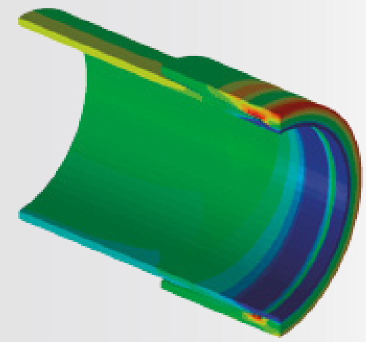
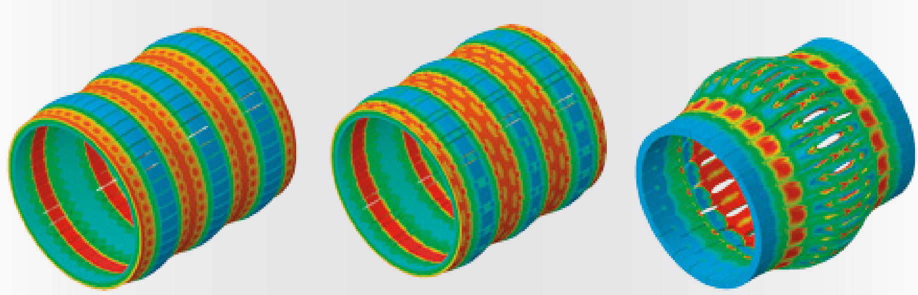


Figure 2. Simulation of a premium connection under curvature loading showed that the critical threads in the connection would experience alternating tensile (red) and compressive (blue) axial strain, causing potential connection fatigue failure under casing rotation loading.

Case 2: Optimization of casing and liner designs for thermal wells

Another area related to casing connections in thermal wells was the combined mechanical-thermal cycle fatigue life. Engineers created a 3D computer model for a "premium" casing connection design to analyze an initially axisymmetric structure subjected to the nonlinear, non-axisymmetric deformations associated with the bending rotation of the casing. The analysis results suggested that a 244 mm connection design could tolerate several hundred thermal cycles with the casing rotation (in a well with up to 12°/30 m curvature) and a somewhat larger number of thermal cycles without (see Figure 2).

With formation-induced cyclic shear loading, the analyses showed that the connection design could endure about one-fifth the number of thermal cycles with casing rotation and only about 8% more thermal shear cycles without. "The results clearly showed that cyclic formation shear loading was much more critical than thermal cycle loading," Xie points out.



Simulation of slotted liners revealed that the staggered (left) and gang (center) slot patterns had sufficient axial strain absorption capacities, while the overlapping (right) slot pattern had limited strain capacity by exhibiting "birdcaging" deformation mode, compromising the sand control functionality (red for high plastic strains).

For the examined scenario, the analyses also established that casing rotation had only a modest impact on fatigue life of the connection design. Giving support to the results, the predictions showed good agreement with physical test results published for connections of the same size and similar design.

Drilling down to FEA results

Advancements in the industry's understanding of design factors for unconventional oil wells is ongoing—for instance, a future study is anticipated wherein the potential synergy effects of plastic material deformation and corrosion mechanisms on the long-term integrity of casing connections will be explored—and Xie looks to continue to advance C-FER's research capabilities and expertise with FEA-based projects. "With the help of Abaqus, we've been able to gain a far greater understanding of complex nonlinear and multiphysics processes in these unconventional recovery applications," he says. "The software has helped us fine-tune existing designs and predict the behavior of new ones for clients, long before they begin to drill and operate the wells. Overall, FEA has played a key role in improving the safety of oil and gas extraction while also providing significant business benefits from reduced downtime and failures."

For More Information

www.cfertech.com
www.3ds.com/SCN-June2012

New Abaqus 6.12 Release—Now Available

The current release of Abaqus 6.12 delivers a number of powerful, user-requested enhancements for multiphysics, modeling and meshing, and performance. Key enhancements for solving complex multiphysics problems in Abaqus 6.12 includes a new magnetostatic analysis capability for modeling common electrical and industrial applications, such as magnetic recorders.

The new release also provides several performance enhancements including a novel substructure generation capability in the AMS eigensolver, which delivers significant performance improvements and reduces disk space requirements. In one example, performing a frequency analysis and generating a substructure for a powertrain model with 13 million degrees of freedom (DOF) using the AMS eigensolver in Abaqus 6.12 takes less than four hours, compared to 17 hours in previous versions.

Mesh parts can now be converted into geometry with Abaqus 6.12. This enhancement is useful for creating geometry from deformed meshes (e.g., a stent), creating an interior acoustic cavity inside of a vehicle, as well as a number of other industry applications.

Additional Key Features of Abaqus 6.12:

Multiphysics:

- **Nonlinear Viscoelasticity:** Improved matches with experimental data are now possible using nonlinear viscoelasticity based on the parallel network approach. This enhancement is particularly useful in applications involving polymer materials in the automotive and life science industries.
- **Smoothed Particle Hydrodynamics (SPH):** Abaqus provides a new capability that allows continuum elements to be converted into particles on-the-fly based on user-defined criteria. This enables an entirely new set of workflows to be performed in Abaqus.
- **Porous Media Flow:** New support for modeling fluid flow through porous media, such as water seeping through soil or a paper filter. This enhancement has a wide variety of useful industry and environmental applications, such as packed-bed heat exchangers, heat pipes, and thermal insulation.
- **Non-Newtonian Viscosity:** Simulation of flows involving fluids found in many industrial, technological, and biological applications (e.g., foams, emulsions, dispersions, and suspensions) can now be achieved using the new non-Newtonian models in Abaqus.

Modeling & Visualization:

- **Mesh-to-Geometry:** Mesh parts can now be converted into geometry. This enhancement is useful for creating geometry from deformed meshes.
- **Boundary Layer Meshing:** Wedge elements can now be generated along selected boundaries of a part or a cell, and the interior is filled with tetrahedral elements. Adding layers of small elements along the walls allows improved analysis of boundary effects in fluid flow and heat transfer analyses.
- **Node Dragging:** Nodes can be repositioned with a new "dragging" feature.
- **Abaqus/Viewer:** Visualization functionality in Abaqus/Viewer has been extended to visualize preprocessing modeling entities, such as interactions, pre-defined fields, loads, and more.

Performance:

- **VCCT Enhancements:** Performance and robustness improvements were made to the Virtual Crack Closure Technique (VCCT) in Abaqus/Standard, which allows the delamination of composite structures to be simulated much more quickly.
- **Contact and Constraints:** New edge-to-feature enhancements allow handling of contact between edges and surfaces, and provide value in a broad range of applications, including electrical connections.
- **Adaptive Mesh Refinement for CEL:** The adaptive mesh refinement for Coupled Eulerian-Lagrangian (CEL) improves performance in models in which large deformation exists, such as airbag deployment, by automatically refining and de-refining the mesh as needed in Abaqus.
- **Direct Sparse Solver using Multiple GPGPUs:** Support for multiple GPUs in the direct sparse solver provides faster execution on computers with multiple GPU cards.

Add-ons:

- **CATIA V6 Associative Interface (AI):** The CATIA V6 AI provides a one-click option to easily transfer part or assembly models to Abaqus/CAE, while retaining the attributes defined in Abaqus/CAE during the re-import of geometry from CATIA V6.
- **Translator Improvements:** The existing fromnastran and fromansys translators were enhanced with new robustness, functionality, and diagnostics to support the growing interest in users moving from ANSYS and NASTRAN to Abaqus. Also, a new fromdyna translator is available to convert DYNA models into Abaqus.

For More Information
www.3ds.com/abaqus

SIMULIA V6R2013 Extends Capabilities for Advanced FEA and Multiphysics

The SIMULIA V6R2013 product portfolio provides a scalable and integrated suite of realistic simulation and simulation management applications. The product suite, which includes DesignSight, CATIA Analysis, ExSight, and Simulation Lifecycle Management (SLM), enables sharing of simulation methods, models, and data, which facilitates close collaboration between designers, simulation specialists, managers, and extended product development teams.

The following are some of the enhancements available in V6R2013:

ExSight

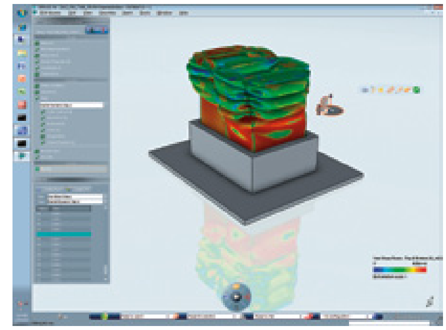
- Shell elements, important in aerospace applications, are now supported and allow users to vary thickness and other properties in different areas within the same model.
- Material properties for composites can now be imported from CATIA Composites Part Design (CPD) for efficient modeling and design updates of composite parts.
- High-speed events and deformations, such as mobile device drop tests, can be simulated using automatic general contact, eliminating the need to manually pair contact surfaces.
- Numerous modeling enhancements including additional types of loads and restraints have been added, increasing the range of simulations that can evaluate the real-world behavior of designs.

DesignSight

- The "digger" tool has been extended to evaluate results, enabling users to rapidly peel away exterior layers by clicking through the model.
- Users can now group and dock viewing panels within the application for a more organized working environment and better use of on-screen space.
- Report generation has been enhanced to allow users to select specific information they want to include and automatically create professionally presentable Word documents. This feature enables quick sharing of meaningful results to drive collaborative decision making.

SLM

- Data retention capabilities allow users to eliminate unneeded files to save space and reduce data complexity.
- Users in multiple departments can now work collaboratively with the new task sign-off feature.



Powerful and high-performance results visualization capabilities in ExSight provide the user with best-in-class experience for results postprocessing and decision making.

- A "My Simulations" interface eases new user adoption of SLM by providing capabilities to create and run simulations.
- Product structures are now accessible through a filtered view, allowing users to easily share content and improve design collaboration.

For More Information
www.3ds.com/simulia/V6

Isight and SEE 5.7 Deliver Key Enhancements

The current release of Isight and SIMULIA Execution Engine (SEE) 5.7 deliver significant new features and enhancements in the areas of licensing, integration, optimization, postprocessing, and infrastructure.

New features for modeling and simulation integration in Isight 5.7 include the Isight Distributed Execution Engine, which is an in-the-box Distributed Resource Manager (DRM) that leverages Fiper technology to distribute the sim-flow computations over designated hardware stations.

A number of postprocessing enhancements are available, including a self-organizing map to cluster Isight datasets by using component history data. This allows users to gain insight into a combination of characteristics that usually go together, such as the combination of high-fuel efficiency, low-engine displacement, and low-vehicle weight.

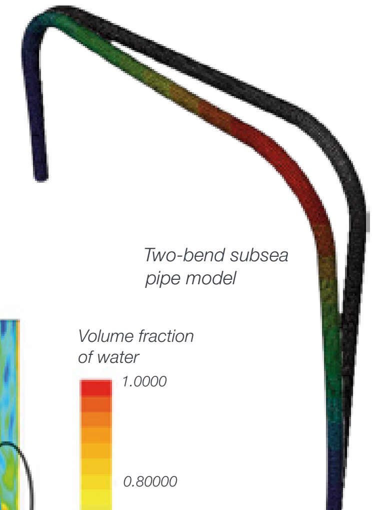
The SEE 5.7 release introduced a new capability to end a single work item on the dashboard, as opposed to ending an entire job. This enhancement addresses the problem that an application run by a component on an SEE/Station can get 'stuck', and the only option was to end the entire job in order to free resources.

Additional Key Features in Isight 5.7:

- **Isight Token-based Licensing with FlexNet:** Provides access to a wider range of Isight components. (Note: Isight and Abaqus are now supported on the same license server.)
- **Automatic Database Clean-up:** Set automatic expiration dates for jobs in the database and perform manual cleanup of jobs on a specified job date.
- **Abaqus Component Enhancements:** Allows control of CATIA V5 parameters directly in the Abaqus component.
- **SIMULIA R&D Developed V5 Component:** Integrate CATIA V5 applications in sim-flows as part of every Isight distribution and SEE execution.
- **Pack-and-Go Utility:** Collects copies of all disk files in a sim-flow in one central location reducing user error and effort when sharing sim-flows.
- **Dependent Variable Sampling by Design Matrix Filtering:** Filter out user-defined constraints of DOE factors.

For More Information
www.3ds.com/isight-see

Co-simulation of Two-phase Flow in an M-shaped Subsea Piping Component



Components in a subsea production system require different types of pipelines, such as a jumper (a short U-shaped section of pipe to connect one pipeline to another), to transport fluids. The internal flow in pipes involves an interaction between fluid and structure, which is important to understand since their interaction can generate high amplitude vibrations, also known as "flow-induced vibration." Consequently, these vibrations can result in fatigue damage of the structure. This phenomenon has become a great concern in the oil & gas industry where subsea jumpers are exposed to this type of vibration when transporting production fluid. The industry is currently putting a lot of effort into investigating vibration-induced fatigue cases to prevent negative effects on revenue, production, environmental safety and health.

Production fluid flowing through subsea components is usually a mixture of oil, gas, and water. When a gas and liquid flow through a pipe, a potential slug flow is formed, and consequently this generates vibration issues in the structure. A slug is an intermittent flow in which long gas bubbles are separated by chunks of liquid causing large pressure fluctuations and corrosion. The amplitude of the vibrations increases and creates a potential risk of failure of the pipe when the natural frequency of the structure is close to the frequency of the slugs as they are transported along the pipeline.

To assess the impact of this type of flow in the structure, Leonardo Chica, a researcher at the College of Technology, University of Houston, conducted an analysis of the fluctuation of stress with time to predict the number of cycles that the jumper can withstand without failure. The best option to represent this fluid-structure interaction problem is to perform a two-way coupling simulation or co-simulation between Abaqus and a computational fluid dynamics (CFD) program, such as CD-adapco's STAR-CCM+. In this process, the pressure fluctuations are exported from the CFD tool into Abaqus, and then Abaqus computes the stresses and displacements. These displacements are exported back to the CFD program and the cycle starts again. Both programs run simultaneously and exchange data at each time step.

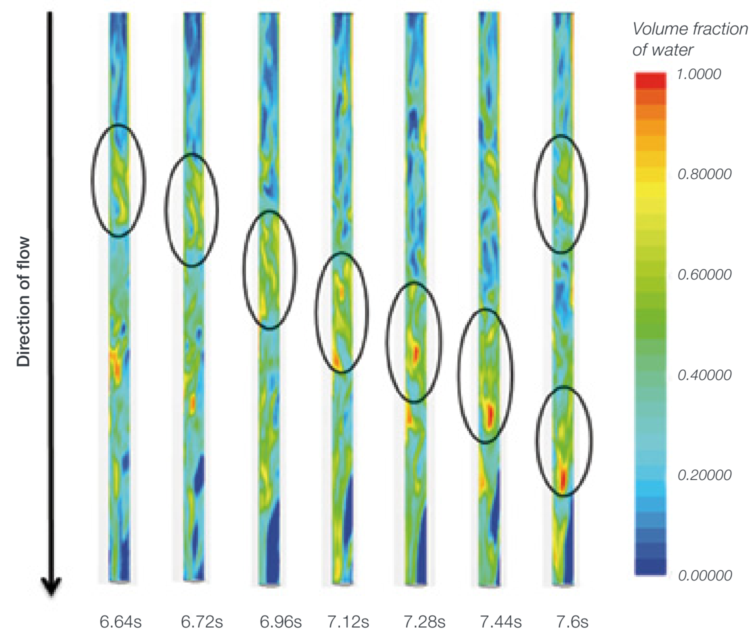


Figure 1. Slug travelling in vertical section of two-bend model.

To set up the analysis, we imported the CAD model and then extracted the natural frequencies in Abaqus. Next, the simulation was set up in the CFD program with the appropriate mesh and physics, and the co-simulation was initiated to communicate the CFD code to Abaqus. After initializing the solution with 50% air and 50% water, the results showed that irregular slugs are developed at the vertical section of a two-bend model. Slug frequency was determined to be 1.0417 Hz (see Figure 1), which is close to the fundamental natural frequency (1.079 Hz), so amplitude of the vibrations could be intensified and the fatigue life of the jumper might be reduced. In this case, the co-simulation results of the von Mises stress vs. time graph obtained in Abaqus show a sinusoidal pattern with a response frequency of 0.167 Hz. Based on the material's S-N curve, fatigue life is infinite (below the fatigue limit curve), due to the small stress range, and the two-bend structure can withstand cyclic loading from the pressure fluctuations of the two-phase flow.

In this initial investigation, only water-air mixture was simulated to understand the behavior of this two-phase flow and to determine the response in the jumper.

For future work, oil-gas-water flow will be simulated and analyzed to compare with experimental results. The Fluid-Structure Interaction (FSI) analysis should also be extended to include the entire jumper model in order to draw solid conclusions about fatigue damage.

This type of FSI co-simulation is becoming more valuable in subsea engineering to understand how the internal or external flow affects the fatigue life of subsea components. The Abaqus co-simulation capability for FSI allows the user to perform a co-simulation between Abaqus and third party software, such as STAR-CCM+. One of the advanced features of Abaqus is to perform either a one-way coupling or two-way coupling simulation depending on the magnitude of the displacements. This selection would be made on a case-by-case basis to achieve a balance between computational cost and accuracy of the results. Either way, co-simulation for FSI is rapidly becoming a requirement in the subsea industry to provide greater reliability, safety, and performance in complex subsea systems.

For More Information
www.tech.uh.edu
www.3ds.com/SCN-June2012

Multi-disciplinary and Multi-point Design Optimization of a Centrifugal Compressor Impeller

Recently Mehrdad Zangeneh, professor of thermofluids in the Department of Mechanical Engineering at University College London (UCL), has been considering aero/mechanical issues arising from design of high-speed turbomachinery, such as centrifugal compressors or radial turbines used in turbochargers. The major design bottleneck in many of these applications is the iteration between the aerodynamic design, where optimum performance is required at multiple operating points, and mechanical design. One way to reduce the aero/mechanical design time is to use multi-disciplinary design optimization (MDO). Conventional approaches for MDO based on parameterization of blade geometry usually require a large number of design parameters.

However, by parameterizing the blade using the blade loading and 3D inverse design code TURBOdesign1 by Advanced Design Technology to generate the blade geometries, the optimization can be performed efficiently and with a very small number of design parameters. Typically, five to 10 loading parameters can cover as much design space as 30 to 100 geometrical parameters, resulting in a significant reduction of computational time and costs. Furthermore, the 3D inverse design code automatically satisfies the specific work at the design flow rate without the need to use any constraints in the optimization.

In an application of this approach, the Eckardt centrifugal compressor impeller was used. First, the compressor was analyzed by means of Computational Fluid Dynamics (CFD) in CD-adapco's STAR-CCM+ and Abaqus Finite Element Analysis (FEA) software. The results compared to the available test data. This provided initial information on the stage performance to be used as reference for improvements and validation of the numerical analysis.

In this study, only four design parameters were used to represent the 3D blade geometry. Isight was then employed to generate a Design of Experiments (DOE) table and the resulting 22 designs were analyzed with both multi-point CFD and FEA. An important aspect of the optimization process is the capability of the different codes to operate in a seamlessly integrated manner. By making use of the existing compatibility between the different blade generation, CFD, FEA, and optimization codes, this process only required minor human intervention.

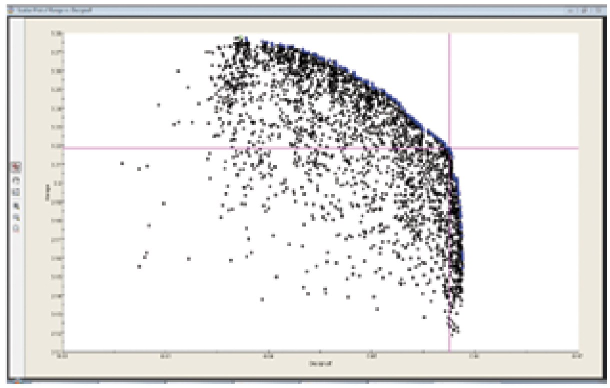


Figure 1. Pareto front showing minor improvements in one performance parameter can significantly affect the contrasting performance parameter.

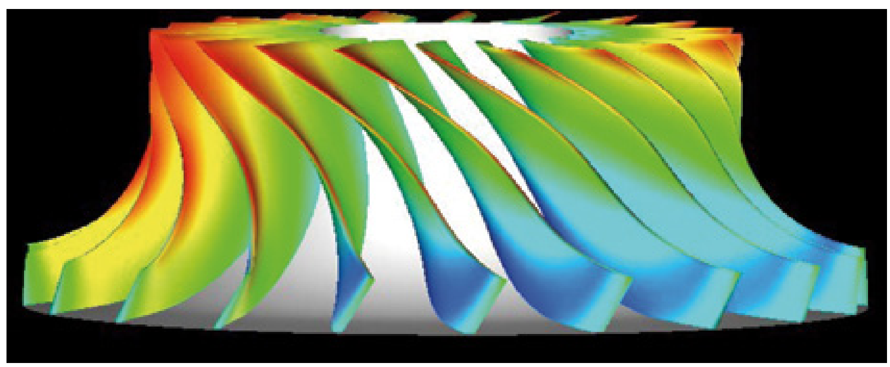


Figure 2. The geometry of the resulting optimized impeller provides the best compromise between design efficiency and flow range and subject to the required stress level.

After the CFD and FEA results were inserted into the resulting DOE table, Isight generated the Response Surface Model (RSM), an efficient mathematical regression method that correlates each performance parameter with different design parameters. Performance parameters can represent any type of data. Then designers can use their own validated methods to evaluate aerodynamic performances, mechanical reliability, production time, and manufacturing costs.

Once the accuracy of the RSM is tested and confirmed, it is possible to run a MDO using the RSM. In this case, a Multi-objective Genetic Algorithm (MOGA) is used to find the Pareto front, which is a trade off between different contrasting objectives subject to certain constraints. MOGA requires a large number of performance parameter evaluations. By running the MOGA on the fast Response Surface approximation, the performance parameters are evaluated very quickly and without the need for expensive CFD or FEA computations. The point selected on the resulting Pareto front (marked with the pointer in Figure 1) represents the best compromise between the operating range and design impeller efficiency. The resulting impeller geometry is shown in Figure 2.

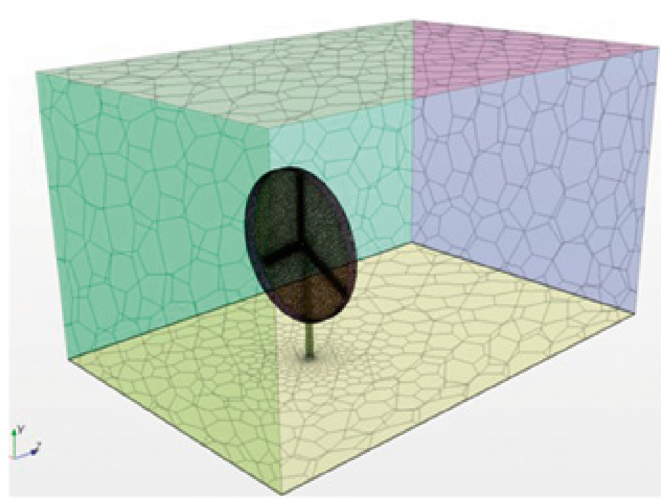
The proposed optimization process is generally applicable to all types of turbomachinery for all design requirements, potentially providing turbomachinery manufacturers with the capability to create initial designs customized to the specific customer requirements very quickly.

For More Information
www.ucl.ac.uk
www.3ds.com/SCN-June2012

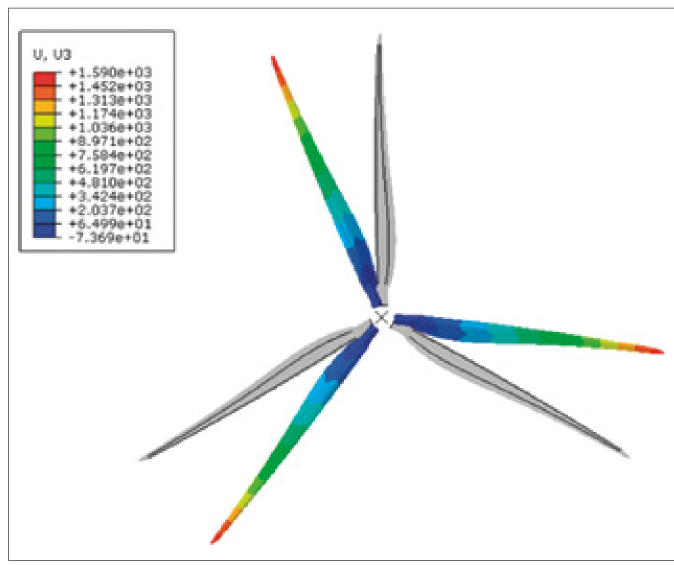
Fully Coupled Fluid-Structure Interaction Analysis of Wind Turbine Rotor Blades

Over the past few years, the use of wind turbines as a source of renewable energy has grown dramatically. While the basic operating principle of a wind turbine is simple, the detailed aerodynamic and mechanical behavior of the systems during operation is quite complex. Of particular interest to blade designers is the coupled interaction between the rotor aerodynamics and the response of the structure. Including this interaction effect in the design process will allow for a better prediction of the blade loading and deflection.

A newly published SIMULIA Technology Brief describes the process to perform a fully coupled fluid-structure interaction (FSI) solution using Abaqus and STAR-CCM+ from CD-adapco. In this example, Abaqus/Standard is used for the structural simulation of the blade and STAR-CCM+ is used for the fluid simulation.



CFD mesh representation.



Bending deflection (mm), sequentially coupled FSI analysis.

Wind Turbine Example

It is well known that the airflow around a wind turbine blade induces a number of complex aerodynamic phenomena. As a result, accurate estimates of the blade service loads are not easily predicted. A number of design methods have evolved over the years, each using differing levels of simplification and approximation.

Computational fluid dynamics (CFD) is a powerful analytical technique and is regularly used in the design of wind turbines. By directly coupling a CFD analysis to a structural finite element analysis, the strengths of both tools can be applied simultaneously, allowing designers to capture the real-time interaction of fluid and structural behavior.

In this study, the structural model of the blades is created in Abaqus/CAE. A single blade is approximately 48m in length and modeled with 18,471 S4R composite shell elements. The blades are connected to a rigid hub with a kinematic coupling constraint.

The CFD model contains a rotating region in the immediate vicinity of the blades, and a fixed region far away from the blades. The two regions interact through a sliding boundary and the dynamic mesh capability in STAR-CCM+. Partitioning the CFD domain into two regions permits use of appropriate mesh densities for different regions of flow significance. The complete three-step workflow, building from individual Abaqus-only and CFD-only analyses to the definition of the fully coupled FSI analysis, is summarized in the Technology Brief.

A fully coupled bidirectional FSI analysis allows more accurate assessments of the blade stresses and dynamic displacement. A dynamic analysis yields higher magnitudes of stress compared to the one-way sequential static analysis. The oscillatory displacement of a blade tip through a single rotation is a dynamic behavior that can be studied only in fully coupled FSI analysis. Peak pressure distribution in the wake region for the deformable blade is on average 10% higher than that of a rigid blade. The analysis shows the contours of pressure and velocity magnitude in the air and compares the torque at the turbine hub for the CFD-only 6-DOF and fully coupled FSI analyses. Note that the inclusion of blade flexibility and fluid-structure interaction modifies the time history and magnitude of the torque.

Consistent with the higher torque, the out-of-plane force distribution along the leading edge of the deformable blade in a fully coupled FSI analysis is higher, on average, when compared to that of the rigid blades.

Download the complete Technology Brief to learn more about the geometry, model, and analysis approach to achieve higher-fidelity simulation results with fully-coupled multiphysics simulations.

Acknowledgement

SIMULIA would like to thank CD-adapco for collaborating on the CFD portions of this Technology Brief.

For More Information

www.cd-adapco.com
www.3ds.com/SCN-June2012

2012 SIMULIA Regional User Meetings

Our Regional User Meetings (RUMs) provide a valuable forum for discovering and understanding how experts in engineering and academia are applying the latest simulation technology and methods to accelerate and improve product development. Attend the upcoming RUM in your area to learn about the latest enhancements to our products and the ongoing strategy of SIMULIA.

Learn more at: www.3ds.com/rums

Americas

September 25............Beechwood, OH
September 26–27........Chicago, IL
September 27............Providence, RI
October 17–18...........Plymouth, MI
October 23.................Houston, TX
October 23–24...........Mississauga, Ontario
October 25.................Santa Clara, CA

Europe/Middle East/South Africa

September 17–18........Hamburg, Germany
September 20.............Helsinki, Finland
September 25–27........Cheshire, United Kingdom
October 4–5...............Poland
October 16–17...........Czech Republic
October 16–17...........Velizy, France
November 5–6............Linz, Austria
November 8–9............Oud-Turnhout, Belgium
November 20..............Madrid, Spain

Asia Pacific

October 8............Bengaluru, India
October 11–12......Korea
October 15–16.....Japan
October 18–19.....China

Contours of pore pressure

New Technology Brief

Abaqus/Standard Coupled Simulation of Thermally Assisted Gravity Drainage of an Oil Sand Formation

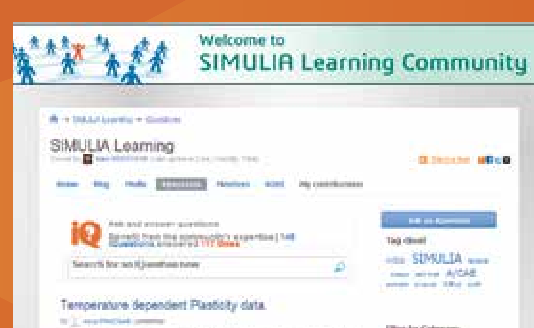
Abaqus/Standard provides an analysis capability that allows for coupling between heat transfer, pore fluid flow, and displacement. In this Technology Brief, we demonstrate how this capability can be used to simulate the thermally assisted gravity drainage of an oil sand formation.

Download the Tech Brief: www.3ds.com/SCN-June2012

Join the SIMULIA Learning Community

This online community provides a venue for you to ask and answer questions, read and make comments on blog posts, view tutorials, download tech briefs, papers, and case studies, access regional news and training schedules, and much more.

Sign in today at: www.3ds.com/simulia-learning



DASSAULT SYSTEMES

Delivering Best-in-Class Products

CATIA
Virtual Product Design

EXALEAD
Information Intelligence

SOLIDWORKS
3D for Professionals

NETVIBES
Dashboard Intelligence

SIMULIA
Realistic Simulation

3DSWYM
Social Innovation

DELMIA
Virtual Production

3DVIA
Online 3D Lifelike Experiences

ENOVIA
Global Collaborative Lifecycle Management

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes, in all industries, in more than 80 countries. For more information, visit www.3ds.com.

CATIA, SOLIDWORKS, SIMULIA, DELMIA, ENOVIA, EXALEAD, NETVIBES, 3DSWYM, 3DVIA are registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA