DASSAULT SYSTEMES

SIMULIA Community News

September/October 2012
www.3ds.com/simulia

Caterpillar Rolls to Success with FEA

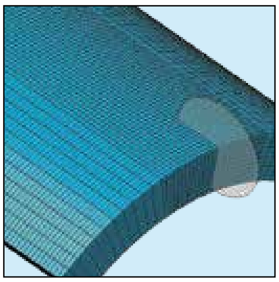
SIMULIA

DASSAULT SYSTEMES

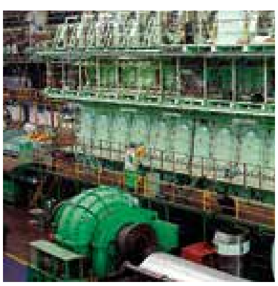
SIMULIA
Community News

September/October 2012
www.3ds.com/simulia

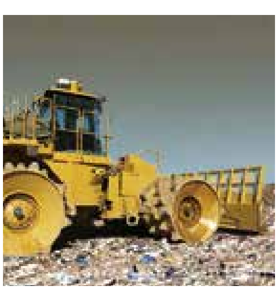
Spotlights



4 Tenaris
Digs Deep into Hydraulic Cylinder Performance



6 MAN Diesel & Turbo
Realistically Simulates Engine Down to the Last Detail



12 Caterpillar
Rolls to Success with FEA

On the cover: Liqun Chi, Senior Engineering Specialist, Product Development & Global Technology Division

Greg Zhang, Engineering Specialist, Industrial & Waste Group

Also In This Issue

3 Welcome Letter
Dale Berry, Sr. Director of Technical Marketing, SIMULIA

8 IE Applications
Jack Cofer, Sr. Technical Marketing Specialist, Industrial Equipment Industry, SIMULIA

10 Case Study
Mazda Balances Performance and Weight in a Steel Car Body

14 Product Update
What's New in SIMULIA V6R2013x

15 Technology Brief
Simulating Pipeline Rupture

16 Case Study
NSE Composites Validates Design that Captures 12% More Energy

19 Alliances
- NUMECA International
- Veryst Engineering, LLC

20 Academic Update
- University of North Carolina at Charlotte Simulates Wear of Tillage Tools
- National University of Ireland Galway Models Cortical Bone Fracture Using XFEM

22 Tips & Tricks
How to Use Assembled Fasteners in Abaqus/CAE

23 News and Events
- SIMULIA Community Conference–Save the Date!
- Inside the Learning Community
- New Technology Brief
- New Customer Testimonial Video

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

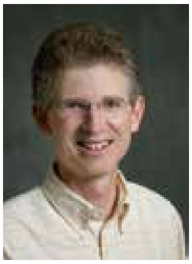
Contributors
Liqun Chi and Greg Zhang (Caterpillar),
Mihaela Cristea (Tenaris Dalmine),
Tore Lucht (MAN Diesel & Turbo),
Takehisa Kohira (Mazda),
DM Hoyt (NSE Composites),
Emer Feerick (National University of Ireland Galway),
Gaurav Goel (UNC at Charlotte),
Pierre-Alain Hoffer (NUMECA International S.A.),
Jorgen Bergstrom (Veryst Engineering, LLC),
Parker Group,
Dale Berry, Jack Cofer,
Lori Bonynge, Jill DaPonte,
Lance Hill (SIMULIA)

Graphic Designer
Todd Sabelli

OCTOBER_SCN_V12_VOL 2

SIMULIA

Industrial Equipment Innovation



It's a pleasure to be writing for *SIMULIA Community News* (SCN) once again. My last note appeared in the October 2010 edition. A lot has happened since then, both within the Dassault Systèmes community and for me personally. I'm proud to say I just sent my oldest son off to college (chip-off-the-block engineering major), and it's fascinating to compare notes about our experiences. How things have changed!

My freshman year was traditional—sitting in very large, impersonal lecture halls filled with quaking freshman, many of whom wouldn't make it through the first year. But if you read the books, went to class, and passed the tests, you'd be okay. My son's experience is nearly the opposite; there's lots of large and small group interaction and projects already starting in his freshman year. Individual work within a collaborative environment is the key. Facebook, LinkedIn, Twitter, and other online tools are not just for fun; they're essential in keeping up with the very fast pace of instruction. In short, engineers are being taught about social communities from the very beginning of their education.

Within SIMULIA we have a keen focus on our global community of users, as well as on smaller groups who have an equal interest in sharing their applications or industry workflows. This effort is something we've done in the past, but our reach is expanding. We're working hard to leverage traditional face-to-face meetings, our online Learning Community, customer support website, and social media. The goal is to stay in daily contact with everyone in our user community and, just as important, help our members connect with each other.

While we're broadening our interactions, the role of our SIMULIA Technical Marketing group remains the same—to help users recognize and maximize the value of simulation workflows in their industries through knowledge and awareness building.

This issue of SCN has a special focus on industrial equipment (IE). While most industry labels are self-explanatory, allow me to define IE. Basically, the industrial equipment industry includes those companies that manufacture industrial products *bought by other companies*. The italics are to distinguish IE from Consumer Goods and Consumer Packaged Goods, which cover companies that manufacture products *bought by consumers*. IE includes manufacturing equipment and machinery (pumps, valves, pipes), turbomachinery, agricultural and construction equipment and vehicles (combines, motor graders), diesel engines, metal processing and forming equipment, and general industrial equipment (elevators, forklifts).

At the core of our IE community activity is technical lead Jack Cofer. (You can read about our application strategy on page 8.) Jack has been traveling widely throughout many regions of the world in the last few years so many of you may have already met him at SIMULIA meetings, professional conferences, or at your own companies.

The user case studies in this issue provide excellent examples of the spectrum of companies that constitute the IE industry: **Caterpillar** accurately models the behavior of mobile compactors and landfill waste; **Tenaris** analyzes hydraulic cylinder performance with Abaqus FEA crack simulation; and **MAN Diesel & Turbo** uses the eXtended Finite Element Method (XFEM) capability for a weld redesign against fatigue cracks. These three customers represent the wide variety of simulation applications and workflows within industrial equipment.

We invite you to be an active member in our growing and vibrant global community. If you haven't yet joined the online SIMULIA Learning Community (https://swym.3ds.com/#community:73/home), you should check it out. It's a great resource for tutorials, product info, and interesting blog posts, and is another way to further strengthen your professional network. I look forward to seeing many of you at the 2013 SIMULIA Community Conference in Vienna, Austria.

Dale Berry
Sr. Director of Technical Marketing
SIMULIA

Earth mover with hydraulic cylinders on boom, arm, and bucket.

Tenaris Digs Deep into Hydraulic Cylinder Performance with Abaqus FEA Crack Simulation

In the hands of a skilled operator, an earth mover makes construction-site work look easy. The smooth flow of the shovel as it digs into the soil and lifts its heavy load seems effortless. But much like a human arm with interconnected bones, muscles, and tendons, this labor-saving machine must harness a complex array of parts to perform in perfect synchrony.

Key to the earth mover's fluid movements is an array of hydraulic cylinders, located along the boom, arm, and bucket, that act as actuators to convert hydraulic energy into mechanical energy (similar technology is also used for cranes, presses, and industrial machinery). Filled with pressurized oil that transmits force to move a piston, each cylinder barrel (housing) is a seamless steel tube designed to resist fatigue from the very high number of operating cycles that it endures.

Hazards on the job site

However, operator error, accidental shovel overloads, and accidental impacts from falling debris are regular occurrences on a job site. So Tenaris, a leading supplier of seamless tubes for earth-moving machinery manufacturers worldwide (customers include Caterpillar, Volvo, LT Komatsu and Dong Yang), devotes considerable R&D resources to anticipating such events. By studying the effects of a wide range of operational and environmental stresses on a cylinder's service life, Tenaris can fine-tune material design to best tolerate such extremes and improve safety for thousands of machine operators worldwide each year.

"The failure mode and the limit load beyond which a tube rupture can occur are extremely important to identify," says Mihaela Cristea, a researcher in the structural integrity department at the Tenaris R&D center in Dalmine, Italy, one of four such centers around the globe. "Our OEM customers are looking for high-yield strength above all, so fracture toughness is the key parameter for ensuring integrity and safety."

"Leak" is preferable to "break"

Fracture toughness plays a critical role in the 'leak before break' behavior that is desirable in order to avoid the hazards that can result from a sudden, brittle fracture, says Cristea. "In all situations ductile fracture, or cracking, is usually preferred for several reasons. Unlike brittle fracture, which occurs rapidly without warning, ductile materials plastically deform, slowing the process of fracture and providing a window of time to act."

> We aim to reduce the number of long and expensive full-scale tests.
>
> **Mihaela Cristea**, Researcher, Tenaris

Particularly for hydraulic cylinder applications where the variable loads induce fatigue and the environment can be very severe (low temperatures decrease fracture toughness), the ideal housing combines high-tensile properties and fatigue resistance with good toughness. To reach these materials goals, Tenaris has been developing new high-toughness (HT) grades of steel from which they can manufacture cold-drawn precision tubes with improved combinations of toughness and strength.

New materials must withstand both internal and external forces

The primary stress on a hydraulic cylinder during normal operation comes from the internal pressure generated by the oil. This is much greater around the circumference of the tube (hoop stress) than along its length (the longitudinal stress is only about half the hoop stress). So as Tenaris develops its new materials, they look to maximize this circumferential yield strength. But, in addition, they need to demonstrate that each material can also withstand a wide range of physical stresses that can happen unexpectedly on a job site. "Critical defects, from which fatigue cracks can initiate, occur on the external surface of the tube," says Cristea.

To prove out the performance and quality of their products, the Dalmine R&D center has been conducting extensive full-scale testing of its hydraulic cylinder products for many years. This process begins with detailed mechanical characterization and fracture tests on tube materials. Then artificial defects are manufactured on the external surfaces of full scale tubes by means of electro-discharge machining. The tubes are then pre-cracked to different depths, capped, and chilled to minus 20 degrees Centigrade. Finally, they are pressurized with an alcohol water mix (aluminum filler bars are inserted inside the tubes to take up space and thus avoid extreme volumes of high-pressure fluid being released during bursting) and subjected to burst tests.

"With this 'build and break' test program we have been able to demonstrate the performance of our products to our customers in a way that's close to the real-service behavior of the components," says Dalmine researcher Marco Spinelli. "But the test regimen is complex and quite expensive. We wanted to find out if numerical models could effectively replicate our experimental results, reduce our costs, and save us time."

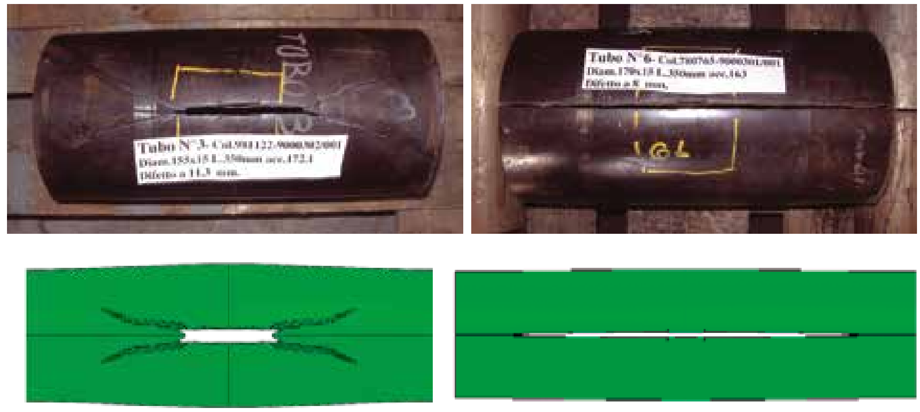
Going deeper than stress and structural analysis to 3D crack prediction with Abaqus FEA

So the team turned to Abaqus unified finite element analysis (FEA) for realistic simulation of their fracture toughness testing regime. They were already familiar with the power of FEA, notes Spinelli, using it for stress distribution analysis in welded-joint applications and structural analysis of complex components. "The suite of software applications available with Abaqus is widely used for product design in our R&D center, mainly due to its pre- and post-processing advantages," he says. "We felt we had the tools in hand for deepening our inquiries into the performance of our new materials."

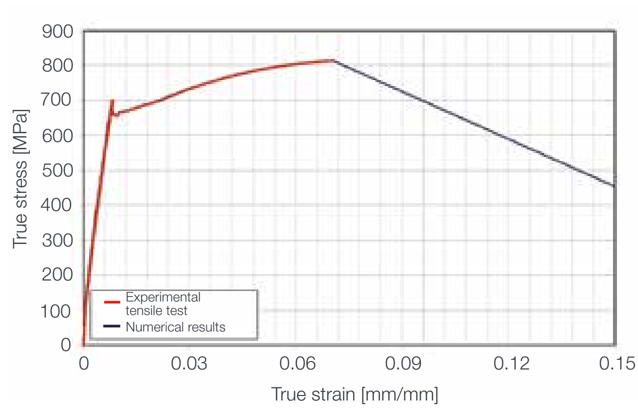
Since they already had recorded a great deal of accurate real-world test data, the group was able to create very precise FEA models of their cylinders using the materials definitions, geometries, crack dimensions, boundary conditions and burst pressures that were identified during their previous experiments. While the focus was on HT materials, low toughness material models were also examined to evaluate the overall performance of the software for predicting failure. The Tenaris team decided to try two methods in Abaqus for 3D crack modeling: ductile/fracture in Abaqus/Explicit and the new eXtended Finite Element Method (XFEM) capability in Abaqus/Standard.

Within Abaqus/Explicit, the team found they could take advantage of tube symmetry to create 1/8 models and reduce simulation time. Pressure was applied to the inner surface of the model, ramping up from zero to burst (1600 bar). Crack occurrence was simulated by employing 'element deletion' to signal when the plastic strain in a particular part of the model reached the critical value previously identified through the real-world burst tests: When the failure criterion is met for a particular element, that element drops out of the model, leaving a space behind. The series of connected, deleted-element spaces creates the appearance of the crack.

The Abaqus/Explicit simulation phenomenon is essentially static until the opening of the initial crack, after which the analysis becomes dramatically dynamic. Just as in the case with the real-world tests, the models demonstrated the more desirable ductile-failure mode when the material was HT steel and a brittle-failure mode with low-toughness steel. "When we compared our simulations with the real-world tests, we had excellent results predicting the failure modes for the cases we were investigating," says Cristea.



High-toughness (HT) steel burst test and Abaqus/Explicit model (left upper and lower) and low-toughness steel and Abaqus/Explicit model (right upper and lower).



Strong agreement between experiment and Abaqus FEA (numerical) results under tensile tests.

XFEM provides advanced crack modeling capabilities

After a series of Abaqus/Explicit modeling exercises, the team ran a final test case using XFEM. A relatively new addition to the Abaqus Unified FEA suite, XFEM, currently available in Abaqus/Standard, allows the simulation of crack propagation along an arbitrary path, even cutting across element boundaries in an FEA model. Based on a solution-independent crack initiation and propagation path, XFEM does not require the mesh to conform to geometric discontinuities, thus requiring less mesh refinement in the neighborhood of the crack tip.

"We obtained similar results to the ductile damage module with XFEM," says Cristea. "Going forward, implementing the XFEM capability in the Abaqus/Explicit code will further develop the methodology."

Their research into the use of FEA for crack prediction is already having a positive impact on the R&D workflow at Tenaris. "We aim to reduce the number of long and expensive full-scale tests," says Cristea. "Simulation has enabled us to establish the relative weight of the different factors that affect the in-service behavior of a component and arrive faster at an interpretation of our laboratory results. And where we deal with repetitive cases that were already experimentally validated, FEA offers us a powerful alternative tool for results prediction."

For More Information

www.tenaris.com
www.3ds.com/SCN-October2012

Realistic Engine Simulation Down to the Last Detail

MAN Diesel & Turbo uses Abaqus FEA and XFEM for a weld redesign against fatigue cracks

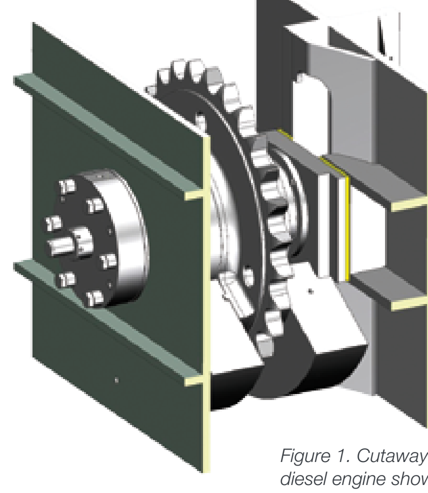


Figure 1. Cutaway model of a marine diesel engine showing dual flyweights.

A century ago this year, the first ocean-going diesel ship in the world, the M/S *Selandia*, embarked on her maiden voyage. She was a technological wonder, and both her hull and engines were built by Burmeister & Wain (B&W) of Copenhagen, Denmark.

B&W is now a part of MAN Diesel & Turbo—a company with 12,000 employees worldwide—and the marine low-speed business unit is located in Copenhagen. The unit has capitalized on B&W's historic expertise to produce new engines that can weigh up to 2800 metric tons and tower 16 meters high. Once such behemoth engines are installed, they have to be serviced in place—and they must require servicing as infrequently as possible.

Reliability and durability, over a long and demanding life, are crucial to marine engines. They are built to perform over 30 years, roughly 6,000 hours a year, at a constant speed of about 100 RPM—a billion revolution cycles on full design load. Under these grueling conditions, fuel combustion and inertia of moving components can potentially cause high-cycle fatigue failure. "It's vital that every part of our engines is designed and analyzed with sufficient safety margins against fatigue loads—right down to the welds," says Tore Lucht, industrial researcher at the R&D department of the marine low-speed business unit.

Simulation of cracks: A strong tool to design against weld fatigue

The criticality of paying full attention to every single weld detail in the huge complex engine structure became a focus for R&D when a butt weld on a low-speed marine diesel engine developed a crack. The weld was on the face of a second order compensator, a large rotating component that dampens engine vibration for greater crew comfort. While the compensator was designed to withstand large loads and stresses, the engine had only been in service two years, logging 13,000 hours and approximately 78 million revolutions of the crankshaft when the crack was observed.

The R&D department conducted a preliminary investigation using Abaqus finite element analysis (FEA). "We switched to Abaqus several years ago because we found it was the best tool for simulating engine structures," says Lucht. "We use it for all structural calculations of our low-speed engines." In the initial investigation they modeled a small cutout from the engine undergoing centrifugal force from the flyweights (see Figure 1). The centrifugal force was that of a MAN B&W 5S60MC-C diesel engine at 100 percent RPM—a 16-metric-ton load.

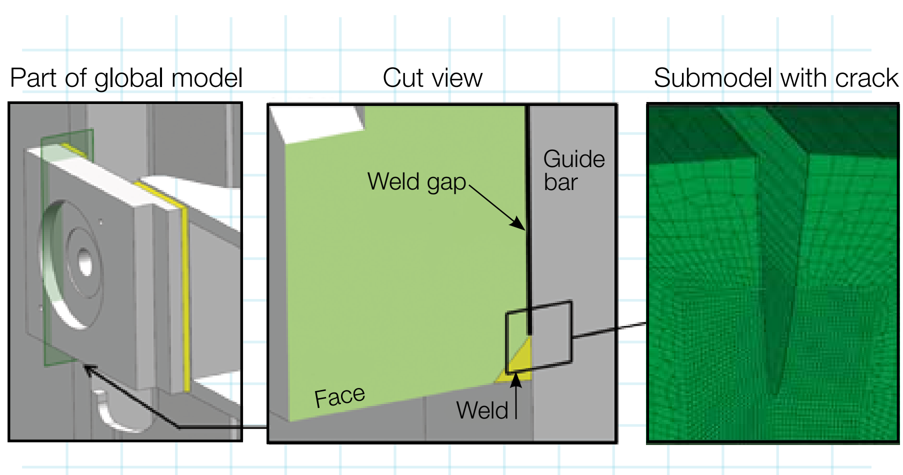


Figure 2. Left to right: A section of the global model of the compensator, a cut view of the weld gap, and an XFEM submodel of the crack in the weld.

The engineers were able to simulate all these loads accurately based on decades of field data from operating engines. "Our engine analyses are highly realistic," Lucht says, "because we put so much effort into benchmarking against literature, measurements, and service."

The initial simulation showed that small areas of the weld root had stress levels above a reasonable design limit. A different weld design might significantly reduce stress amplitude and solve the problem, but how could the engineers make sure that the improvement was sufficient? "Clearly more investigation would be required to determine a repair procedure and a suitable weld for future engines," Lucht says.

The International Institute of Welding (IIW) recommends three different methods to analyze welds: hot spot stress, effective notch stress, and fracture mechanics. The first method evaluates the weld by comparing structural stress at the hot spot to the fatigue (FAT) class of different weld details. Effective notch stress analysis can assess both toe and root stress level, again comparing each to a special FAT class. Fracture mechanics can be used to simulate typical weld defects on any part of the weld, using linear elastic fracture mechanics (LEFM).

Based on previous research projects and experience, the analysts at the R&D department decided that LEFM would give them the most accurate answers, even though it potentially involved a great deal of model preparation. "The applied meshing of the crack can be very time-consuming because a special mesh is required along the crack front," says Lucht. However, the team employed Abaqus' eXtended Finite Element Method (XFEM) technology, which reduces modeling time while offering an enriched environment for exploring fracture failure, even when the crack doesn't follow element boundaries. Lucht adds, "Since we already had advanced numerical models of our engines in Abaqus, that cut simulation time even further."

The engineers submodeled the area of the crack in XFEM with a fine mesh. "Investigating a crack of a few millimeters growing in a four-story engine undoubtedly requires some mesh refinement in the pertinent area," Lucht says. "This submodeling strategy made our task easier and less time-consuming."

Opting to model the most severe dynamic loading on the crack, the engineers excluded contact in the crack definition, leaving a weld gap—which could occur in such a structure due to either residual stresses from welding or from a slight misalignment between the parts that the weld joins.

After calculating the stresses on the original design, the engineers performed similar analyses on proposed welds for the engine repair and for weld designs on new engines going forward.

Residual stress and modeling the welding process

One final load check was needed to confirm that the repair was sufficiently strong: the residual stress state caused by the welding process itself. This stress was not included in the other LFEM calculations, and it could potentially alter the strength of the final weld.

To determine the residual stress state of the weld, the R&D department used a weld simulation tool for Abaqus developed in a previous research project with the Technical University of Denmark: a specialized modeling principle for simulation of a moving heat source by weld filler, body flux, and surface flux. The simulation adds the weld filler incrementally, with the elements representing the weld filler assigned a temperature above its melting point.

The elements were activated in groups with the model change command and with predefined values of temperature, body flux, and surface flux. New groups were automatically activated in subsequent steps as the old groups cooled as a function of the heat transfer. "Realistic simulation of the moving heat source of the weld is a key to this type of simulation," Lucht says. "By adjusting the active parameters like weld sequence, heat, and flux, it has been possible to obtain a high level of validation by comparison to experiments using methods like neutron diffraction measurements."

In this case, the simulation helped the engineers see the result of welding an additional supporting structure onto an engine. This predicted both the deformation of the existing engine structure and the size of the residual stress field around the crack front. The analysis assumed that filler material was welded in only three strings, and the welding was simulated along the side

A modern MAN B&W two-stroke diesel engine.

of the model, where it would introduce residual stresses perpendicular to the only critical weld defect that could cause an opening or closing of the crack.

These final simulation results enabled the engineers to relate both the residual welding stress fields and the crack simulation to obtain realistic fatigue assessments. As expected, large tensile residual stresses remained at the toes of the weld, but the stress level was close to zero at the weld's root. This meant that the stress intensity factor evaluation of the critical root of new weld designs would only be marginally influenced by the residual stresses of welding.

"Even so," Lucht points out, "It would be good practice to take precautions, such as adding a peening step to the weld toes, to limit the influence of high-tensile stresses on the safety margin against fatigue failure, so that we don't introduce a new problem with the repair."

Analysis results: clear sailing ahead

Realistic simulation revealed why the original weld design performed as it did and confirmed that the new design would be safely within the recommended limit curve. The demonstrated lack of tensile stress in the weld root proved out the integrity of the new design. "This combined method of weld process simulation and fracture mechanical evaluation of weld defects with XFEM is a strong tool to evaluate the structural integrity of complex welded structures," says Lucht. "From this analysis, we were able to verify that our new weld design is safe, both for repair of existing engines and for use in future engines."

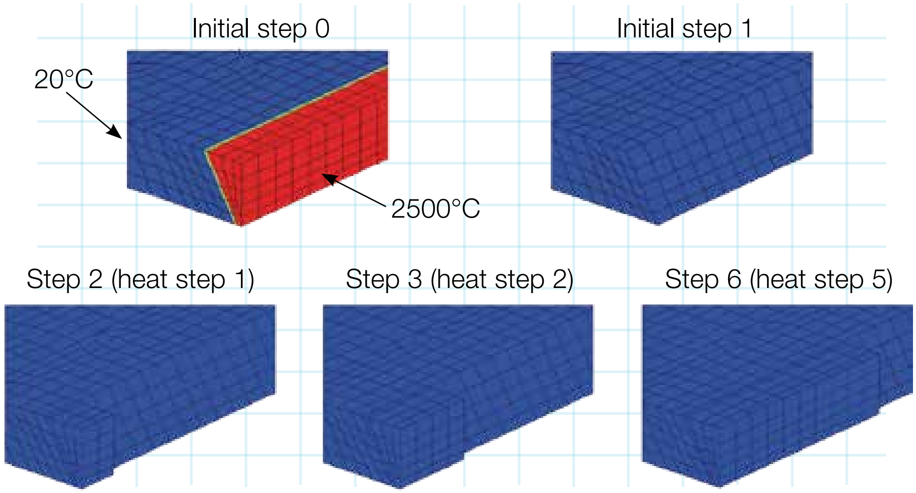
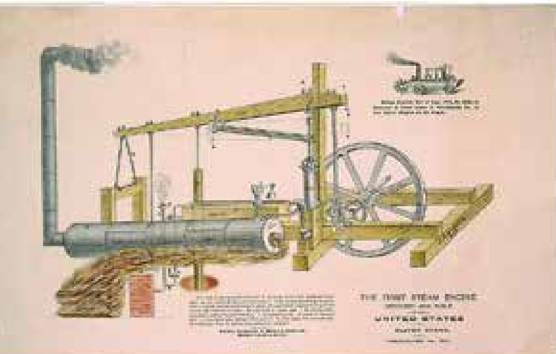
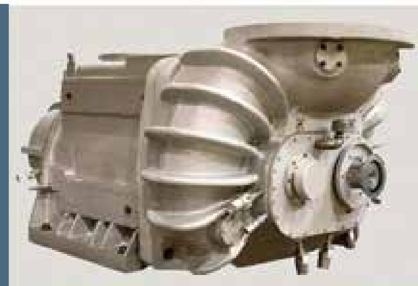
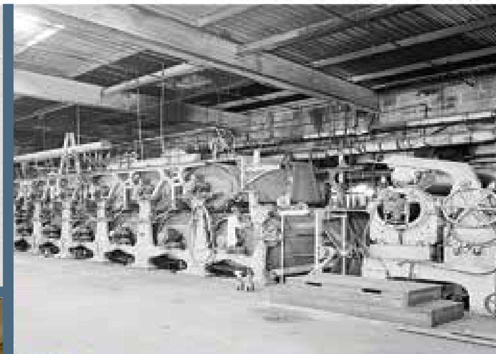
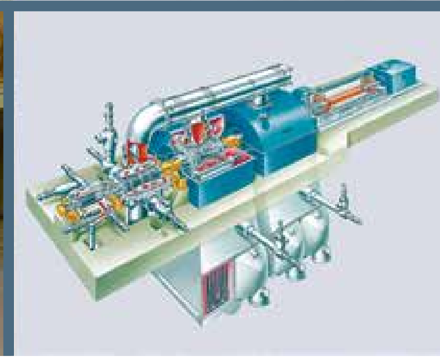


Figure 3. Step-by-step simulation in Abaqus of the welding process, adding weld filler under incremental passes of heat.

For More Information

www.mandieselturbo.com
www.3ds.com/SCN-October2012

The Quest for Innovation with Realistic Simulation

Jack Cofer, Sr. Technical Marketing Specialist, Industrial Equipment Industry, SIMULIA

Ever since a Stone Age man first picked up a rock to crack open a nut and later attached it to a wooden shaft to make an axe, we have been creating and using ever more complex tools to improve the quality of our lives. In the 21st century, this continuing quest for innovation is at the heart of the industrial equipment industry.

Because of increasingly competitive global markets, industrial equipment and manufacturing companies face challenges to reduce cost, shorten development and manufacturing cycles, and improve product life and performance. At the same time, they are subject to strict safety, reliability, environmental, and service requirements.

Similar challenges have been faced at manufacturing companies since the dawn of the first Industrial Revolution in England in the mid-1700s. While the machines back then were powered by water wheels and goods were transported to market in horse-drawn wagons and sailing ships, the manufacturers of textile looms, agricultural reapers and paper-making machines were fiercely competitive. Innovative ideas often created large financial rewards for inventors who made dramatic process improvements or got their machines to market first.

By the start of the second Industrial Revolution in the 1860s—with the introduction of the Bessemer steel-making process, mass production methods (known as the American System of Manufacturing), and the development of railroads and steam-powered ships for distribution—these challenges only became more complicated and the need for innovation accelerated. And today, as global competition combines with faster, larger and more precise and complex machines, 21st century challenges require a step-change adoption and implementation of new technology, methods, and management.

Defining industrial equipment

The industrial equipment industry covers a very wide range of machinery. To best serve this broad market, we at Dassault Systèmes (3DS), divide it into five major segments:

1. **Industrial Manufacturing Machinery** includes build-to-order machines that produce or assemble parts for many different industries, such as machining, packaging, welding, metalworking, textile and paper manufacturing, and even oil and gas exploration.
2. **Heavy Mobile Machinery and Equipment** includes machines that

move themselves, such as agricultural machinery, fork lift trucks, and construction and mining equipment.

3. **Installed Equipment** includes machines that carry or convey other things, such as cranes, conveyor systems, and elevators and escalators.
4. **Industrial Equipment Products** includes machines that are components or subsystems of larger machines, such as mechanical power equipment, heating and refrigeration equipment, furnaces and ovens, valves and fluid power equipment, and a variety of rubber and plastic products. With the exception of aircraft engines, all turbomachines are in this category, along with large diesel and gas engines for ship propulsion and power generation.
5. **Fabricated Metal Products** includes formed and stamped metal parts and just about everything needed to hold together the machines in the other four segments, such as bolts, nuts, screws, plates, springs, wire, and structural metalwork.

A hallmark of industrial equipment is complexity, both in the machines themselves and in the processes for designing them. Most of the machinery in the list above is composed of large assemblies with thousands of parts. They require complex multidisciplinary design processes that encompass such fields as fluid mechanics, aerodynamics, thermodynamics, mechanical stress and strain, materials and metallurgy, electromagnetics, contact/friction, fracture/failure, and fatigue life analysis. These machines are used in a huge variety of manufacturing processes and technologies, such as smelting, forging, casting, stamping, welding, cutting, rolling, molding, and extrusion.

Virtual testing and life prediction

SIMULIA's mission, as part of the Dassault Systèmes **3D**EXPERIENCE Platform, is to provide robust, high-performance, comprehensive multiphysics applications that enable our users to explore, understand, and improve the real-world behavior of their industrial products and manufacturing processes.

Our Abaqus product suite offers powerful technology for both routine and sophisticated engineering analysis problems covering a vast spectrum of industrial applications such as nonlinear stress, fluid-structure interaction, dynamic vibration, multibody systems, contact, impact, thermal coupling, and much more. The Abaqus solver technology is at the core of the next-generation simulation tools being integrated into Dassault Systèmes' **3D**EXPERIENCE Platform for virtual testing and life prediction of industrial equipment. The latest release of Abaqus incorporates recent advances in fracture and failure technology, leveraging the eXtended Finite Element Method (XFEM) to better predict fracture and failure in machines. We are also working with key software partners, including Safe Technology Ltd., to provide relevant solutions, such as their fe-safe add-on for Abaqus to accelerate fatigue life prediction.

Isight and the SIMULIA Execution Engine (SEE) are used to integrate multiple cross-disciplinary models and applications together in a simulation process flow, automate their execution across distributed compute resources, explore the resulting design space and identify the optimal design parameters (subject to required constraints). This toolset enables users to act upon the best options earlier in the design process to detect and fix problems before they can lead to costly scrap and rework.

Simulation Lifecycle Management (SLM) allows users in a small workgroup, or within an extended enterprise, to capture, share, and manage simulation IP for fast, collaborative decision-making. In this way, they can accelerate design decisions and deliver their products to market more quickly.

The integration of these applications is driving a paradigm shift in performing full verification and validation within a virtual design and test environment. By combining robust and accurate physics-based simulation, automated design exploration, physical test calibration, and simulation data management, we are providing a virtual test and life prediction environment that is enabling our users to significantly reduce physical prototype testing of their machines. This paradigm shift, initially established in the aerospace and automotive industries, is now at a level of maturity that the technology and methods are available to make this a practical approach for industrial equipment manufacturers. Virtual testing and life prediction provides huge savings in engineering and manufacturing time and cost, as well as dramatic improvements in machinery performance and reliability.

Applying technology to drive innovation

This issue of *SIMULIA Community News* provides a number of user case studies that detail the application of realistic simulation to reduce physical prototyping, improve reliability, and accelerate innovation. Another excellent industrial equipment example is described in a paper entitled "Thrust Collar Bearing Design Optimization using Isight" presented by ABB Turbo Systems at the 2012 SIMULIA Community Conference. It describes their use of Isight and Abaqus to optimize the geometry of a thrust collar bearing assembly for a large industrial turbocharger over a wide range of operating conditions. Isight delivered "very satisfying" designs meeting all of their stringent requirements, along with the additional benefit of "opening the mind for completely new design features and thus supporting the creativity of the engineers."

Key to improving our technology is close collaboration with our industrial equipment users. By engaging with us in one-on-one meetings, through our online community, and by attending our international and regional user meetings, you can share your requirements and influence our development priorities to further meet your industry needs. I look forward to working with you to better understand your simulation requirements for accelerating the design of innovative, reliable, and cost-effective industrial machines that enable your companies to be successful in the global marketplace.

About Jack Cofer

Jack is responsible for developing and directing SIMULIA strategy for the industrial equipment industry. He has over 35 years of experience in turbomachinery design and optimization achieved through various design and management roles at GE Power Generation, Demag Delaval Turbomachinery, and Engineous Software. He has a B.S. from the University of Virginia, an M.S. from the Massachusetts Institute of Technology, and an M.E. from Northeastern University.

For More Information
www.3ds.com/SCN-October2012

Mazda Balances Performance and Weight In a Steel Car Body

Isight streamlines and automates complex CAE optimization study

The Mazda CX-5 was the focus of a weight-reduction design optimization investigation.

Steel has remained the dominant material in car bodies for over a century by keeping pace with the evolving automobile. Improved corrosion-resistance, more refined mechanical properties, higher-strength characteristics, and advanced manufacturing technologies have kept steel at the top in terms of content in the average vehicle on the road—about 60% by weight today.

Yet weight remains a primary concern for automotive companies, due to its far-reaching effects on fuel efficiency. Other materials such as aluminum, magnesium, and composites are being increasingly considered as potential replacements for parts once made from steel. Although steel's proven reliability means it's likely to remain the primary ingredient in car bodies for some time to come, automobile manufacturers are now approaching the limits of how lightweight a steel car body can be. To fully understand and build within those limits, they are turning to sophisticated computer-aided engineering (CAE) tools that help them optimize their designs, provide the quality their customers demand, and meet ever more stringent mileage goals, emissions standards, and crash test regulations cost-effectively.

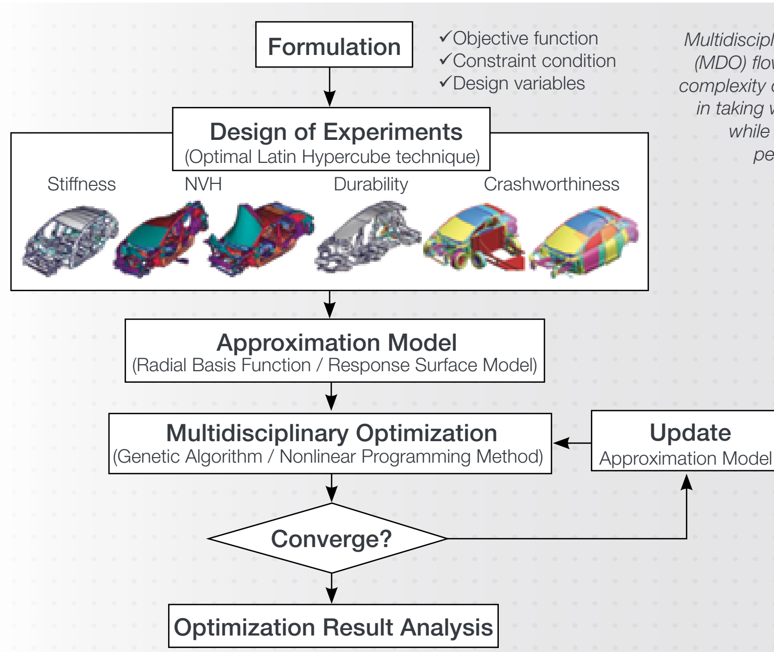
In a recent collaboration, the Vehicle Development Division and the Technical Research Center of Mazda Motor Corporation (Hiroshima City, Japan) developed a multidisciplinary methodology for design optimization (MDO) of a steel body structure based on the company's CX-5 car model. "Optimization technology is essential for solving the problem of how to balance improved performance against reduction in weight," says Takehisa Kohira, technical specialist at Mazda.

Optimization technology is essential for solving the problem of how to balance improved performance against reduction in weight.

Takehisa Kohira, Technical Specialist, Mazda

The team's goal was to identify the lightest gauge (sheet metal thickness) combination of steel parts that would allow them to reach four target performance values—stiffness, NVH quality, durability, and crashworthiness—for the top-hat structure of the CX-5. The group employed a variety of CAE tools to model such diversity of the whole-car-body behavior—including Abaqus, LS-DYNA, Nastran, and others. "CAE helps us improve our designs, increase the accuracy of our analyses, and build fewer prototypes," says Kohira. Abaqus finite element analysis (FEA) evaluated strength, durability of body components, and thermal stress of powertrain components in vehicle development.

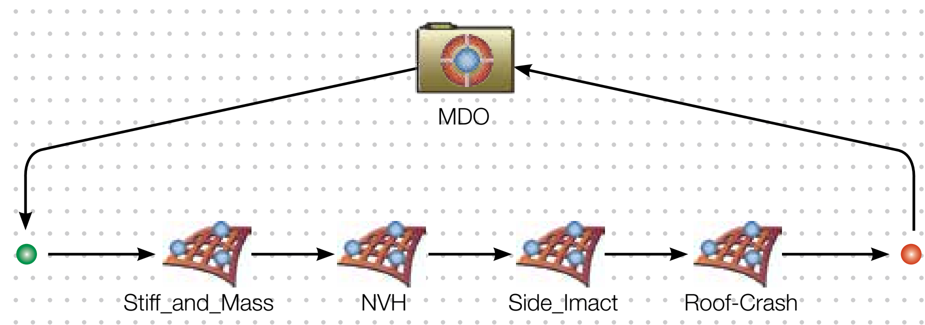
Creating an overarching analysis system that would first optimize vehicle body behavior in each of the four target areas, and then identify a final design that brought all these 'best' characteristics together at the lightest possible weight, was a complex analysis challenge.



Multidisciplinary Design Optimization (MDO) flow chart demonstrates the complexity of the challenges involved in taking weight out of a car design, while retaining a number of key performance characteristics.

Among the target behaviors being examined, stiffness—both static and dynamic—involved primarily linear calculations. NVH analysis, on the other hand, entailed complex multi-physics problems that considered both the physical interaction of frame components and whole-body vibration. Crashworthiness, which is nonlinear to different degrees depending on whether it is front, rear, or side impact, presented the most complexity. "Side-crash analysis concerns only the bending/buckling domain, which can be predicted with an approximation model," says Kohira. "However, front- and rear-crash involve strong nonlinearity due to both buckling and axial compression of a multitude of parts, so the optimization of weight versus performance was particularly complex for these analyses."

To bring all the data together into a 'best-performance' body structure design, the team employed a variety of Design of Experiments (DOE) techniques and approximation models, manually conducting tradeoffs between the different behaviors. The car body's various performance targets were considered as constraints, and the design variables were thickness of material of each body component. "Our end goal was always the minimization of body weight," says Kohira. "But working to achieve that goal through manual data organization and comparison was taking a great deal of time."



Isight process automation and design optimization software enabled Mazda to set up their MDO challenge within an automated workflow that reduced analysis setup and runtime considerably.

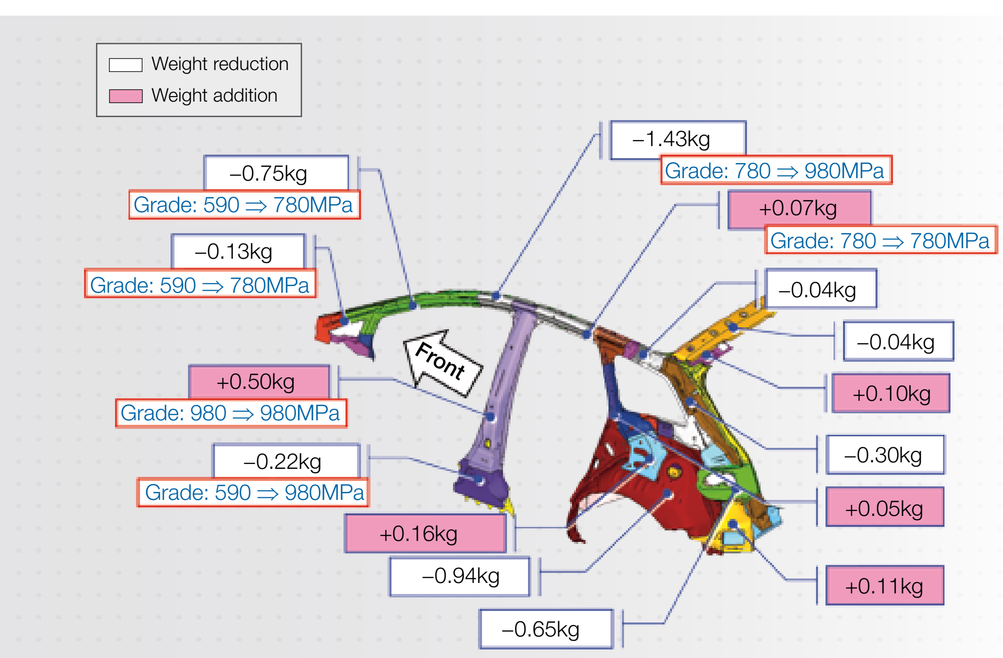
Then the team turned to Isight for process automation and design exploration. "Once we started using Isight, we could more readily understand the limitations of our designs after our DOE studies, which made it easier for us to make decisions," says Kohira. "We could see the design space more clearly and better visualize our results."

Isight helped the engineers integrate all their CAE software into customized, 'drag and drop' workflows that would run all their performance tradeoff sequences automatically. "By using Isight, we could confirm and numerically validate our designers' ideas and be confident in the validity of our designs," says Kohira.

An interesting effect of applying Isight to the car body optimization problem was the ability to categorize body components based on their importance relative to the performance of the whole structure. "During the optimization process, in most cases parts that had a low contribution to performance became thinner," says Kohira. "On the other hand, some of the parts that had a large contribution to performance needed to become thicker and heavier. Optimization with Isight enabled us to balance out these opposing needs while still lightening the overall weight."

The result? The team achieved their goal of a 3.4 percent reduction in weight over the previous design of the CX-5. Their multidisciplinary design optimization protocol is now being used in Mazda's SKYACTIV-BODY technology development program, which is aimed at improving vehicle fuel efficiency through engine and transmission development along with lightweight bodies and chassis.

Going forward, the Mazda team plans to adapt their Isight-automated MDO system to aluminum, CFRP (carbon fiber reinforced plastic), and other materials. "We have refined our steel designs about as far as they can go at this point," says Kohira. "Future designs will incorporate increasing proportions of a number of materials in addition to steel, but we now have the technology in hand to manage even greater complexity."



Left. Initial analysis image of the portion of Mazda CX-5 body side frame assembly showing weight reduction (white labels) and weight addition (pink labels) that resulted from optimizing performance over a number of behavior parameters.

For More Information
www.mazda.com

Caterpillar Rolls to Success with FEA

Abaqus FEA accurately models the behavior of mobile compactors and landfill waste

View of a Cat® landfill compactor, showing the patented drum tip technology that offers greater compaction and traction.

Humans have been throwing things away since the dawn of history. As civilizations rose, so did trash heaps. These days the managing of waste isn't just a matter of collecting refuse—it's a high-volume, sophisticated business concerned with disposing of all that garbage in the safest and most efficient manner possible.

At the top of the heap is the issue of waste compaction in landfills. "Compaction is one of the most important processes in landfill operations," says Greg Zhang, engineering specialist in the Industrial & Waste Group at Caterpillar Inc. "It promotes waste density, prolongs the useful life of the land, and increases site profitability."

Caterpillar produces several models of landfill compactor—immense, multi-ton machines that look like the hybrid offspring of a steamroller and a bulldozer. These behemoths have massive steel drums with patented tip technology that provides greater compaction and traction and promotes ground-surface integrity.

Compactors need to fit the conditions of the sites in which they work, since not all waste is alike. In mainland China, for instance, landfill waste contains significantly more food and organic matter and higher water content than that of industrialized regions in North America and Western Europe. "Variations in waste make for important differences in compressibility and shear strength," Zhang says. "Often, that means we alter the drum and other features to suit the terrain."

> Abaqus is a powerful tool for simulating machine-ground interactions.
>
> **Greg Zhang**, Engineering Specialist, Industrial & Waste Group, Caterpillar

To better tailor compactor designs to individual environments, Caterpillar has been using Abaqus finite element analysis (FEA).

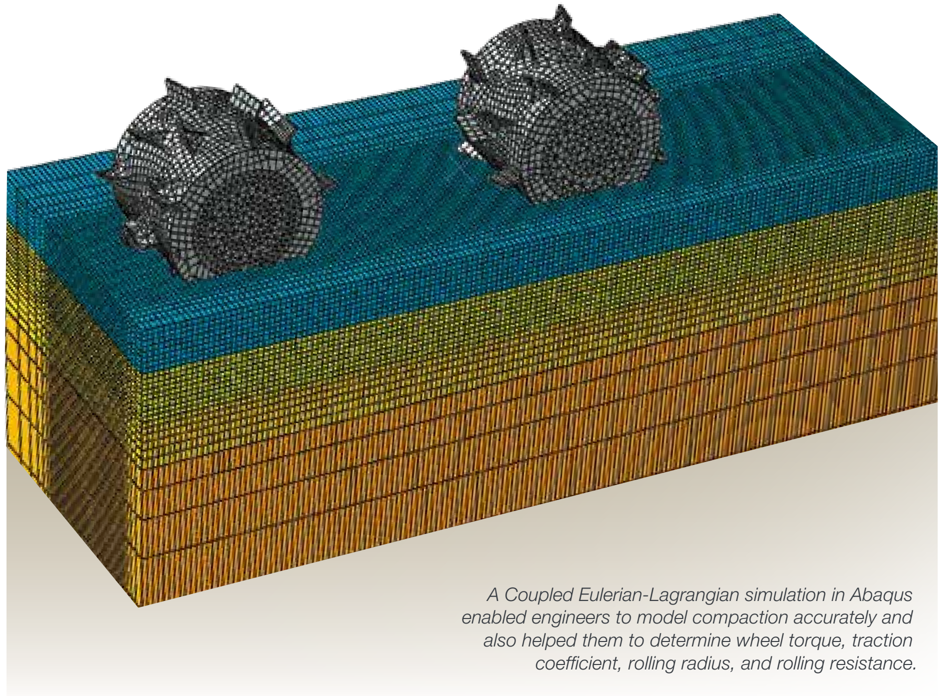
FEA hits the ground running

"Before Caterpillar started using FEA, their engineers basically created new model compactors by drawing on experience," says Liqun Chi, senior engineering specialist in the Product Development & Global Technology division of Caterpillar. "Within a certain range of parameters, that worked well, but outside of them—making bigger or smaller machines than the current product line—you could run into problems of scale." That could mean an expensive redesign after physical testing of the compactor, well into the product development cycle. "Analyzing designs at the concept stage instead made a lot of sense," Chi says.

When the engineers first began employing simulation, their Abaqus models were quite simple: a featureless drum deforming a level landscape. A modified crushable foam model stood in for the loose waste material since it included elasto-plastic attributes with volumetric hardening and could capture both compaction and spring-back. The waste was modeled in multiple layers. The compactor drum made several passes (forward and reverse) over the waste, and the resulting compaction was compared to actual field data. The analysts also simulated the drum running on a typical slope.

The simulation data for compaction correlated well with field data—but the engineers also wanted to capture other information. "The smooth-drum model gave a lower figure for wheel torque than physical testing," Chi says. In addition, the analysis didn't consider the effects of detailed wheel design (tip shapes, number, and arrangement).

To add these features, the engineering team created a drum model with tips for an Asian-market SEM6020 compactor and tried a different method of analysis: arbitrary Lagrangian-Eulerian (ALE), a technique used to control distortion.



A Coupled Eulerian-Lagrangian simulation in Abaqus enabled engineers to model compaction accurately and also helped them to determine wheel torque, traction coefficient, rolling radius, and rolling resistance.

In this type of analysis, however, deformation of mesh elements remained a challenge. "We could accurately capture the behavior of a first pass of the drum," Zhang says, "but the deformation interfered with simulating subsequent passes. And it only worked well with gentle slopes and relatively small wheel slips." The analysts decided to explore another method.

Their final choice was a Coupled Eulerian-Lagrangian (CEL) analysis. This time, the engineers created two different simulation phases for each pass of the drum. In the Lagrangian phase, the elements were temporarily fixed within the compacting material, capturing its deformation. In the Eulerian phase, deformation was suspended and significantly deformed elements were re-meshed automatically. "Now we could accurately model multiple passes of the drum," Chi says.

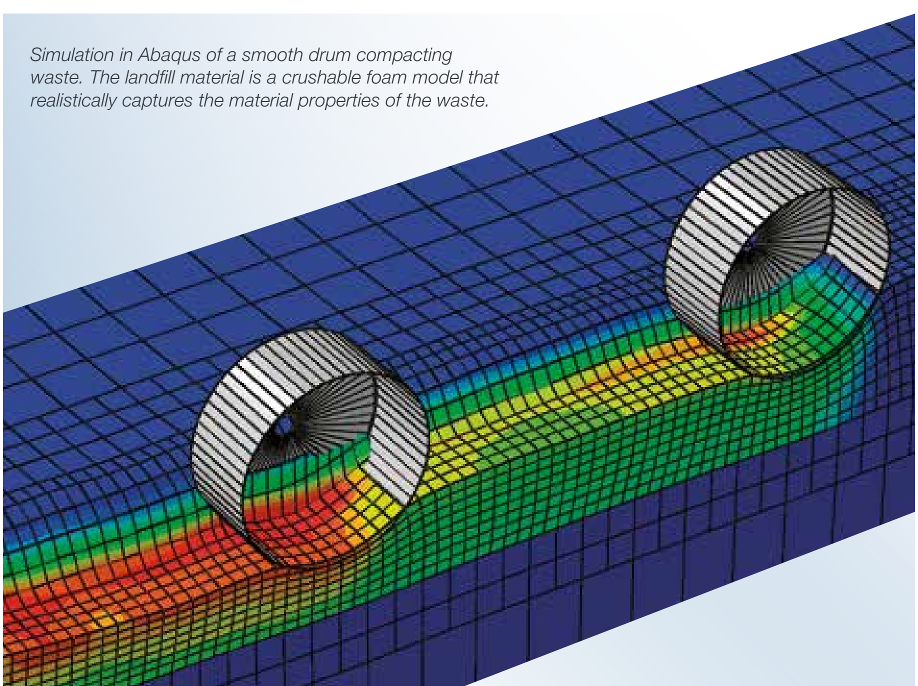
Rolling to results

The CEL method resolved the element distortion issues experienced with previous ALE compaction models. Now the team could simulate multiple passes for the compaction drums, even for drums with detailed tip shapes, accurately predicting the behavior of both drums and landfill waste compared to physical tests. "Abaqus is a powerful tool for simulating machine-ground interactions," Zhang says. "We were able to provide other product development teams—such as powertrain and wheel groups—with valuable torque, traction coefficient, rolling radius, and rolling resistance data."

The additional data ensures that every size compactor will perform as required, in whatever environment—improving metrics from productivity to fuel consumption.

"We now use CEL in Abaqus for all our multi-terrain loaders," Zhang says. "It helps us design the machinery as well as the drum tips." Analyses are currently being performed on a range of Chinese and U.S. landfill compactors, including the popular American model 836.

"We're also migrating the mesh from this model to the analysis of other rigs such as soil compactors, and even vibratory soil compactors, for construction," says Zhang. "Modeling these other types of equipment will require some finer mesh, but the analyses should provide us with performance benefits comparable to our landfill compactor simulations."



Simulation in Abaqus of a smooth drum compacting waste. The landfill material is a crushable foam model that realistically captures the material properties of the waste.

For More Information

www.caterpillar.com
www.3ds.com/SCN-October2012

SIMULIA V62013x Delivers Key Enhancements

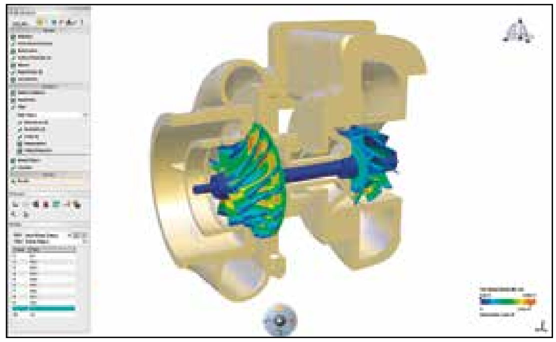
SIMULIA provides several new developments in the V6R2013x release including enhancements to the ExSight, DesignSight, and Simulation Lifecycle Management (SLM) product families as well as the CATIA V6 Analysis product portfolio.

ExSight is the expert-level product family for advanced nonlinear finite element analysis (FEA) and multiphysics simulation on the V6 platform. ExSight leverages Abaqus FEA technology and provides advanced capabilities for modeling, meshing, simulation and visualization of results for analysis experts.

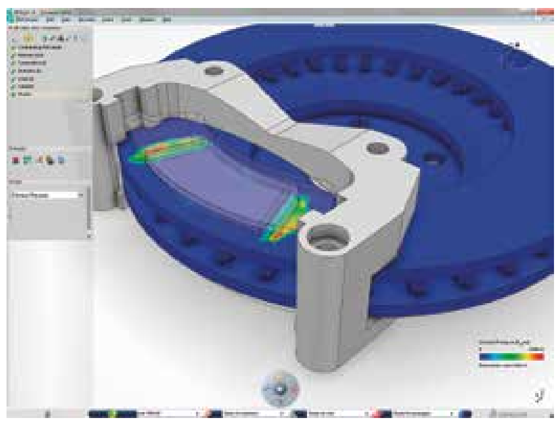
The V6R2013x release adds several new modeling and visualization capabilities to ExSight. Beam elements, which have various applications for aerospace and offshore structures, are now supported and allow users to define their own beam profiles. In addition, for cases where composites structures need to be modeled, continuum shells are available for better accuracy, and in the case of powertrain analyses, gaskets are also offered to the end-user. Additional procedures have been added in this release, including mode-based linear dynamics, usually for noise and vibration applications, and buckling, for analysis of unstable structures. The release includes breakthrough enhancements in results visualization such as visualization leveraging remote computing resources for higher performance, and the ability to display the surrounding assembly to a set of simulation results.

DesignSight enables designers in product manufacturing industries to evaluate realistic product performance during the design phase. V6R2013x provides a number of functional enhancements for defining new types of loads, restraints, and interactions.

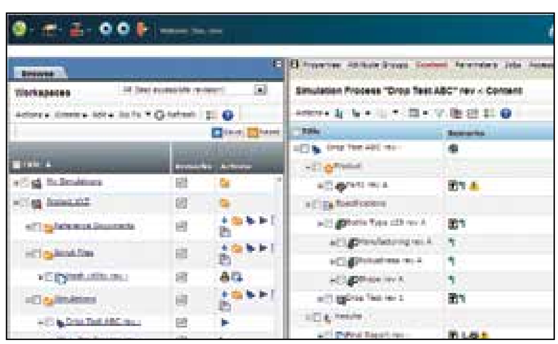
The latest release now offers realistic thermal and structural contact interactions with an easy-to-use user interface, enabling users to understand how parts influence each other within an assembly, greatly expanding the scope of DesignSight's applications. In addition to these interaction enhancements, the release provides several new loads and restraints, such as centrifugal force, convection loading, and restraints around and along any XYZ axis. These features extend the toolset available for users and broaden the scope of what can be modeled.



With ExSight, material rendering can be applied on select components during results visualization to obtain results images with lifelike representation.



DesignSight can model contact between components in an assembly to simulate accurately how loads are transferred between parts.



Scenario Definition (SCD) is part of the SLM product family and allows users to reference their existing geometry and requirements data within a simulation and the simulation results, ensuring a seamless flow of data throughout the product design and validation process.

Simulation Lifecycle Management (SLM) enables users to manage simulation data, capture and deploy company best practices, distribute simulations across computing resources, and share the simulation results to improve collaborative decision making.

Existing SLM customers will benefit from additional enhancements in V6R2013x to streamline the product development process thanks to further integration of simulation with the rest of the enterprise solution. To provide better support of ENOVIA Engineering, Designer and Requirement Central data allows for an even more seamless product development process. For example, values defined in Requirement Central can directly be used in simulation workflows. Dedicated search tools for retrieving simulation data based on simulation-related classification are now exposed to all V6 users (not just SLM users), ensuring any user can find the simulation data they need. SLM also continues to provide new functionality for SLM advanced users, including the support of array (or multi-valued) attributes, improved execution communication, and flexible simulation job log configuration for administrators.

For More Information
www.3ds.com/simulia

Simulating Pipeline Rupture with Finite Element Analysis

Abaqus FEA predicts the initiation and evolution of damage in metals, providing an alternative to laboratory structural testing

Although metal pipelines are manufactured to be durable, they are still susceptible to damage. A common example is a gouge from a backhoe bucket or other heavy equipment. The resulting deformation needs to be comprehensively evaluated to determine whether the pipeline is still functional, or requires repair or replacement.

A body of assessment guidelines for determining the fitness-for-purpose of a damaged pipeline has been developed. Many of these methods rely on experimental results and semi-empirical procedures; as such, their validity may be limited when considering loadings, materials, or specific damage configurations that are outside the scope of their assumptions.

Computer modeling of pipeline damage with finite element analysis (FEA) can complement the existing methodologies by predicting the effects of such outside variables. Simulation can provide important information about the performance of different materials and pipe geometries being considered.

To study how FEA could predict damage initiation in an internally pressurized pipe of API X65 steel with a gouge defect, a team from SIMULIA reproduced an experiment published by Oh in the International Journal of Pressure Vessels and Piping in 2007 using Abaqus FEA software, which provides two types of damage initiation criterion: ductile, based on the nucleation, growth, and coalescence of voids; and shear, based on shear band localization. The team focused on the use of the ductile criterion.

The geometry of the model under consideration is shown in Figure 1. A simulated gouge, 100 mm long, was introduced into the pipeline.

A quarter-symmetric mesh of second order hex elements was generated, and internal pressure loading was applied. End forces were applied to simulate experimental closed-end conditions, and the loads were increased linearly with time. In general, the specification of damage initiation is included in the material definition and must be used in conjunction with a plasticity model. In this analysis the Mises plasticity formulation was used. The mesh is shown in Figure 2.

The ductile damage initiation criterion is a phenomenological model. This criterion was included in the analysis by specifying the equivalent plastic strain at damage initiation as a function of stress triaxiality and strain rate. Stress triaxiality is known to play a role in damage growth.

To calibrate the FEA models, test data from the Oh study was compared to a corresponding FEA model. The test specimens were round, notched bars, loaded in tension until complete fracture was achieved. Each test was compared against a corresponding FEA model. It was noted that as notch radius decreased, yield and tensile strengths increased, but strain to failure decreased. This behavior is consistent with the increasing triaxiality of the stress state as the notch radius decreased.

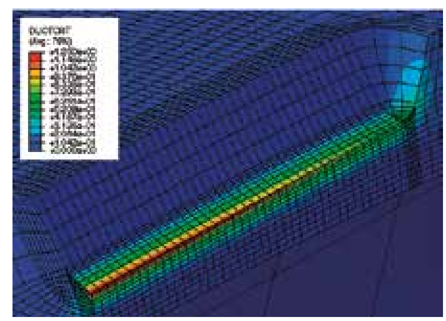


Figure 3. Abaqus FEA simulation of damage initiation in a pipe gouge.

A FEA contour plot of the damage initiation output variable is shown in Figure 3. Damage has initiated when this variable is greater than 1.0. From the contour, it can be seen that the critical element in the mesh is in the root of the notch at the intersection of the symmetry planes. Further x-y plotting of the initiation criterion in the critical element provided a more precise determination of the failure pressure: The simulation demonstrated that the threshold of 1.0 was crossed at a pressure of 24.97 MPa. The experimentally determined burst pressure was 24.68 MPa.

SIMULIA's FEA results compare favorably with the full-scale experimental burst test data collected by Oh. While laboratory results remain valuable for validating computer models, simulation can provide a lower-cost alternative to extensive structural testing.

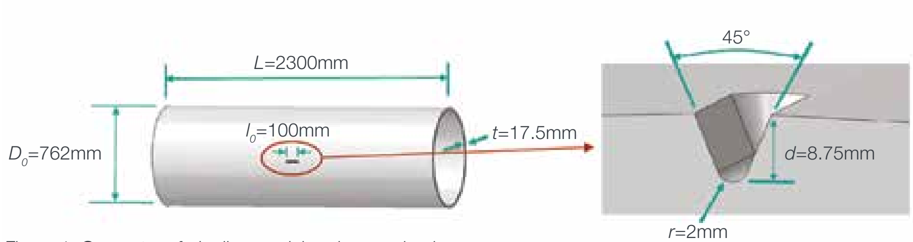


Figure 1. Geometry of pipeline model and gouge in pipe.

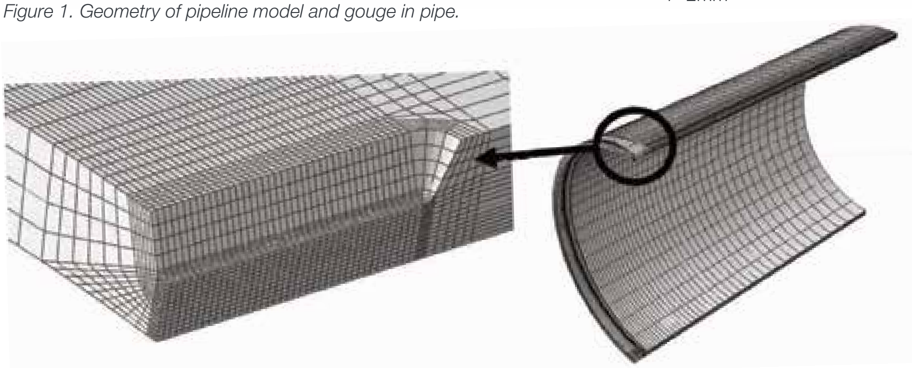


Figure 2. Quarter symmetric mesh of the pipe, with close-up of detailed mesh at the gouge.

For More Information
www.3ds.com/SCN-October2012

There's a New Twist in Wind Blades

NSE Composites used Abaqus FEA to validate a Sandia-funded sweep-twist design that captures 12 percent more energy

Figure 1. Sweep-twist adaptive rotor (STAR) prototypes were installed on a Zond 750 kW turbine at the TerraGen wind site in Tehachapi, CA. Note the aft sweep of the tip and the trailing edge curvature of this innovative blade design.

The basic physics and economics of wind turbine blades are relatively simple. For one, their power output is roughly proportional to the square of blade length. This relationship pushes designers to create increasingly longer blades for harvesting additional kilowatts. Secondly, as blades get longer, weight increases—by approximately the cube of the length—leading to higher raw material costs. This correlation sends designers in search of weight-efficient geometries that are strong and rigid enough to weather the increased loading inherent in longer blades.

Navigating a maze of engineering challenges such as these can lead to interesting design directions. At the United States Department of Energy's (DOE) Wind Energy Research Program at Sandia National Laboratories, the result has been the development of a sweep-twist adaptive rotor (STAR). This innovative curved blade was proposed in earlier theoretical research and had been garnering increasing interest for use in utility-scale applications. The new configuration is seen as a way to reduce operating loads on ever-lengthening blades. If successfully commercialized, the outcome would be larger, lighter, less-expensive, and more productive wind turbines.

> Using bladeMesher and Abaqus we can look at 20 different sweep geometries in a couple of weeks. If you were trying to evaluate all of those on a test rig, you'd be talking about a couple of years.
>
> **DM Hoyt**, Principal Engineer, NSE Composites

In 2004, Knight and Carver (K&C) Wind Group, a San Diego-based wind blade manufacturer, was awarded a DOE contract to develop STAR. Partnering with Sandia, K&C was responsible for design, fabrication, testing, and evaluation of a sweep-twist prototype. They began by assembling a team of specialized companies and academic institutions, one of which was Seattle-based NSE Composites, who were brought on board to perform the finite element modeling (FEM) of the new design.

"NSE had done a lot of analyses over the years on composite aircraft and helicopter aerostructures for companies such as Boeing," says DM Hoyt, one of NSE's founders. "Plus, we were already troubleshooting another blade problem for K&C and wanted to diversify our customer base to include more renewable energy, so the fit was a good one."

Hoyt and his partners at NSE have been using Abaqus as their finite element analysis (FEA) tool for years. As their projects moved toward larger and more complex models, the software's ongoing developments in simulating composites, crack generation, and fracture kept pace.

"Simulation has been a great asset for both our aerospace and wind energy work," says Hoyt. "It enables us to explore new ideas and look at the performance of multiple designs and materials, while minimizing expensive testing."

Sweep-twist blade basics

Rather than a traditional linear profile, a sweep-twist blade has a distinctive gently curving tip (or "sweep") with curvature towards the trailing edge (see Figure 1). Theoretically, this planform shape allows the blade to respond to turbulent wind gusts through a process of controlled twisting and bending: As the blade twists, it sheds loads that would normally be translated as material stresses to the root (or base) of the structure. In nature, a similar sweep can be seen in the wing shape of birds that migrate long distances and the characteristic profile of whale tails and dorsal fins.

The engineering upside of twist-coupling is the ability to create longer wind blades, while avoiding the higher loads that typically accompany increased length. Reducing loading—not only on the blade root but also on the turbine itself—enables a lighter blade design with lower raw material costs and helps lessen fatigue stresses on the rotating machinery. In early calculations, the STAR design promised a decrease in fatigue loads of 20 percent using a tip twist of three degrees. But as the design progressed, longer blades that capture more energy with no increase in load were pursued.

Beyond the potential advantages of altering the traditional length-weight-cost relationship, twist coupling is seen as a financially attractive solution for tapping low-wind-speed sites (defined as having an average velocity of 5.8 meters per second at a 10-meter height). These sites—in contrast to the high-wind-speed locations that have been the focus of wind-mining to date—are abundant in the U.S.'s mid-section and closer to major power-load centers. If the cost benefit proves favorable, development of low-wind locations could increase potential domestic wind farm area by a factor of twenty.

Understanding turbine behavior—without the wind

"Over the years wind blades have become more and more high tech. The industry is pushing the limits of design and materials," says Hoyt. "As that happens, engineers need to tighten up the loose legacy tolerances and manufacturing controls that originated in boat-building technology and adopt the more rigorous analyses that we have always done for complex aerospace structures."

Of particular use in wind blade analyses with FEA, notes Hoyt, is Abaqus' ability to handle composite properties and control material orientation. It can calculate blade-tip deflection (to avoid "tower strike") and accurately predict both torsional response (including twist angle, which is key to load-shedding) and shear-compression buckling stability (associated with sweep-twist) of composite sandwich structures. An additional capability key to wind blade analysis is the extraction of accurate equivalent beam properties directly from a solid 3D FEM. These bending and twisting definitions are used in wind-blade-specific dynamics codes to predict the overall performance of the turbine.

"During the preliminary design phase, the type and amount of input data is often limited," says Hoyt. "In the wind projects we've been involved with to date, there hasn't been a high-fidelity CAD model available to use as a basis for the FEM." So at the start of the STAR analysis, the NSE team only had the blade's basic geometric parameters—the planform shape, the airfoils, and the chord lengths—to work with. The desire for high-fidelity FEA at a design stage when only the basic parameters of the blade have been defined led to the development of NSE's bladeMesher software, which is able to create a solid 3D mesh of the blade from the partial data.

"Our software splines the geometry defined at several locations on the outer mold layer

Continued

A Snapshot of STAR's Composite Makeup

At the STAR project's outset, total rotor diameter for the prototypes was set at 56 meters (approximately 184 feet). Like most commercial blades, the STARs were to be fabricated using fiberglass and epoxy. Composite design included a stressed-shell approach in which the top and bottom shells are connected by a single shear web, rather than the more typical double web construction (see Figure 2). As the design went through numerous iterations over the course of the project, specific composite materials were carefully chosen for each blade feature.

In the sweep-twist blade design version that NSE analyzed with Abaqus FEA, unidirectional roving was called out for the blade's spar caps, and double-biased fabric covering a balsa core was the choice for the shear webs and shell panels. For the skin, a mix of dual-biased and spanwise-oriented (along the length of the blade) unidirectional fabric was selected during preliminary design tests for its ability to follow the curvature of the blade during fabrication (see Figure 3). This combination increased the overall stiffness-to-weight ratio of the blade while improving torsional response—a key factor in the twisting and load-shedding capability of the design. The maximum curvature at which fabrication and layup would become problematic from a material standpoint was also investigated.

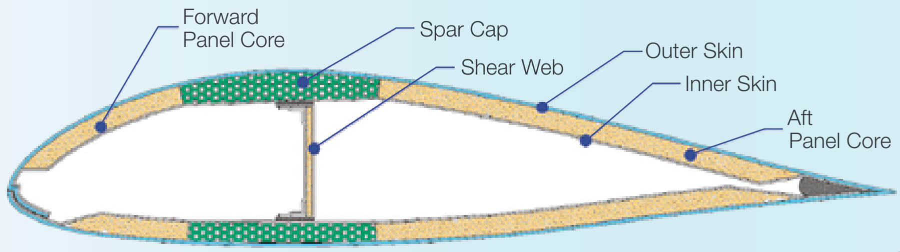


Figure 2, top. Cross-section of the prototype sweep-twist blade airfoil showing major composite construction features.

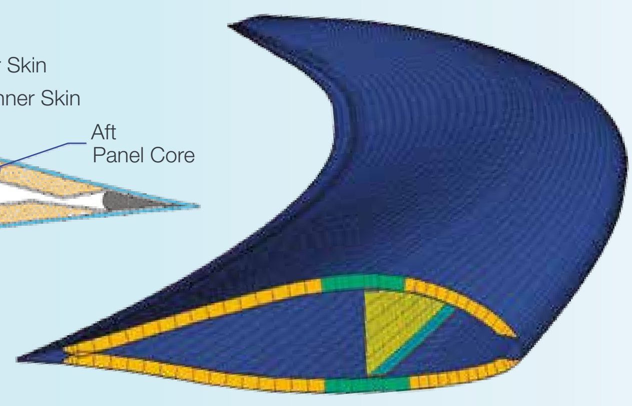


Figure 3, right. This Abaqus finite element model view illustrates how the unidirectional composite fabrics follow the curvature of the blade.

(OML) of the blade and combines it with the composite material thicknesses specified at each location to generate a mesh with the true thickness details," says Hoyt. "This solid mesh and material definition is then imported into Abaqus where we perform a detailed FEA. We have found that a solid FEM has many advantages over shell element FEMs, which have traditionally been used for blade analysis. These benefits include a more accurate prediction of twisting behavior and the ability to analyze stresses in the adhesive joints between structural elements."

As the design of the blade progressed, the team explored new airfoils and made adjustments to the sweep geometry to hone in on the optimal amount of twisting. The bladeMesher software enabled rapid updates to the solid FEM based on the new geometry, allowing the team to quickly assess the effect of each change. Abaqus' task was to confirm the earlier section analysis predictions, which were performed using constant-section-equivalents to estimate the effective beam properties of the blade.

To determine whether the sweep-twist geometry would shed loads as predicted, two wind scenarios were applied to the model: an operating load and an extreme-wind conditions case (50-year gusts at 156 miles per hour). The analysis was used to predict the blade deflection and twist, perform detailed stress calculations, and investigate potential shear buckling due to the increased twist inherent in the design (see Figure 4).

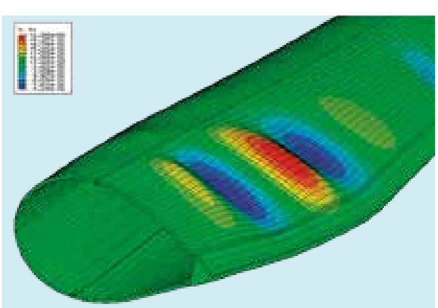


Figure 4. Post-processing of the Abaqus simulation allows users to see and understand potential buckling near the blade root.

For normal loads, there was excellent agreement with the tip deflection results from the section analysis. For extreme wind cases, strain value comparisons were also good. In further detailed studies, the engineering team found that their design met critical buckling limits at more than five times those of extreme wind conditions—a large margin of safety driven by the fact that stiffness to limit deflection, rather than ultimate material strength, was the key structural criteria (see Figure 5).

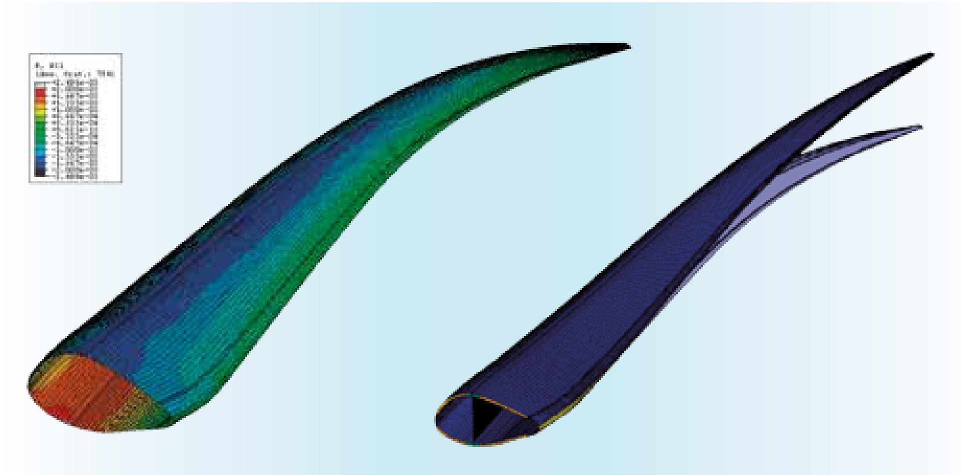


Figure 5. Abaqus FEA simulations illustrate stress/strain results for the STAR blade prototypes on the left (note the compression stresses on the upper, or low-pressure, side of the airfoil as the blade bends) and overall deflection results on the right.

Static physical testing of prototypes—for shear strain, blade deflection, and twist angle—followed FEA. The static test verified twist response under operating load conditions. Using fatigue testing, the K&C team was able to validate a 20-year lifespan for the new design, a timeframe equivalent to the current industry standard.

The prototypes were also field-tested, generating extensive data (as well as power) for several months in Tehachapi, CA—site of the TerraGen commercial wind facility and also one of the largest wind generation areas in the country.

A simple twist with major industry implications

With simulation and testing complete, the sweep-twist design's promised load-shedding response resulted in what Hoyt characterizes as a "very impressive" 12 percent power output boost over similarly-rated turbines now in operation—without load increase.

"I think it's the future of blade design," says Hoyt. "People are actively pursuing it." GE and Siemens, two of the biggest players in the industry, are currently developing swept blades. A start-up, Zimitar—founded by researcher Mike Zuteck, who first conceived the sweep-twist design—has just won a $4 million DOE contract to pursue the technology for bigger offshore turbine installations. Questions are in the air: Will it scale to a 50- or 60-meter blade? How will even longer blades respond?

At the 5th Sandia Blade Workshop (in spring 2012), engineers discussed possible downsides. Will torsional loading due to sweep introduce other problems like new modes of aeroelastic flutter? Or will it produce too much fatigue stress in the adhesive joints? There may also be issues on the manufacturing and transport side stemming from the harder-to-handle curved shape, says Hoyt. "But I wouldn't be surprised if in ten years a lot of production wind blades had some sweep in them.

"We use simulation all day, every day in our company. We can explore design space variables essentially with the click of a button and avoid a lot of early physical testing," Hoyt continues. "For example, using bladeMesher and Abaqus, we can look at 20 different sweep geometries in a couple of weeks. If you were trying to evaluate all of those on a test rig, you'd be talking about a couple of years."

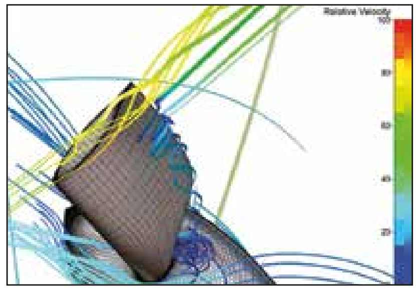
With all of the demonstrated benefits—savings in design time, reduced testing, and increased power output—this new twist on a traditional design is lining up to be a big win for wind.

For More Information
www.nsecomposites.com

Fluid-Structure Interaction for Flexible Wind Turbine Blades

The aero-elastic analysis is a critical step in the design process of wind turbines towards understanding of the dynamic response of the blades in terms of maximum deformation and stress. This analysis is not restricted to safety requirements but can also be used to develop passive control systems, which would take advantage of the fluid-structure interaction with composite materials. Indeed, wind turbine blades bend and twist during operation, effectively altering the angle of attack, which in turn affects loads and energy production. Power control objectives in terms of aerodynamic torque versus wind speed can be achieved by using aero-elastic tailoring. Especially for wind turbines deployed in remote locations, passive control could further reduce the risk of failure and thus enhance robustness and safety.

Since the blades are made of composite materials, the high loads can be reduced by selecting appropriate orientations for the fibers. To accurately predict the passive attenuation of the loads due to fiber orientation, complete aero-elastic simulations have been carried out on several configurations, which differ with one another in composite lay-up and overall sweeping of the blade geometry.



The prediction of aero-elastic effects requires devoted numerical tools to handle the complexity of the phenomena. The unstructured finite-volume flow solver FINE™/Open of NUMECA International can be coupled to any code integrating the coupling algorithm MpCCI developed by Fraunhofer Institute. The structural solver Abaqus has been chosen for this project given its wide material modeling capability and its extensive range of material models. The flow solver computes the three-dimensional flow around the blades and transfers the computed loads to the mechanical solver at each time step. The displacements are therefore computed and transferred back to the fluid solver, which deforms the mesh. The transfer of the solution between the fluid and solid meshes is ensured by highly accurate geometrical algorithms. The developed framework can complement existing methods in the design process of wind turbines by providing accurate prediction of aero-elastic effects on complex industrial configurations.

For More Information
www.numeca.com

PolyUMod™ Library for Advanced Modeling of Polymers

Realistic simulation of polymers and biomaterials can be challenging because the material response is characterized by significant nonlinear effects that cannot be captured by simple constitutive laws. Equally important but often forgotten is the material model calibration process. It can be a daunting task to manually select the material parameters for a complex constitutive model in order to achieve good agreement with experimental data.

Veryst Engineering provides solutions to both needs with our PolyUMod user material library and MCalibration software. PolyUMod provides Abaqus UMAT/VUMAT subroutines for elastomers, foams, thermoplastics, and thermosets that accounts for complex nonlinear effects including creep, relaxation, anisotropy, temperature and rate-dependence. MCalibration enables automatic calibration of these and all built-in material models from experimental data using advanced optimization algorithms.

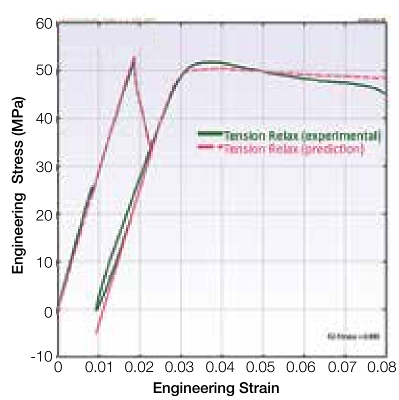


Figure 1. Calibrated PNM model in MCalibration

Here we show an example where we calibrate a material model for Poly-L-lactic Acid (PLLA), a biodegradable material with numerous applications in products such as absorbable medical implants. We obtained experimental data for this material using an unconventional loading/unloading time history that captures its nonlinear viscoplastic nature. Based on this, the Parallel Network Model (PNM) from PolyUMod is chosen to model PLLA.

The rheological representation of the PNM model we selected for PLLA consists of two networks: network A is characterized by Arruda-Boyce hyperelasticity and network B has, in addition to hyperelasticity, a viscoplastic part governed by power law flow rule and double exponential yield evolution. MCalibration is used to calibrate this PNM model to the PLLA experimental data. Figure 1 shows a MCalibration snapshot that has the calibrated model's predicted behavior and parameters. The predicted behavior matches experimental data closely. MCalibration outputs the Abaqus UMAT/ VUMAT definition corresponding to the PNM model, which is used in simulation of this PLLA material.

For More Information
www.veryst.com

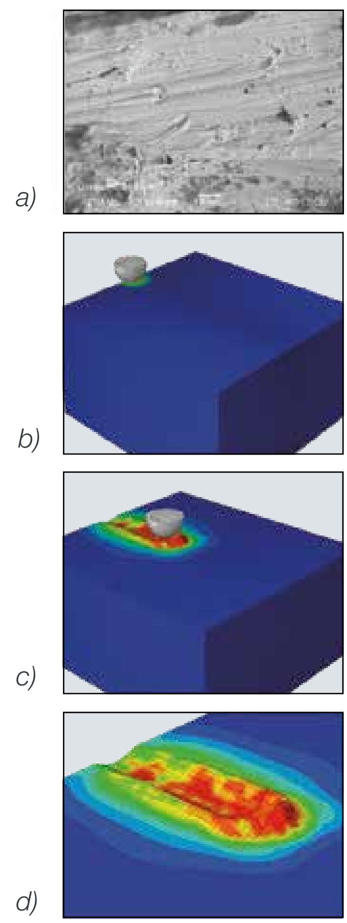
University of North Carolina at Charlotte Simulates Abrasive Wear of Tillage Tools

High energy requirements and rapid tool wear are two major issues of concern to farmers during soil-plowing operations. While the first issue has been more or less addressed by the advent of high-powered tractors, the second issue, namely, tillage tool wear, has received very little attention. Experimental work conducted at the National University of Colombia at Medellin shows that the tillage tools wear out rapidly with a wear-rate of almost 10 grams per kilometer of plowing. Such rapid rates limit the operating lifetime of these tools and result in significant downtimes for changing tools and therefore increase in tillage costs.

The experimental work being carried out at NUC-Medellin consists of three plowing assemblies, carrying one tillage tool each, attached to a tractor used for soil plowing. The speed of tractor is in the range of 3-5 km/h. A single tillage tool is 12 cm long and 6 cm wide and made of a heat-treated steel. The soil type used for experimental studies is a sandy soil in which the main particles are the sand particles that typically range from 0.5 mm to 2 mm in size. The observations from the experimental studies identify that the tool failure is primarily due to the sliding sand particles moving on the tool surfaces, although some wear due to gravel (particles that are of size greater than 2 mm) has also been observed. The worn-out tillage tools from the field tests indicate groove formation on the tool surface, confirming the material loss to be the material displaced from the groove.

Guided by the experimental work, research at the University of North Carolina at Charlotte has been focused on numerical simulations using Abaqus/Explicit to understand the effect various tillage parameters, such as the tillage speed, plowing depth, and sand properties, have on tool wear. As a first step towards understanding the wear process, a single sand particle sliding over the tool surface with a prescribed initial penetration depth is considered. The tool is modeled as an elastoplastic material and the sand particle is treated as a rigid cylinder with a hemispherical tip. The sliding process is carried out in three steps: an initial step to establish contact with the tool surface and initiate an indentation process, a second step to move the sand particle over the tool surface, and a third step to disengage the particle from the tool surface. Various parameters such as the particle speed, initial penetration depth, and particle size are varied over a select range of values (consistent with the experimental values). The motion of the sand particle over the tool surface results in a groove due to the plastic deformation of the tool. The groove geometry along with the classical ploughing theory is used to calculate the material removal rate due to the scratching of the tool surface by a sand particle. This is then used to obtain an average material removal rate due to multiple particles scratching a tool surface.

The numerical predictions for the material removal rates with sand particle sizes from 0.5 mm to 1 mm have been found to be in agreement with the experimental findings. Our results indicate that the finite element model, developed with Abaqus/Explicit, provides a robust tool for understanding the factors that are important in controlling tillage-tool wear during plowing operations.

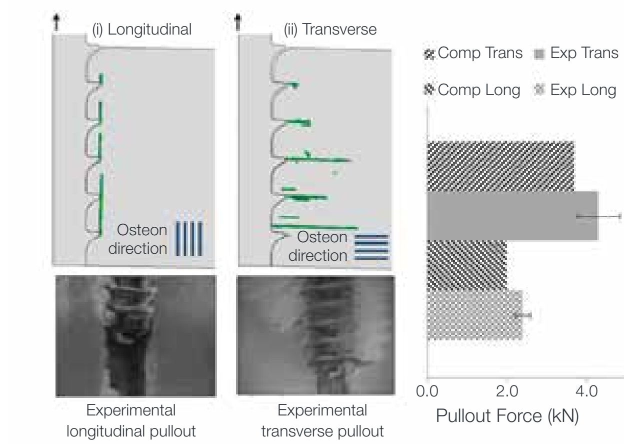


Tool wear during tillage operations: (a) The surface of a worn-out tool, (b) Simulated indentation on the tool surface, (c) Tillage tool deformation, (d) Tool surface deformation.

For More Information

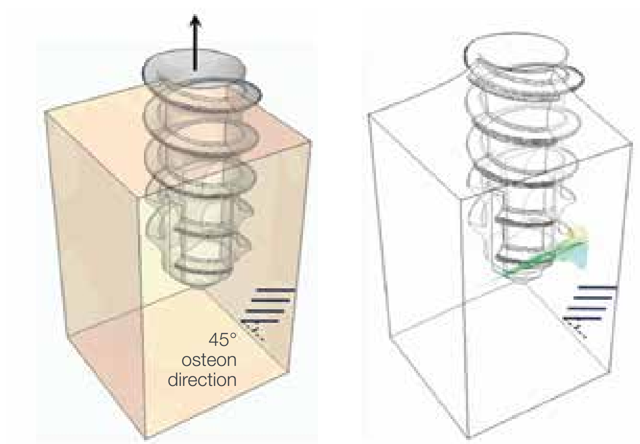
Gaurav Goel
ggoel1@uncc.edu
www.mees.uncc.edu
www.3ds.com/SCN-October2012

National University of Ireland Galway Models Cortical Bone Fracture Using the eXtended Finite Element Method



Feerick and McGarry 2012:doi:10.1016/j.jbiomech.2012.03.023

2D screw pullout model for longitudinal and traverse osteon orientation



3D screw pullout application for 45° osteon orientation

Eighty percent of a human's skeletal weight comes from cortical bone—a compact type of bone tissue mostly present in the shaft of long bones found in the arms and legs and on the outer shells of smaller bones such as vertebral bodies. As a naturally engineered strong composite material, cortical bones provides the strength individual bones need to support body weight, facilitate movements, and protect internal organs. In many cases, when orthopaedic trauma devices that use bone screws as their primary mode of fixation are implanted, cortical bone is the main supporting structure that prevents the screw from pulling out of the bone.

To evaluate device design, researchers at the National University of Ireland Galway are using realistic simulation with advanced modeling techniques for cortical bone fracture caused by pull-out forces applied on the screws. The eXtended Finite Element Method (XFEM) and the UDMGINI user subroutine were implemented to model the fracture of cortical bone. As the microstructure of cortical bone is analogous to a composite, consisting of a fiber and a matrix, Hashin damage-based criteria were developed to simulate the fracture patterns of cortical bone. In addition to defining the initiation criteria, the direction of crack growth was also defined for each criterion. These were based upon experimental observations of cortical bone fracture.

Cortical bone was modeled using anisotropic elasticity. Axial failure strengths were defined for tensile and compressive loading as well as shear failure strengths for each of the criteria developed. A material orientation assigned to the models represented the aligned osteon microstructure. Once damage was initiated, it evolved according to the XFEM-based cohesive segment. Once a defined critical strain energy release rate was reached, the two fragments of the split element were free to displace away from each other. Surface interactions were also applied to the newly exposed surfaces to prevent unphysical over-closure of split elements with a friction coefficient of 0.3.

The criteria developed were applied to the fracture of cortical bone during screw pullout in 2D and 3D. Previous experimental studies revealed that during screw pullout cracks propagate parallel to the osteon direction. Higher pullout forces were observed for cortical bone with osteons aligned perpendicular to the central axis of the orthopaedic screw (transverse). Simulating screw pullout from cortical bone, using the developed failure criteria and the XFEM method, accurately predicted the failure loads and fracture patterns that were observed for experimental screw pullout. Cracks were predicted generally parallel to the direction of osteon alignment during longitudinal and transverse screw pullout, with the latter requiring a significantly higher pullout force.

Previously, models had been developed to simulate fracture of cortical bone at the micro scale that required explicit representation of the micro-structural geometry of cortical bone. Incorporating detailed geometries of the microstructure of cortical bone into macro scale simulations of cortical bone fracture (e.g. during screw pullout) would be too computationally expensive. The possibility of lowering computational demand using simplified material models, with assigned orientations and anisotropic damage initiation criteria, offers an advance on previous models that were limited to the micro scale.

Application of the developed damage criteria using XFEM to predict screw pullout highlights the potential for both device design evaluation and cortical bone fracture fixation studies. A simplified macro scale representation of cortical bone, which accurately predicts both the fracture toughness and crack patterns, offers a level of complexity essential to fracture fixation studies. This complexity is achieved using XFEM without elevating computational demand beyond a viable threshold.

For More Information
www.nuigalway.ie
www.3ds.com/SCN-October2012

Assembled Fasteners in Abaqus/CAE

How to use the Abaqus/CAE Assembled Fasteners tool to efficiently model complex fastener behaviors in a large number of model locations

Assembled fasteners replicate fastener-like behavior in many places. This tool uses a *template model* that contains a fastener-like construct. Once defined, the fastener template can be used many times in a large model such as an airframe or automobile.

Here's What You Do:

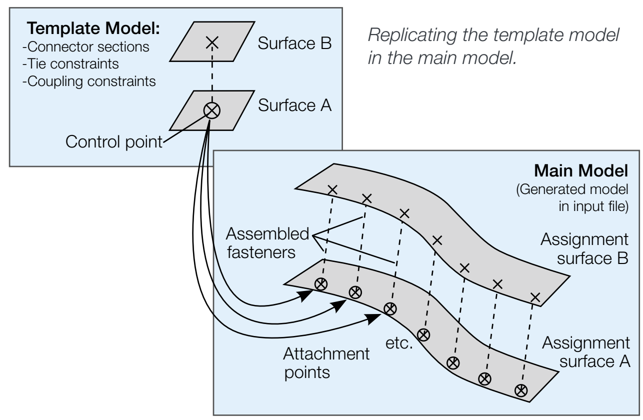
A. Create the **template model** containing a fastener-like construct.

1. Define the template model using any of the following:
    - Beam, solid, cohesive, and connector elements
    - Beam, homogeneous solid, cohesive, and connector section assignments
    - Tie, coupling, and adjust point constraints
    - Mass inertia
2. Assign **names** to all surfaces involved in constraints.
3. Create a single-point set for a **control point**, which will be used to position the template model copies in the main model.

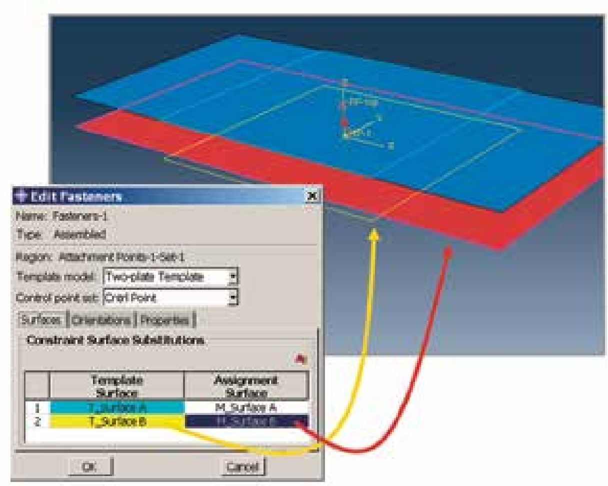
B. In the **main model** (the assembly that will include the fasteners):

1. Create **attachment points** or **reference points** at the locations where the template fastener will be positioned.
2. Assign **names** to the surfaces that will be involved in the assembled fastener constraints.
3. In the Interaction module, select **Special→Fasteners→Create**.
4. From the **Create Fasteners** dialog box that appears, select **Assembled** fasteners, and click **Continue**.
5. Use the **Edit Fasteners** dialog boxes that appear to define how the template model will be read, assigned, and oriented.
    - At each attachment point, the template model is positioned and translated so that the **template model control point** coincides with the **attachment point**.
    - The template model is oriented such that the positive Z-axis of its global coordinate system aligns with the coordinate system specified on the **Orientations** tabbed page of the **Edit Fasteners** dialog box.
        - The default is to orient the Z-axis of the template model along the normal vector of the first surface in the main model.
        - If necessary, you may use a uniform user-defined coordinate system to orient the template.
    - The X-axis of the assembled fastener is then defined by projecting the X-axis of the template onto the first surface.

    When the **Edit Fasteners** dialog box is committed, the template model properties are copied into the main model.
6. Both **field and history output** can be obtained from assembled fasteners in the main model.



Replicating the template model in the main model.



*Pairing template surfaces with main model surfaces in the **Edit Fasteners** dialog box.*

For More Information

Login to view video: swym.3ds.com/#post:8277

2013 SIMULIA Community Conference

SAVE THE DATE! May 21–24, 2013, in Vienna, Austria

Network with peers, engage with SIMULIA managers, and enhance your technical knowledge at our 26th annual international gathering of simulation experts. Many industry-leading companies will be in attendance to share with you how they are using realistic simulation solutions to explore design options, improve product performance, and accelerate innovation. Be a part of the community and spark innovation!

Learn More: www.3ds.com/scc-2013

e-Learning Videos in the SIMULIA Learning Community

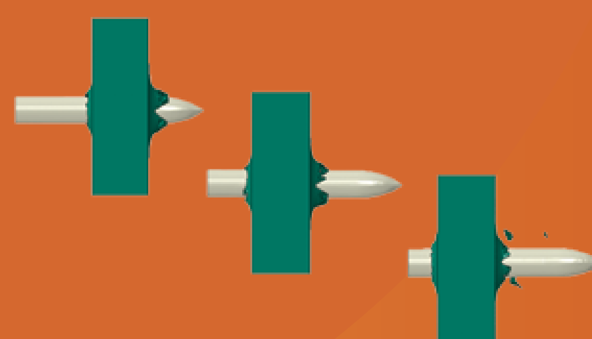
The Surface Mesh and Tetrahedron Fill Meshing Techniques in SIMULIA V6

Get in the know! The SIMULIA Learning Community offers dozens of valuable tutorials and e-Learning videos on capabilities in Abaqus, Isight, and our new V6 simulation products. A recent and popular e-Learning video provides a quick overview of the steps to use the surface mesh capability in SIMULIA V6 and then use the tetrahedron filler to generate a 3D mesh. *(Login required)*

Watch this video: swym.3ds.com/#post:14155

New Technology Brief

Simulation of the Ballistic Perforation of Aluminum Plates with Abaqus/Explicit



Application of high-velocity impact dynamics in today's engineering practice allows to account for the effects of penetrating fragments, accidental loads, and collisions on various structures. Accurate computational simulation of ballistic impact events can provide physical insight into the perforation process that might be beyond experimental ability, shorten the product development cycle, and reduce cost in various industries.

Download the Tech Brief: www.3ds.com/SCN-October2012

New Customer Testimonial Video

Designing-in Lasting Clothing Comfort with Abaqus

Mechanical Design and Analysis Corporation (MDAC) believed in improving clothing designs to provide comfort, enhanced motor skills, in addition to medical improvements in the nervous and circulatory systems. Takaya Kobayashi, a company executive of MDAC, describes how his company applied Abaqus' analysis capabilities and the benefits they derived from doing so.

Watch this video: www.3ds.com/SCN-October2012