SIMULIA
COMMUNITY NEWS

#09 February 2015

INDUSTRY DRIVERS
COMPANIES SEEK INNOVATION, SUSTAINABILITY AND COST REDUCTION

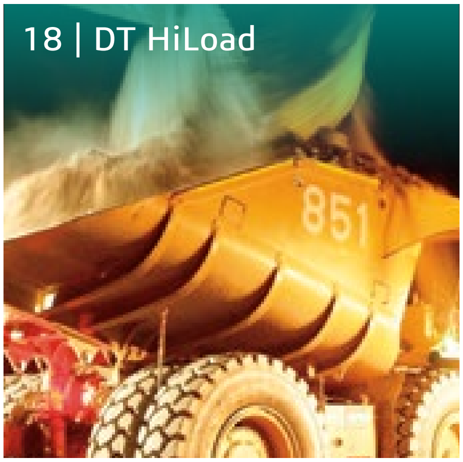
COVER STORY
PLASTIC TECHNOLOGIES, INC.

DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

6 | Plastic Technologies

14 | CEA

18 | DT HiLoad



In this **Issue**

February 2015

Contributors: Sumit Mukherjee (PTI), Jeff Williams (Baker Hughes), Max-André Darizcuren (CEA), Stefan Sicklinger (Technical University of Munich), Ray Sun (DT HiLoad), Parker Group

On the Cover: Sumit Mukherjee, Director, CAE & Simulation, Plastic Technologies, Inc. Cover Photo by Rick Luetkke of Luettke Studio

---

SIMULIA Community News is published by Dassault Systèmes Simulia Corp., 1301 Atwood Avenue, Suite 101W, Johnston, RI 02919, Tel. +1 401 531 5000, Fax. +1 401 531 5005, simulia.info@3ds.com, www.3ds.com/simulia
Editor: Tad Clarke **Associate Editor**: Kristina Hines **Graphic Designer**: Todd Sabelli

3DS SIMULIA

SETTING THE BAR EVEN HIGHER

We are off to an exciting start for 2015, as we continue to explore the real-world behavior of product, nature and life. Each and every year you, our customers, help to push the boundaries of what exists today and what we dream of for tomorrow. As a result, the bar is set high, and we strive to meet it.

We continue to expand our portfolio of products, acquiring SIMPACK and integrating fe-safe, Tosca and Simpoe-Mold in 2014. You are using these tools in many ways—from major tunnel renovations and construction in one of the largest cities in the world to patient-specific heart stent designs and tire material simulations for fuel efficiency and rolling resistance. The list goes on—and although I can't name all of your successes in this letter, I would like to recognize them and wish you many more in 2015.

In this issue, we look at customer applications in four industries: consumer packaged goods, industrial equipment, aerospace and defense, and energy processes and utilities. As you'll read, the value of using SIMULIA products extends beyond single applications and industry use cases.

We see this principal at work in both the Plastic Technologies, Inc. [pg. 6] and DT HiLoad [pg. 18] case studies as they strive to find lightweight solutions for their products. The end result for the consumer packaged goods industry is less material, reduced costs and support for the development of more sustainable products. Similarly, for industrial equipment we see a lowering of costs but also a significant reduction in weight and, in the case study featured in this month's issue, an increase in the capacity of a huge mining-truck tray.

In the energy arena, you'll see a state-of-the-art wind turbine CFD analysis, and our Academic Update section shines a spotlight on the increased need for openness within industry applications. Whether you are committed to safeguarding the robustness of the product through its lifecycle (as with the CEA case study [pg. 14]) or challenged to reduce cost and eliminate waste, we are thrilled to partner with you.

There are many reasons to be excited and inspired for the coming months based on accomplishments in 2014. We embarked on a new five-year collaborative research agreement with the United States' Food and Drug Administration. This project will utilize the technology developed through Dassault Systèmes' Living Heart Project and will initially target the development of testing paradigms for the insertion, placement and performance of pacemaker leads and other cardiovascular devices used to treat heart disease.

We also moved into a new facility that was designed in the spirit of collaboration—a hallmark of our work environment—and includes such features as a Customer Visit and Training Center and state-of-the-art meeting and collaboration rooms. Add to that our redesigned website (www.3ds.com/simulia)!

Let's work together to keep this exciting momentum moving forward! I am eagerly anticipating seeing many of you in May at the SIMULIA Community Conference (SCC) in Berlin, Germany. Technical content will be front and center. The abstracts received (which number more than ever before) offer exciting, new industry applications in an impressive variety of areas, and show the Power of the Portfolio at work. Our new training day promises to deliver best-in-class learning opportunities for attendees. You can register at www.3ds.com/scc.

Best wishes,

Bruce Engelmann
SIMULIA VP & CTO

SIMULIA COMMUNITY CONFERENCE

MAY 18–21, 2015 | BERLIN, GERMANY

Please join us for the biggest, best
SIMULIA Community Conference ever!

DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

TECHNICAL CONTENT

Hear the latest about our mid-2015 versions of Abaqus, Isight, Tosca and fe-safe during our Technology & Product Updates. Speak with SIMULIA developers and experts during our Meet the Expert sessions.

TRAINING DAY

We have revamped our Advanced Seminars day into an all-new Training Day on May 18, including three technology seminars, two hands-on training classes and two portfolio application courses.

FORUMS

In Berlin, we'll feature not just one, but three forums showcasing the latest in Multibody Simulation (SIMPACK), Fatigue & Durability (fe-safe) and Optimization (Tosca and Isight).

NETWORKING OPPORTUNITIES

Whether you are seeking feedback or discussing your point of view, the SCC provides ample opportunities to interact with your peers and SIMULIA experts.

For more information, visit www.3ds.com/SCC

PTI TAKES A DEEP DIVE INTO CONTAINER SIMULATION TO GAIN THE LIGHTWEIGHTING ADVANTAGE

With PET (polyethylene terephthalate) firmly ensconced as the material of choice for plastic container manufacturers, the focus has shifted to designing lighter-weight versions of current packaging to wring costs out of the equation and quench consumers' thirst for more sustainable products.

Although PET has been around for more than 40 years, advances in resin technology and conversion equipment have aided manufacturers' lightweighting efforts as they try to improve their environmental profile. Research conducted in 2010 for the International Bottled Water Association (IBWA) found that the average gram weight of a 16.9 oz. single-serve water bottle had shrunk by 32.5% over the prior eight years[1]. While the predominant current weight of 500ml PET water bottles in the United States is now as light as 8 grams (g), new launches are trying to squeeze that down even further.

Yet zeroing in on the optimal design that can offset lightweight material composition with a structure that still meets stringent performance requirements is no easy matter. In fact, it's a delicate balancing act that has sent brand owners, bottle manufacturers, resin and machine manufacturers scrambling to embrace innovative design processes that can help them achieve their goals.

At Plastic Technologies Inc. (PTI), a leader in plastic package design services, the go-to practice is simulation-led design that combines an in-house virtual-prototyping tool with finite element analysis from Dassault Systèmes' realistic simulation application SIMULIA, a key component of the Perfect Package Industry Solution. Simulation enables the PTI design team to greatly reduce the amount of time and resources spent on building and testing physical prototypes.

PTI has a broad swath of clients ranging from consumer product companies to plastic processors and material suppliers. In a typical year, the firm performs as many as 500 unique design developments and an average of five iterations per design for clients, and it recently capped off its 10,000th bottle design since its inception in 1985. PTI employs state-of-the-art CAD tools, including Dassault Systèmes' CATIA application, in the development of innovative designs for its customers.

Abaqus is often employed in concert with PTI's proprietary virtual-prototyping software. This is used to simulate the reheating of preforms, replicate the blow molding of containers and predict the material thickness distribution of associated mechanical properties. The data is then used as input for FEA studies that explore the highly nonlinear deformation of different containers under various types of loading conditions.

> "If new package designs can be simulated before any samples are made, bottle and preform design iterations happen more quickly and cheaply, speeding up the entire design chain and resulting in quicker time to market."
>
> —Sumit Mukherjee, Director, CAE & Simulation,
> Plastic Technologies Inc.

---

[1] An analysis performed by the Beverage Marketing Corp. for the International Bottled Water Association, www.foodbev.com/news/earth-day-finds-weight-of-plastic-bottles-reduced-by-32#.VC13nOvgWes

Abaqus has become an essential part of PTI's development process, helping designers address mounting workload pressures by more quickly screening designs, identifying the best lightweighting opportunities, optimizing the production process and identifying root-cause failures.

"Our simulation results help in the development process in multiple ways," says Sumit Mukherjee, Director, CAE & Simulation at Plastic Technologies. "It helps screen the most promising candidates that are then refined further for optimum performance. It lets us provide critical feedback on what geometric features, material properties and thickness distribution will guarantee better performance and possible lightweighting for existing containers, and it encourages more innovative thinking to facilitate creative, viable solutions."

ADVENTURES IN CONTAINER PERFORMANCE

PTI has two primary objectives when lightweighting container designs for its customers: to achieve materials savings without greatly affecting structural performance and to enhance both container structure and preform designs to improve the efficiency of material distribution so that each grain of material is maximized.

In one study that explored plastic water bottle performance during lightweighting, PTI used Abaqus in simulations of what happens to top-load strength and/or side-wall rigidity under varying pressures to ensure that a lighter plastic water bottle would not buckle under loading or stacking conditions. Surprisingly, it was revealed that the top-load strength of a plastic bottle drops nearly in half from 19 lbf (pound force) to 12 lbf as the container is light-weighted from 17g to 14g—a critical finding that allowed the company to redirect its design efforts quickly.

The simulated data was physically validated by molding samples at both 17g and 14g, and the predictions were within 90% of the actual observed values.

Abaqus was tapped to explore the orientation and wall thickness of the bottle sidewall to identify the optimal preform dimensions that would result in the desired top load. "Abaqus allows us to incorporate the right loads and boundary conditions along with the time scale of the particular applications," Mukherjee says. "Going forward, the potential for incorporating the structural dynamics with fluid contact interactions [as is typical of all containers encapsulating a fluid product] is perhaps the single most redeeming feature of Abaqus for us."

DESIGN CHALLENGES COME IN ALL SHAPES AND SIZES

PTI also studied top-load and side-load performance of oval and other non-round containers to determine the different outcomes produced by different preform heating methods during the blow-molding process. In this case, PTI engineers learned that the "preferential heating" method is the preferred option for oval-shaped containers as it resulted in a more uniform weight distribution with better empty and filled top-load strength.

"This ability to analyze different container shapes and preform designs in a relatively short time meant a wider variety of design features could be rapidly evaluated while providing a good learning tool for future design recommendations," Mukherjee says. "This opens the door for evaluating more creative concepts that may fall outside of the scope of the previously prototyped concepts and allows for the development of more robust designs much more rapidly."

GOING BEYOND LIGHTWEIGHTING

Abaqus FEA is not only facilitating PTI's lightweighting efforts, it also plays a role in other areas of the design-and-build workflow, including optimizing a more complex blow-molding process and aiding in ongoing efforts to identify and fix product failures.

Consider the design and manufacture of wide-mouth PET containers, which are produced via a single-stage blow-molding process as opposed to the conventional two-stage process. The large neck diameter of this class of PET containers requires the single-stage process because the retained heat from injection molding can be harnessed for preform stretching and blow molding. Preform design is particularly critical because there is little room for redistribution of material with this approach. Historically, however, the complexity of the single-stage process has precluded PTI and others from accurately simulating

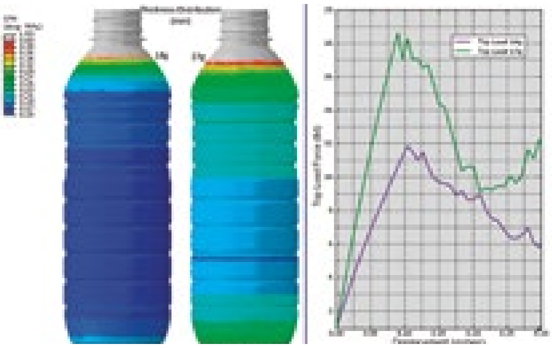


Figure 1. Top Load strength is predicted to drop nearly in half as container is light-weighted from 17 to 14 grams with thickness distribution comparison on left.

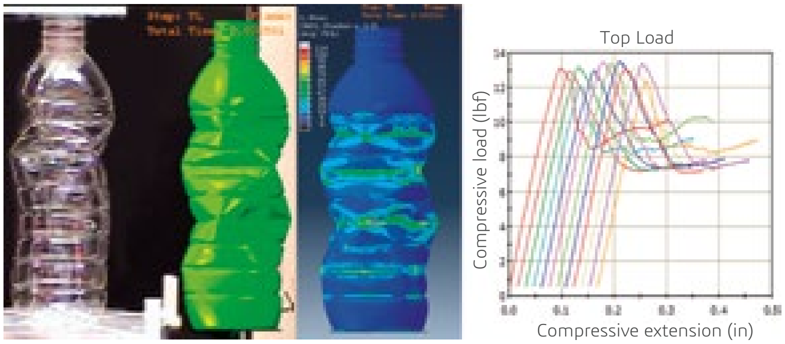


Figure 2. Comparison of actual and simulated Top Load performance of 14g container.

preform design and the actual blow-molding process in a virtual world, continuing a reliance on costly physical prototypes.

Now, building on its success modeling blow-molded wall-thickness distribution on round and oval containers, PTI was able to create its first-ever analytical model that addresses these specific challenges, simulating a pure single-stage process on a 116g 3L container that was blow molded without any additional reheat conditioning. By inputting the sidewall thickness and mechanical property outputs from virtual-prototyping software into Abaqus FEA, PTI wanted to predict container top-load resistance, the goal being to zero in on the optimal container size that could hold a kilogram of printer color toner.

PTI's simulations of this previously hard-to-replicate design and manufacturing process proved to be nearly identical to the results of physical testing. The failure mechanism and deflection at max load were also near mirror images between the simulation and the actual tested bottles. Abaqus' precise contact algorithms and its ability to efficiently map material properties to individual nodes and elements was instrumental in helping PTI pull off a simulation of this magnitude, Mukherjee says. "We had used other software in the past, but it didn't meet our needs for large deformation of thin-walled plastic articles with nonlinear material properties."

Mukherjee notes that having access to all the different solver modules, including structural dynamics and computational fluid dynamics (CFD) in Abaqus, and the option to scale to a higher number of CPUs based on complexity and need, is enabling the team to tackle a wider variety of challenging problems.

GETTING TO THE ROOT OF PRODUCT FAILURES

As packaging becomes an integral part of a company's brand appeal, new containers are testing current processes and domain expertise, introducing novel challenges in the way of package failures. Yet since quality standards vary from one producer to another, there is no absolute guidance on what constitutes failure.

PTI decided to further evolve its simulation efforts to tackle root-cause failure analysis and, ultimately, improve its quality efforts. Armed with its own virtual-prototyping software tools along with FEA software and M-Rule® models, PTI was able to zero-in on reoccurring failures and resolve them in a fraction of the time and with far fewer resources than were previously required using standard prototype-and-test methods.

In one exercise, engineers tested their new failure-analysis process on a 64-ounce hot-fill juice container, which exhibited bulging in the area of the logo panel on sporadic numbers after being filled. Tracking the errors through the system, PTI determined that the failure sample was being generated during the filling process. Further analysis revealed that material distribution inconsistencies were not the cause of failure, prompting PTI to turn its attention to possible external influences. A thesis emerged that high product temperature coupled with filler pressure spikes was the impetus for the panel failure.

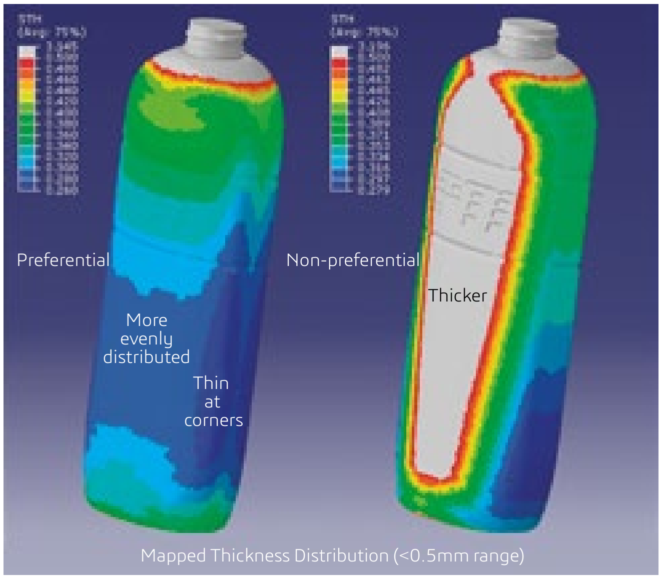


Figure 3. Abaqus FEA simulations revealed that the "preferential" heating method (left) is the better option for oval containers because it resulted in more uniform weight distribution and top-load strength.

In order for simulation to serve as an effective replacement for the traditional root-cause failure inspection process, PTI had to conduct modeling efforts on a grand scale. The model's build began with the replication of the material distribution and mechanical properties, but also included reproducing the blow-molding process in a virtual world as well as the physical aspects of the environment, including conveyors and filling heads. Once the information input was complete, PTI conducted numerous simulations to realistically replicate the failure mechanism. With those validated, the team could then use the model as a baseline to compare design iterations that addressed the failure.

As a result of its efforts, PTI was able to rule out heavy-weighting the juice container design to a 75g version to address the deformation problem and instead concluded that a geometric modification to redesign the logo panel was the optimal resolution to the problem. This very process of continually modifying designs to eliminate failure is exactly where simulation can save time and money.

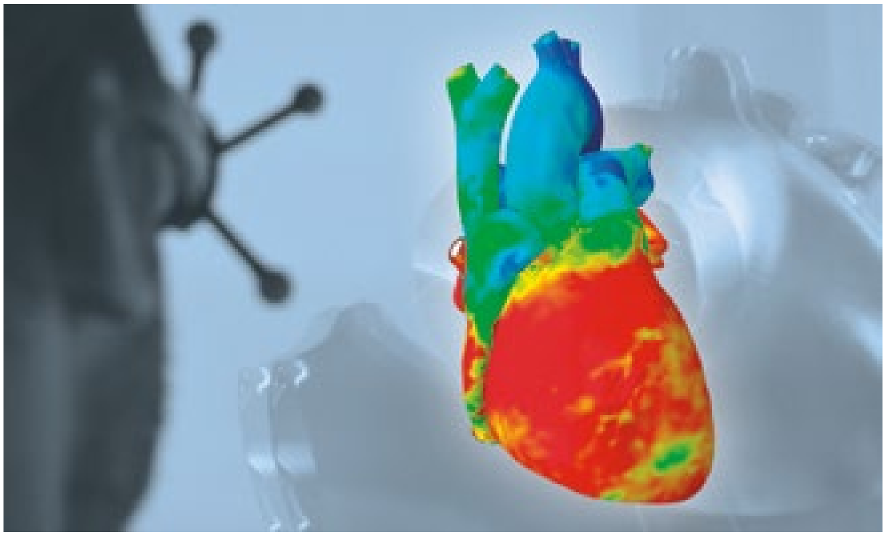
"If new package designs can be simulated before any samples are made, bottle and preform design iterations happen more quickly and cheaply, speeding up the entire design chain and resulting in quicker time to market," says Mukherjee. "A wide variety of design features can be rapidly evaluated, providing a good learning tool for future design recommendations. This opens the door for more evaluating more creative concepts that may fall outside the scope of the previously prototyped concepts and allows the development of more robust designs much more rapidly."

For More Information

www.plastictechnologies.com

DASSAULT SYSTEMES SIGNS RESEARCH AGREEMENT WITH THE FDA FOR "LIVING HEART PROJECT"

The project accelerates towards next generation patient experience for treatment and diagnosis of heart diseases through key milestones and crowdsourcing



If we use simulation to transform the world around us, isn't it time to use simulation to improve the world ***within*** us? If we provide doctors with the same realistic simulation technology that other professionals use to virtually design, test and validate products before they have been built, could we transform the patient experience to make it more effective, more personal?

This is the goal of The Living Heart Project, an initiative launched by Dassault Systèmes in January 2014 to partner with the medical and biomedical community to develop definitive realistic simulation models of the human heart. For other industries that have used simulation in the past, it has taken a sustained and committed partnership to achieve such a goal, and indeed they have succeeded. To address healthcare, this means coordinating members of the research, medical, manufacturing and regulatory communities needed to join forces to bring their expertise to develop standard computational models of heart behavior, making this model a more precise representation of a moving heart than existing cardiac simulations. From its launch a year ago with three committed members, more than 35 contributing member organizations have signed on to help build and validate the model, including the United States Food and Drug Administration.

Through this accelerated, crowd-sourced approach, we were able to produce the first commercial-grade simulated model of the human heart. Further adding to project momentum is the FDA, which has signed a five-year research agreement that will initially target the development of testing paradigms for the insertion, placement and performance of pacemaker leads and other cardiovascular devices used to treat heart disease. The hope is to not only fast-track the approval processes for new cardiac devices but to inspire innovation within medical device design and manufacturing and provide patients with a greater sense of security and empowerment when it comes to their own health.

The researchers have teamed with the Medical Device Innovation Consortium (MDIC) with the goal of accelerating the approval process of medical devices while spurring innovation, improving patient reliability and reducing costs. The project has already been used to validate the efficacy of a novel valve assist device before insertion in a real patient and to understand the progression of heart disease.

"Computational modeling and simulation has the potential to revolutionize the medical device and healthcare fields by accelerating innovation and providing comprehensive evidence of long-term safety," said MDIC President/CEO Bill Murray. "It holds the promise of going beyond empirical testing through human clinical trials to evaluating the interaction of devices with the human body that is not obtainable in any other way. The Living Heart Project is a leading example of a new tool that offers the medical device community a heart simulation that could be validated for use from device design to regulatory submission."

At Dassault Systèmes, we recognize the time is right to unite this need with the solution that is at our fingertips. Imagine extending this approach even further – to orthopedics, the brain or eyesight? We could potentially simulate every system in the body. Imagine the benefits to humanity if we continue to push these boundaries. We can use technology to not only improve the process and overall patient care but to empower individual patients as they undergo treatment, and we hope to inspire other communities to follow suit.

For More Information
www.3DS.com/heart

A CARDIOVASCULAR STENT OPTIMIZATION WORKFLOW: HOW THE SIMULIA PORTFOLIO POWERS DESIGN INNOVATION

Cardiovascular disease is the leading cause of death worldwide and is projected to remain so for decades, according to the World Health Organization. Surgical intervention, including the use of balloon angioplasty and stent insertion, continues to be a lifesaver for many patients.

If you are a designer and/or manufacturer of medical stents—cardiovascular or otherwise—your team is continually searching for the most efficient way to produce your product to the highest standards of quality demanded by the life sciences industry and the Food and Drug Administration. You may be asking, "Is our current stent design the most optimal?" "How can we pinpoint what makes a better design?" And, "As we create that 'better' design, how can we accurately assess its fatigue life?"

You may already be using computer-aided engineering to help answer some of these questions. But you may only be scratching the surface with your inquiries so far. Simply delivering a few stress and deformations plots, and letting someone else extrapolate about durability and lifespan, is no longer sufficient.

SIMULATION AS A KEY DRIVER FOR DESIGN INNOVATION

Your company's senior management has become keenly aware of how advanced simulation can drive ***innovation***—and they are demanding more complete, actionable solutions to the design challenges their teams face. They are demanding innovation, not simply simulation, requiring a powerful set of tools that will help deepen your knowledge base about the geometry and physics of stents, expand your skillset and boost your ability to achieve optimum, verifiable results.

This technical demonstration will show you how a suite of tools composed of FEA, process integration, design optimization and fatigue analysis can be equipped to focus on, and drill down into, the many and varied challenges of stent design to foster innovation and promote product quality.

The example provided here is of a generic coronary stent design, not a specific product. We will follow a workflow that starts with SolidWorks CAD modeling, through Abaqus FEA, to parametric optimization with Isight and finally non-parametric shape optimization with Tosca. You will also see how fatigue life assessment with fe-safe can be used as an adjunct to the workflow whenever the analyst desires additional insight into the results of design decisions.

AUTOMATING STRUCTURAL ANALYSIS

First a 2D CAD model of a proposed stent design is created in SOLIDWORKS. Using Python scripting, the model is imported into the Abaqus/CAE environment for 2D meshing, then extruded to obtain a 3D mesh and wrapped (cylindrically) to obtain the final stent mesh.

With the stent model in hand, the next task is to construct the other components of the simulation required for full finite element analysis in Abaqus. These include the blood vessel in which the stent is being balloon-deployed, the devices that expand and then crimp the stent before it's placed inside the vessel, and finally the cyclic pressure load on the interior surface of the vessel caused by blood flow (Figure 1).

A PARAMETRIC WAY OF OPTIMIZING THE DESIGN

So far we have demonstrated the kind of stepwise processes that you would carry out manually using SIMULIA's Abaqus and fe-safe toolkits. You've arrived at a new stent design idea that meets your minimum reliability criteria. Now it's time to apply the powerful process automation and optimization capabilities of Isight that will provide you with even deeper insights by tweaking the relevant parameters in order to determine whether yours is truly the best design possible.

Begin by identifying the parameters in your model that you would like to query, such as length, radius or thickness of a stent segment. You can then set up a Design of Experiments (DOE) sequence in Isight that will automatically modify each parameter, produce a new CAD output file for each modification,

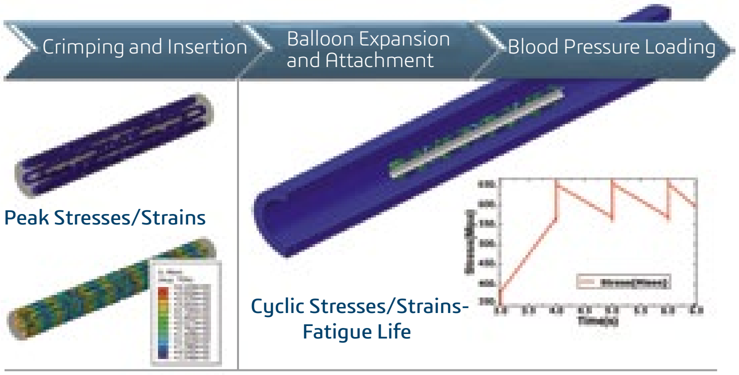


Figure 1

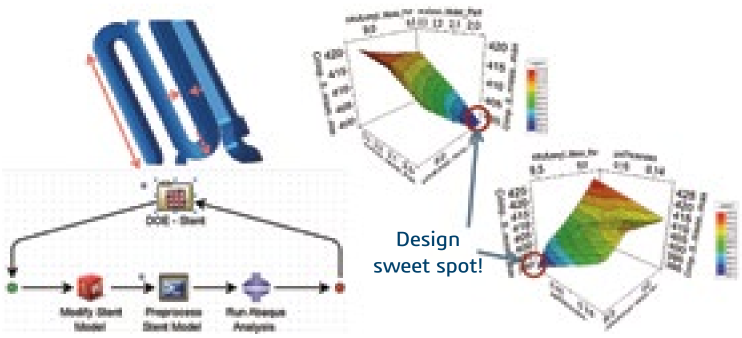
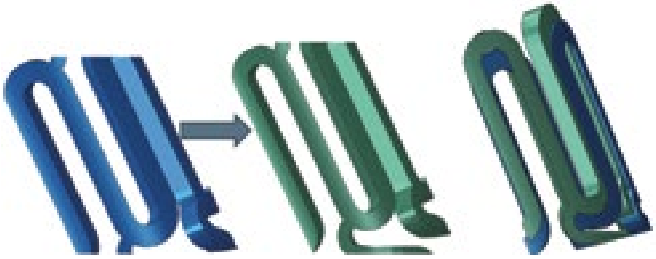


Figure 2

convert each one into an Abaqus analysis and provide feedback about peak stresses in the form of a response surface that maps the results of the parameter modifications (Figure 2).

The example shows the Isight workflow (Figure 2, lower left) used to set up the DOE runs. At right are two views of response surfaces from a stent-crimping simulation, with the "design sweet spot" (blue portion) identified.

Now let's compare the original stent design against the DOE-optimized one:



The figure at far right in this image shows the overlap of the baseline and optimized models. The thickness of the optimized stent design has been slightly reduced; the extrusion thickness has increased while the radius has decreased.

A NON-PARAMETRIC WAY OF OPTIMIZING THE DESIGN

Now you have a stent model that has been optimized using the parametric approach in Isight. Yet you can gain even deeper, richer insight into your design by next using Tosca for non-parametric, shape optimization (Figure 3).

Tosca works by querying the sets of nodes where peak stresses are occurring and searching for the configuration that has the least amount of peak stress. A control algorithm within Tosca homogenizes the stress distribution inside each stent strut, while adhering to defined constraints within the intrinsically, cyclically symmetric model.

Tosca modifies the design shape until, in six iterations in this case, it reaches an optimum decrease in Mises stress values of 13%—significantly more than the 5% reduction that was achieved with the non-parametric DOE-based approach in Isight alone (Figure 4).

WHY PARAMETRIC AND NON-PARAMETRIC APPROACHES?

From the example presented here, you can see the significant value of using Isight and Tosca together to optimize stent designs. It may seem that parametric and shape optimization are two different, even mutually exclusive approaches, but by using them both you end up with a richer, deeper analysis that ensures that you have arrived at your best possible design.

Although the two tools can be set up in whatever sequence is preferred, in the case of stents, where the analyst tends to start from an existing design, it's ideal to begin with a parametric (Isight) approach and then take the results into Tosca for shape optimization. And fe-safe can be used at a number of points alongside the workflow to prove out how such sequential optimization results in measurably increased fatigue life.

ACCESS SIMULIA PORTFOLIO TOOLS FOR STENT OPTIMIZATION ON A SINGLE TOKEN

The full extent of SIMULIA's portfolio for stent design can be explored with the Extended Token program, which allows you access to all solutions—Abaqus, Isight, Tosca and fe-safe—on a single token. The advantages of working with this complete toolset are many:

- Deeper understanding of loading conditions, durability and reliability
- Powerful capabilities for design exploration of materials and geometries, parametric and shape optimization and manufacturing tolerances
- Insights into fatigue and failure through evaluation of stress concentration and cyclic loading
- Support of future patient-specific modeling with customized geometries and loading conditions

For More Information

Email Manoj Chinnakonda at
manoj.chinnakonda@3ds.com

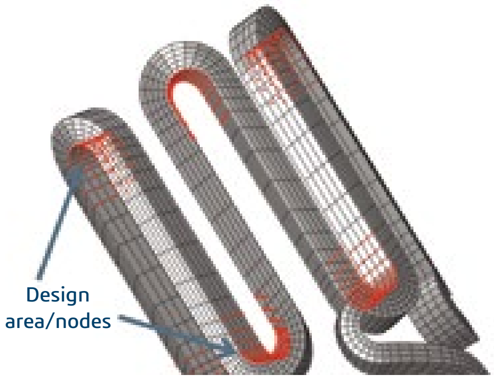


Figure 3

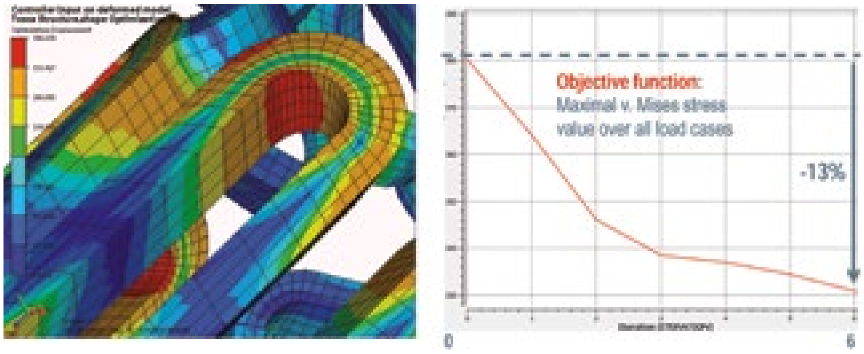


Figure 4

Q&A: BAKER HUGHES' JEFF WILLIAMS EXPLAINS HOW TO ACCELERATE THE ITERATION TRAIN WITH ISIGHT

Baker Hughes is one of the leading oil & gas service companies in the world, with a long history of technological inventions that have revolutionized the petroleum industry. It has been using Abaqus and Isight as key strategic product development tools for some time. We recently interviewed Jeff Williams, project engineer for the wellbore construction group, about his work on downhole seal design for deep-sea wells. He presented a paper on the subject at the 2014 SIMULIA Community Conference.

How does the global push to develop deeper, hotter petroleum reservoirs affect your customers' demands for ever-more robust equipment?

Williams: Currently, the price of oil doesn't seem to make deeper exploration cost effective...but our customers know that this is a highly dynamic market. The project timelines for these deep reservoir finds is much longer than the typical well plans. They plan these projects five to 10 years out, so they want to ensure the metallurgies and technologies are feasible and in place early in the process. Additionally, as tool developers, we are held accountable for the functionality and safety of the products that go into our customers wells...so robustness is paramount. As the wells go deeper, the cost and safety risks increase dramatically.

Why is downhole seal design so critical? Describe the new feature you developed for your "zero-extrusion" seal and the surprising promise it showed.

Williams: Seals in a well construction are necessary for various reasons, but the particular seal I worked on was for a Liner Top Packer. Essentially, a typical wellbore construction consists of concentric layered tubulars...from top to bottom of the well going smaller and smaller diameters. Each tubular "layer" needs two key elements: a way to anchor to the previous "layer," and a way to seal off from the other "layers." For our existing "zero extrusion" seal cross section, I optimized the geometry to accommodate higher pressure differentials. On the path of optimizing, I found that split-rings placed in key locale helped boost the performance even more. Without optimization software like Isight I am not convinced we would have achieved these higher capacities.

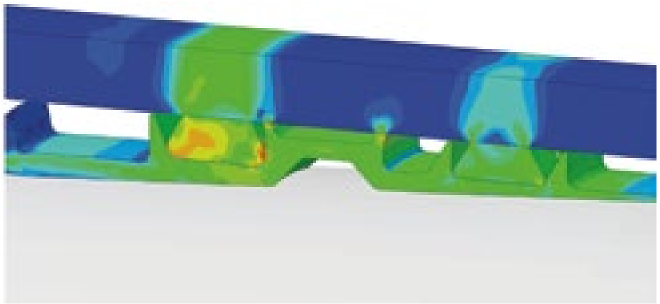
You moved from 3D to a 2D model in order to run and optimize your local Abaqus and Isight models on fewer cores in less time. Talk about how you combined a DOE loop with an optimization loop to give you a trustworthy final output from your 2D iterations.

Williams: When doing either optimization or DOE algorithms separately, there is always a risk of coming to a potential false "dead end" solution, or falling into a valley of the 3D solution subspace. In order to miss the false solutions, combining the two is a recommended practice.

Would an experienced designer have been able to come to the same conclusion?

Williams: I wouldn't think so...it didn't come intuitively to me either. I remember making a note of this potential problem in a SIMULIA Isight training...and have been applying this technique ever since.

> "Anytime you can minimize and/or improve the impact on the supply chain of a product, the better off you will be with new product introduction."
>
> Jeff Williams, project engineer, Baker Hughes



Abaqus FEA analysis of new seal concept for a downhole tubular, developed by Baker Hughes using Isight optimization. [image is from Williams' 2014 SCC paper.]

Your 2014 SCC paper reported results that were "astonishing." How did Isight help you push the boundaries of what could be achieved in seal design and define a new threshold in performance?

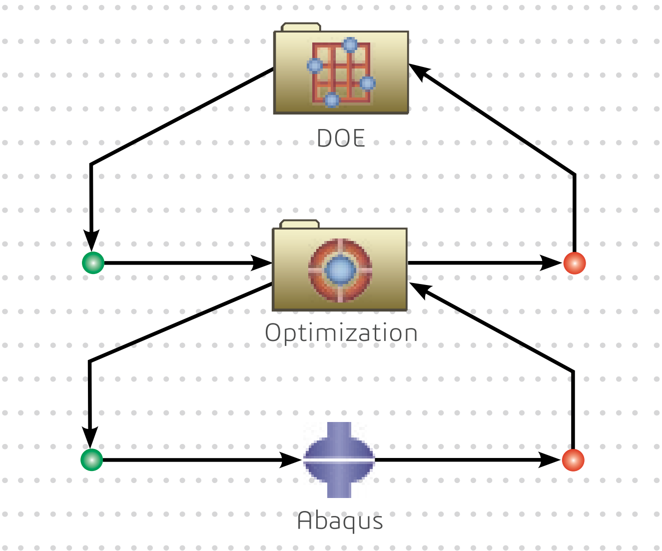
Williams: This is a perfect example of being stuck in a paradigm. You are taught certain rules and inherit designs that have been through years of empirical testing. My role with the company has been to try and ignore the noise and begin my own unbiased evaluation of existing products...see how they work with FEA and then see how I can improve them. Baker Hughes has always been on the cutting edge of new seal design, and that will continue. What I set out to do was see how our existing portfolio could be improved with simple changes. That is the astonishing part...I made simple changes to our existing portfolio geometry with potentially groundbreaking results.

How will such a "simple" seal optimization exercise change your business?

Williams: Anytime you can minimize and/or improve the impact on the supply chain of a product, the better off you will be with new product introduction. Optimizing existing seal packages and methodologies within our company allows us to minimize the impact on the supply chain while still meeting our customers' more aggressive goals.

What other areas will Abaqus and Isight help you explore in the future?

Williams: Let's just say there are even more exciting things coming for our Liner Hanger product line this year, where Ultra-HPHT could be considered fringe technology...we will be rolling out a new mainstream HPHT seal design we are all thrilled about. I hope to tell everyone the development story at next year's SIMULIA Community Conference.



Isight sim flow path utilized.

For More Information

The full text of Jeff Williams' 2014 SCC paper is available at www.3ds.com/fileadmin/PRODUCTS/SIMULIA/PDF/scc-papers/jumping-iteration-train-using-isight-advance-downhole-14.pdf

TERA 100 supercomputer

SIMULIA AND CEA COLLABORATION IMPROVES HIGH-PERFORMANCE COMPUTER SIMULATION WITH ABAQUS

FEA architecture is optimized for parallel processing on TERA 100 supercomputer

Ever since French citizens Pierre and Marie Curie discovered the radioactive properties of radium in 1898, nuclear research has been a driving force for technological innovation in France. The country's atomic energy commission, the Commissariat à l'Energie Atomique et aux énergies alternatives (CEA), was established by General Charles de Gaulle after the Allied victory in World War II with the goal of pursuing nuclear applications in energy, medicine and defense.

The CEA/DAM (Military Applications Division) has since developed into one of the largest scientific computing complexes in Europe (one supercomputer used for defense, two others for industrial partners and researchers throughout the region), with more than 150 engineers working in the fields of software development, scientific computing, and computer architecture and operation.

TEST BAN SPURS THE USE OF SIMULATION

As one of the three nuclear powers in NATO (along with Britain and the U.S.), France stopped real-world testing of its arsenal in 1996 (Figure 1). However, to ensure that the country's nuclear deterrent would remain effective into the future, the CEA/DAM committed to a simulation program that would deliver high-performance, petaflop-class computing to its users.

"Our goal is to ensure the reliability and safety of our nuclear defense system through the use of simulation," says Max-André Darizcuren, deputy head of the CEA Sciences of Simulation and Information Department (DSSI). Darizcuren spearheaded a 2004 project for managing the lifecycle of weapons through computer-aided engineering. "The idea behind this program is that, once you validate a simulation model, fewer and fewer physical tests are needed," he says.

The software that CEA/DAM has used all along—for thermo-mechanical multiphysics simulation as well as accompanying studies on safety and security—is Abaqus FEA from Dassault Systèmes SIMULIA.

ABAQUS PROVES ITS WORTH

CEA first began using Abaqus in two of its five research sites only, but was so impressed with the accuracy of simulation results and the benefits delivered to various projects that the software was adopted throughout the organization. "We were already using other Dassault Systèmes products (CATIA V5 and ENOVIA SmarTeam)," says Darizcuren, so the acquisition of Abaqus Inc. by DS in 2005 provided a comfort level for further investment in the SIMULIA/Abaqus solution.

"We are committed to simulating all the processes involved in maintaining our deterrent," he says. "We have our own proprietary software for specific capabilities, but Abaqus is among the rare commercial codes we use with it. Abaqus helps us safeguard the robustness of the 'product' throughout its lifecycle. This goes from theoretical studies to manufacturing, performance, even to dismantling."

CEA/DAM's earliest Abaqus simulations were largely 2D, and assumed axisymmetry of the product to keep model size down. However, test data demonstrated that the objects being simulated were not always axisymmetrical: asymmetric 3D models would more accurately capture the full picture in many instances (Figure 2).

NEXT MOVE: SUPERCOMPUTING

CEA was clearly in need of more compute power in order to manage the broad range of advanced simulations that would capture all such data and realistically predict multiple phenomena (models could potentially contain more than 100 million degrees of freedom). Then in 2010 the organization purchased a TERA 100 supercomputer (from Bull) dedicated to the deterrence only. At that time, it was the most powerful machine in Europe (it's still in the top 12 regionally, and ranks 35th worldwide).

Figure 1. Abaqus simulations are performed on the French alternatives energies and Atomic Energy Commission's TERA 100 supercomputer, which supports maintenance of France's nuclear deterrent following the country's 1996 test ban.

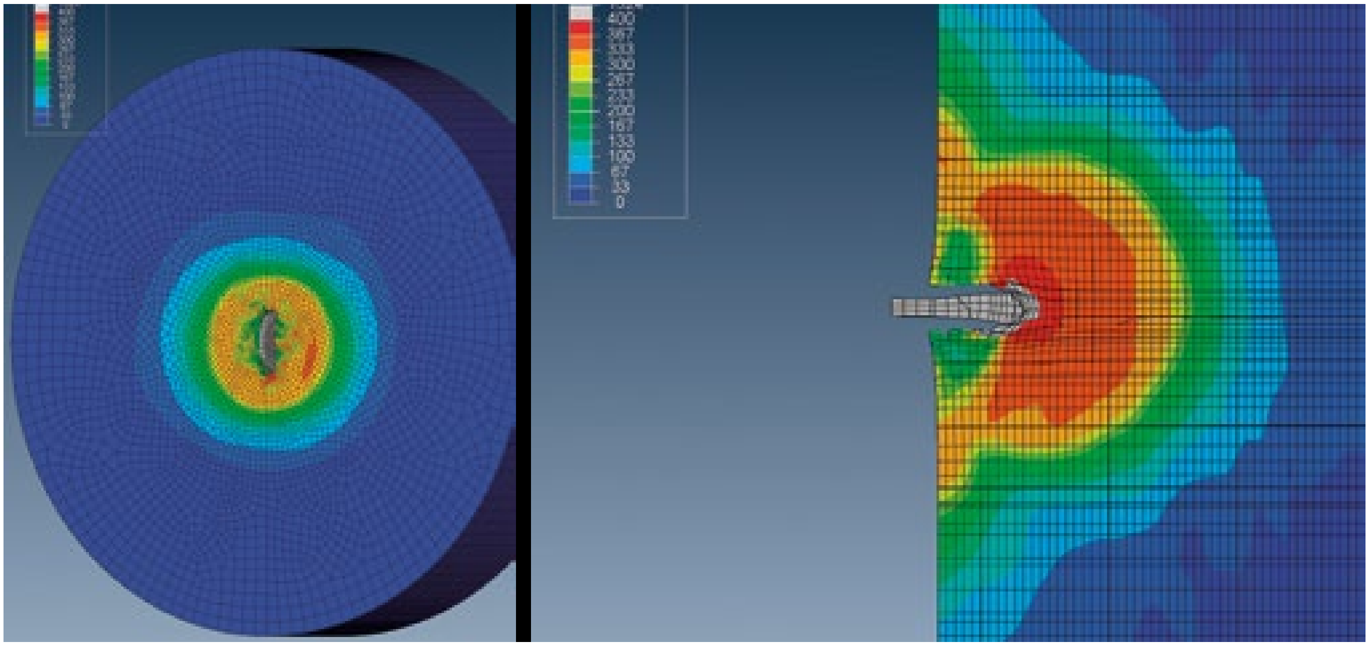


Figure 2. Abaqus simulations of impact test on aluminum plate.

CEA collaborated closely with SIMULIA in a project aimed at customizing the architecture for parallel computing to optimize the use of Abaqus on the TERA 100. "We want to maximize the productivity of the hardware infrastructure available for Abaqus simulations," says Francis Belot, the CEA's IT infrastructure technical manager Françis Belot. "The TERA 100 is well tuned for our own codes, which are very scalable (simulations often run on more than 4,000 cores), and the challenge was to develop the same level of performance with Abaqus on this particular machine over the long term."

The project tuned the Abaqus software to the new machine environment, helped the team master the use of Abaqus on the CEA/DAM infrastructure, determined the limits of model size on that infrastructure and optimized the number of cores used (generally between 64 and 256) per model type. Improvements have been realized in accuracy/validity of the models, multiscale intensive simulations, and increasing use of full 3D models.

The know-how acquired operating TERA 100 is being shared across two other CEA supercomputers, one used for European research and the other for a dedicated industrial partner.

"We've now significantly increased the performance of Abaqus on the TERA 100," says Darizcuren. "We're achieving parallel computing on hundreds of cores on a regular basis, irrespective of the method of calculation used." This significantly reduces simulation time and enables CEA engineers to shorten their design cycles (Figure 3). "We hope to maintain our technical collaboration with Dassault Systèmes in the future in order to ensure that CEA achieves the targeted models that are key for our program. Going forward, it is clear that we will need Abaqus to scale on even more cores."

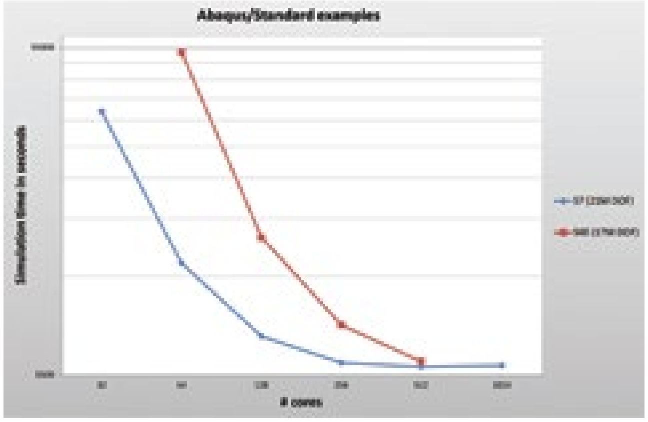


Figure 3. Graph showing overall time needed to run two different Abaqus/Standard models (S7 and S4E) on the TERA 100 supercomputer, versus the number of computer cores used. "DDL" = Degrees of Freedom. For implicit solvers, the scalability is linked to many factors, such as memory availability, speed of input/output, density of stiffness matrix and so on. In these particular examples, Abaqus shows runtime gains on up to 512 cores. The sweet spot (the optimized number of cores used/time run) lies between 128 and 256 cores.

To that end, CEA will be bringing a TERA 1000 machine online in 2017, a step on the road map to exaflop range. "The demand for increasingly powerful simulation is growing fast across every industry," says Darizcuren. "From the success we are seeing with our collaboration with SIMULIA, we are certain that Dassault Systèmes will be identified as a key player in high-performance computer software in the future."

For More Information

www.cea.fr

SIMPLEWARE AND ABAQUS WORKFLOWS OPEN UP NEW SIMULATION OPPORTUNITIES

Our joint efforts address new workflows in reverse engineering, 3D printing, materials science, oil & gas and non-destructive evaluation

By Gareth James

Obtaining high-quality computational models from image data such as CT and MRI can be a challenge, particularly when dealing with complex geometries like the human body or composite materials. CAD-based approaches often simplify structures and reduce simulation accuracy. Simpleware has established software solutions for Abaqus users, opening up a more direct route from 3D scans to design and simulation applications; this covers industries such as the life sciences, materials science, oil and gas, non-destructive evaluation, reverse engineering and 3D printing.

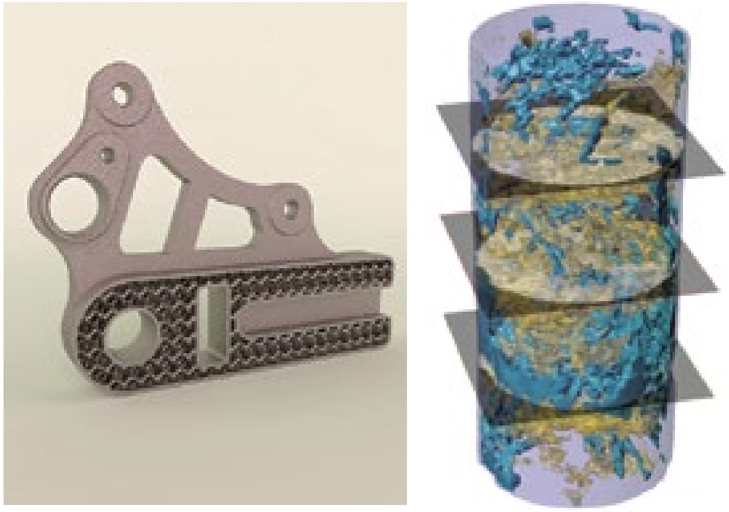
With Simpleware, users can import and explore image data using powerful visualization and animation tools. Regions of interest can be easily segmented and data processed to enhance image quality. Extensive measurement and statistical tools can then be used to quantify data. Options are also available for integrating CAD objects with image data, as well as for calculating the effective material properties of scans through newly implemented homogenization techniques. Processed data can be exported as STL, surface meshes and point clouds to CAD packages and 3D printing, or as high-quality volume meshes for CAE.

Dedicated options are available for exporting Simpleware models to Abaqus for simulation, enabling comprehensive insights to be gained into material structures and design attributes. As with Abaqus/CAE, powerful scripting capabilities, including Python compatibility, are available in Simpleware to allow routine workflows to be automated and customized.

For many years, multi-part Simpleware models containing anatomical data and CAD-designed implants have been used to simulate medical device performance in Abaqus. More recently, however, the potential uses of Simpleware software and Abaqus have expanded to complex industrial applications, including oil and gas, reverse engineering and 3D printing.

In the case of high-growth industries like oil and gas, typical software workflows include generating 3D models of core samples in Simpleware software for Abaqus simulation. With Simpleware software, pore networks can be visualized, statistics generated for fractures, grains and other features, and multi-part meshes prepared for export to Abaqus to simulate processes such as fluid-structure-interaction.

Similar success has been found in the field of industrial reverse engineering and non-destructive evaluation. For example, automotive and aerospace R&D researchers can scan parts, create models and export to Abaqus to simulate the impact of wear and tear and corrosion. This approach can be used to improve part quality.



(Left) Weight-saving lattice structure added to an industrial part for Additive Manufacturing. (Right) Visualization of a core sample in Simpleware ScanIP.

In terms of 3D printing, exciting new developments are emerging for reverse engineering printed objects for finite element analysis, enabling detailed validation of designs before they go into production. Scans can be converted into 3D models and processed in Simpleware software before being exported to Abaqus to investigate physical properties. Validated designs can then be 3D printed, with Simpleware software also capable of adding weight-saving lattices to models.

As scanning modalities become more sophisticated at representing structures, compatibility between image processing and simulation software will be crucial to ensuring robust and flexible workflows. Simpleware and SIMULIA are ideally positioned to meet these complex challenges, significantly reducing modeling time and increasing model quality when preparing and investigating image data.

For More Information

info@simpleware.com

www.simpleware.com

Case Study

MINING DUMP TRUCKS ROLL ON FOR THE LONG HAUL WITH ABAQUS

DT HiLoad simulates payloads and fatigue on steel truck trays for lightweight, durable designs

Mines are a demanding environment for workers—and for their dump trucks as well. These behemoth machines, designed to be exceptionally rugged, face daunting environments and challenges.

"Dump trucks for mining sometimes travel over terribly uneven road-haul surfaces," says Ray Sun, Senior Engineer, DT HiLoad Australia (Perth, West Australia), a leading global manufacturer of customized truck trays. "They carry enormously heavy payloads—sometimes overly so—and they undergo continual fatigue and wear." Obviously, strength and durability are prime considerations in designing truck bodies.

And there's another vital design criterion for the trucks: Their bodies must be as lightweight as possible. A lighter body can devote more weight to its payloads and also save fuel, improving mining efficiency and productivity as well as potentially reducing the size of a company's truck fleet.

The tray is to a mining truck what a bed is to a pickup truck: its reason for existing. DT HiLoad's trays are like works of art, with sleek front walls that jut above the driver's cab and culminate in a guard canopy that extends protectively over the cab, looking as streamlined and graceful as the prow of a ship.

But it's not the aesthetics that sell the truck trays: It's their ability to be customized for, and hold up under, all manner of loads and environments. DT HiLoad's Hercules high-performance dump truck trays operate worldwide on nearly all makes and models of mining trucks in a varied range of mines—usually open-pit—from coal to gold and even diamonds.

Load characteristics such as density and stability are often quite different from site to site. "Usually, every customer and even every mine has different trucking conditions," Sun says. "For instance, material composition, site restriction, and haul road surface can vary a great deal. That's why each tray project can be different." In addition, overloaded haulages are a fact of life in mining, providing additional load and fatigue on the trays.

Designing a tray is no small endeavor. The largest one the company supplies is for a Caterpillar 797F with a maximum capable payload of 380,000 kg (418 tons) and an overall truck length of nearly 50 feet. "The strains that heavy payloads put on a truck body are considerable," Sun says. "And it changes with every turn, start-up and application of the brakes."

Given the drive to achieve light weight in a truck tray without sacrificing strength and long life, computer simulation of these components under loads is a must. The company has used finite element analysis (FEA) since its founding in 2003. Their tool of choice? Abaqus software from Dassault Systèmes SIMULIA.

The dump truck tray extends over the driver's cab and can be as long as the entire vehicle.

LOADING UP FOR ANALYSIS

"Abaqus was introduced to us by our initial FEA contractor," Sun says. "They proposed it as the most suitable analysis software for our application. It provides us with almost full control of the objects we simulate." It was also a plus that Abaqus is compatible with SolidWorks and fits seamlessly into product development at DT HiLoad.

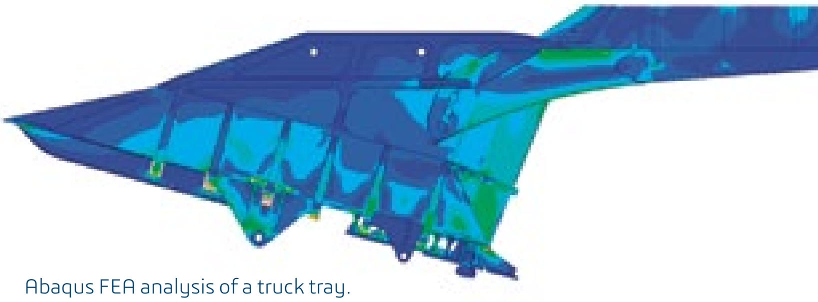
DT HiLoad purchased Abaqus software through Simuserv Pty Ltd., which is headquartered in Melbourne and has offices in Perth. "Since 2002, Simuserv has provided high-quality simulation consulting services to a range of industries," says Gerd Diegelmann, director at Simuserv. "The Abaqus Unified FEA product suite offers powerful and complete solutions for both routine and sophisticated engineering problems, and we both recommend it and offer support. Simuserv has worked closely with us to help us get the most out of our FEA," says Sun, "They've helped us achieve a high degree of confidence in the accuracy of our analyses. Their Perth office has been extremely convenient to work with."

DT HiLoad currently uses Abaqus to design truck trays, and in the future they envision expanding their analyses to other tasks, including service. "When we have time, we hope to apply Abaqus to the Wear Management Program," Sun says. This maintenance program greatly reduces body operating costs and eliminates the installation of traditional, time-consuming, expensive liner plates that increase truck weight unnecessarily and thereby reduce the payload. At some point, engineers at the company want to apply in the Wear Management Program the lessons learned by analyzing truck trays.

FINITE ELEMENTS AND FLEXIBILITY

With truck-tray design, a crucial task is ensuring continuous transition of stiffness throughout the tray, eliminating highly stressed areas as much as possible and keeping fatigue buildup to a minimum. "We couldn't build quality truck trays without simulating their performance," Sun says. "FEA is crucial to helping us find the optimum design configuration for each new application."

The unique design concept for a DT truck tray differs greatly from conventional dump-truck beds. Other manufacturers' trays are often rigid and bulky to bear up under the loads imposed by payloads. By contrast, DT HiLoad's Hercules curved truck tray is made entirely from flexible, hard-wearing steel plate, offering greater fatigue management than its stiffer counterparts. "The DT tray uses its flexibility to help dissipate excessive energy—both dynamic and static—throughout the tray," Sun says.



Abaqus FEA analysis of a truck tray.

Engineers model the entire truck tray, but some areas are of special interest. "The transverse beams and rear support rails are the most important features to simulate," Sun says. "This is because, though the tray body with its curved floor front wall and canopy acts as a flexible structure and mitigates excessive localized loads, the tray requires some stiffness to carry a payload."

The main frame—the beams and rails—compensate for that stiffness, resulting in their undergoing more loading, and more fatigue. "They are the hardest-working components of the tray," Sun says. One of the tasks for the Abaqus analysis is double-checking to confirm that they are still strong enough after lightweighting. Another area of interest is the joint at which the front wall of the tray meets the truck bed floor.

Analyses are nonlinear and static, primarily concerned with the highest downward loads on the tray. The loads applied are based on field data such as service reports and specific customer feedback and requests, as well as the experience that DT HiLoad has gained from having over 1,000 truck bodies in use around the world. The largest vertical load simulated recently is 360,000 kg (about 397 tons) under 1-2Gs. "We compared the 1G results to those from a physical strain gauge test performed on the same tray," Sun says. "We found very satisfactory correlation, ensuring that the simulation process is accurate."

Their analyses enable engineers at DT HiLoad to prove out truck tray flexibility, check for points of rigidity, and calculate fatigue loads for points that need reinforcement.

ROLLING OUT RESULTS

The Hercules truck tray has now evolved to the Phase X body. Using Abaqus, DT HiLoad has been able to reduce truck tray weight up to an astonishing 50 percent. The new designs cut fuel costs, lower fatigue and loads on the tray and, most importantly, increase the payload the trucks can carry.

"Increased payload is the major benefit our customers value," Sun says. "If they can move the same amount of material with fewer trucks, their savings in both outlay and energy costs are considerable. Under these circumstances, ROI on a truck is arrived at faster and can be significantly increased."

For More Information
www.dthiload.com

SIMULIA CO-SIMULATION ENGINE PROVIDES NEW INSIGHTS INTO THE MULTIPHYSICS OF WINDPOWER

Abaqus structural analysis couples with fluid and control software to model complex signal-field interaction for many industries

Unlike a tree, a towering wind turbine should only bend slightly with the wind to avoid any collisions between its components. Its blades—now as long as 100 meters in some commercial designs—must withstand, yet also change pitch in response to the onslaught of variable wind speeds, transforming rotational velocity into electrical power.

If the wind gets too strong the turbine must be stopped as quickly as possible without damaging its expensive components. Understanding the multiphysics that result when blades, tower, generator, gearbox and wind interact is an exercise in extreme complexity for windpower engineers. But it's also a critically important exercise in terms of helping the wind industry optimize turbine performance to reduce the overall cost of this increasingly prevalent source of clean, alternative energy.

"Modeling flow conditions for wind turbines is especially challenging," says Stefan Sicklinger, whose Ph.D. thesis [https://mediatum.ub.tum.de/download/1223319/1223319.pdf], written at the Technical University of Munich, Germany, explores the topic in great depth. "Scalable multiphysics simulation tools are required in order to design and predict the performance, durability and safety of these machines. Such simulations are highly complex because they involve multiple disciplines, but it pays off for a product like a wind turbine for which testing is very expensive—or even not possible at all. The next generations of larger wind turbines are very nice candidates for this kind of modeling because you can't do any testing on them at the real scale."

Sicklinger's Ph.D. work focused on developing new methods and algorithms for co-simulation. In collaboration with Dassault Systèmes, he took on the problem of developing highly realistic models of wind turbine function by investigating new numerical methods. His thesis solved several industrial examples, ranging from a fully coupled fluid-structural-signal interaction with closed-loop control to a fully coupled emergency brake maneuver of a wind turbine with flexible blades. He presented much of his work at the 2014 SIMULIA Community Conference; this case study reflects updates from his recently finished thesis.

The simulation of the emergency brake maneuver of the NREL wind turbine involves the interaction of the generator/gearbox, flexible composite blades, control unit and the three-dimensional flow field. These recently developed co-simulation methods also have potential applications in a number of areas besides wind, Sicklinger is quick to point out.

"This type of field co-simulation can benefit many different industries," he says. "You could model the relationship between an automobile tire and its anti-lock braking system. You could investigate the active shape change of a wing—from an aircraft to a turbine blade—under different flow conditions. Or you could explore the ways in which a duct control unit shapes air inflow to reach a preset temperature. With co-simulation it is possible to analyze and optimize the interaction of control units (signals) with the technical product (fields & signals)."

ALWAYS START WITH GOOD DATA

To begin his wind turbine research, Sicklinger wanted to start from CAD models that were based on real-world data. "Clearly, an experiment needed to be the basis for my simulations in order to validate them fully," he says.

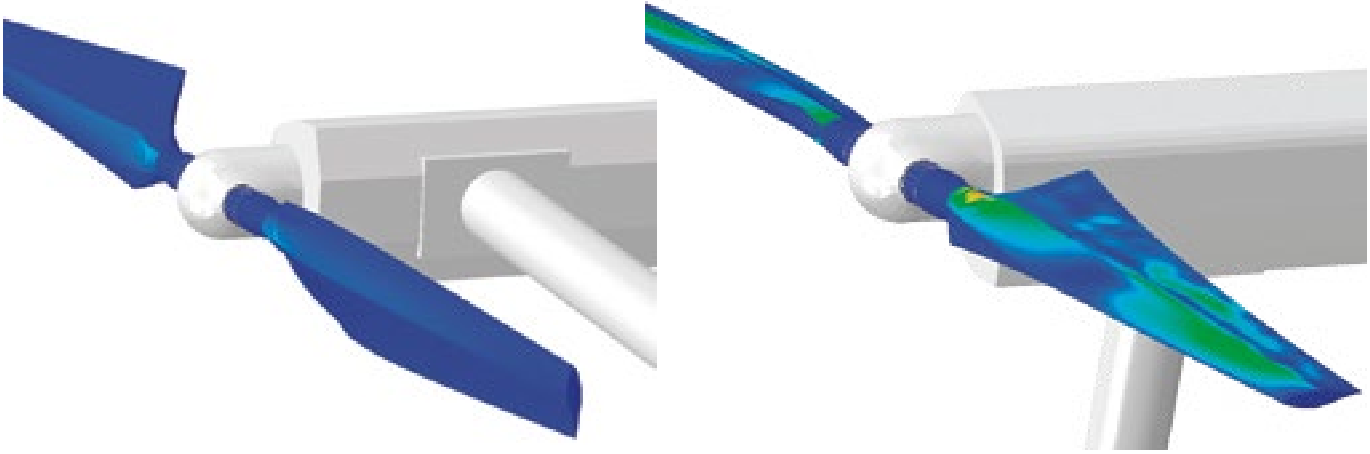


Figure 1. Stresses on outer composite layer of flexible wind turbine blades at different points in time during the emergency brake maneuver. Images courtesy Stefan Sicklinger.

Luckily, substantial measurement data resources were available from the NASA AMES' National Renewable Energy Laboratory (NREL) in Mountain View, California, where the world's largest wind tunnel is located. Commonly used for determining low- and medium-speed aerodynamic characteristics of full-scale aircraft and rotorcraft, a wind tunnel also provides the perfect setting for exploring full-scale, 3D aerodynamic behavior of wind turbines.

NREL, a facility of the U.S. Department of Energy's Office of Energy Efficiency and Renewable Energy, developed its windpower test objectives to meet recommendations of an international science panel of wind-turbine aerodynamics experts; the data is made available to researchers like Sicklinger to help improve and validate enhanced engineering models for designing and analyzing advanced wind-energy machines.

During the "Unsteady Aerodynamics Experiment Phase VI" at NASA AMES, a research wind turbine with lightweight carbon-fiber blades that measured 10 meters (33 feet) in diameter was set up in the 24-by-37-meter wind tunnel and operated at different angles during wind speeds up to 90 kilometers per hour generated by six huge fans. Probes integrated into the blade surfaces and other turbine structures recorded the pressure coefficients generated on the turbine at different wind speeds.

"The NREL data provided a major advantage for validation of my research because, unlike measurements taken in the field, a wind tunnel can deliver a constant inlet velocity profile," says Sicklinger.

LINKING IT ALL TOGETHER WITH THE SIMULIA CO-SIMULATION ENGINE

To solve the coupled physics challenge of linking four model types—CFD, structure, multibody dynamics and control—into a full-picture solution of an operating wind turbine, Sicklinger employed the SIMULIA Co-simulation Engine (CSE) and a research tool called EMPIRE. This allowed him to explore in great detail how the components of the structure respond to the fluid (wind), are influenced by each other's presence and react to feedback (from the control unit) by adjusting blade pitch.

Due to the research nature of his work, Sicklinger used OpenFOAM, an open-source finite-volume based solver, to analyze the three-dimensional turbulent flow field of air around the blades and rotor. The flexible composite blades (modeled with Abaqus/Standard) and the generator/gearbox (modeled with an in-house code) were connected to the CFD solver through SIMULIA's CSE. To extend the co-simulation to a fluid-structure-signal interaction, MATLAB was added to model the pitch control unit, which varies the angle of the blades in response to the strength of wind flow (Figure 1).

To Sicklinger's knowledge, his work represents a "first" in terms of combining four physics in a wind turbine simulation with such a high level of model fidelity. "We've now got a fluid-structure simulation where the turbine startup procedure is truly physically accurate," he says. His analyses depict the realistic behavior of a turbine startup, run, and emergency stop procedure employed when winds exceed the generator's capabilities.

"You see the turbine going up to speed, pitching its wings very slowly, because it's rotating," he says. "Then once the simulation is running at full speed you can investigate phenomena like local flutter of the flexible blades during unusual events like emergency brake maneuvers. You really need a high-fidelity model like this for such situations."

MODELING THE EMERGENCY BRAKE MANEUVER

So what exactly happens when a wind turbine has to execute an emergency stop maneuver?

"You can't really brake a big turbine the same way you would a car," says Sicklinger. "There is so much energy being generated that in a high wind a sudden stop could completely melt down the whole gearbox. There have also been situations where turbines have overheated and caught fire when they spin too fast in high winds."

In a properly designed wind turbine, when the wind exceeds a preset velocity, sensors provide feedback to the control unit, which automatically slows the rotation down by pitching the wings 90 degrees to reach 100% stall. The blades can no longer extract any energy from the wind flow and slowly come to a standstill. Sicklinger's finished models predict this entire cycle with a high degree of realism [http://youtu.be/vDDsAljF0ug].

Sicklinger ran his massive simulations on a 184 Intel Sandy-Bridge supercomputer; there were approximately 62 million unknowns solved per time step (10,000 steps in total). "It was important that the analyses could be carried out in parallel, because mine was a fairly large model, around one million degrees of freedom on the structure," he says. The analysis was performed with a new co-simulation algorithm that was developed within a research project between SIMULIA R&D and Stefan Sicklinger [http://dx.doi.org/10.1002/nme.4637].

OPENNESS AND ROBUST SOLUTIONS FROM SIMULIA

"My work concentrated on those features that directly influenced the fully coupled simulation," he says. "We found that the CSE is truly as open as they say: it's easy to use and it adapts to a lot of different hardware and software environments, allowing us to develop interfaces to these different simulation tools very easily."

"Abaqus/Standard also proved to be a very robust solver, extremely capable with geometric nonlinearities. Handling the large rotation of a wind turbine was no problem from the point of view of accuracy."

For More Information

www.tum.de

INCORPORATE STRUCTURAL OPTIMIZATION WITH TOSCA STRUCTURE IN ISIGHT WORKFLOWS

Tosca Structure is the structural optimization solution within the SIMULIA Portfolio offering topology, sizing, shape, and bead optimization capabilities. The simulation-based optimization process is completely managed by Tosca Structure itself. Within an optimization the chosen FEA solver is automatically executed obtaining the necessary FEA results to find the best design possible.

Isight as SIMULIA's process integration and design optimization (PIDO) tool offers the automation of complex engineering workflows.

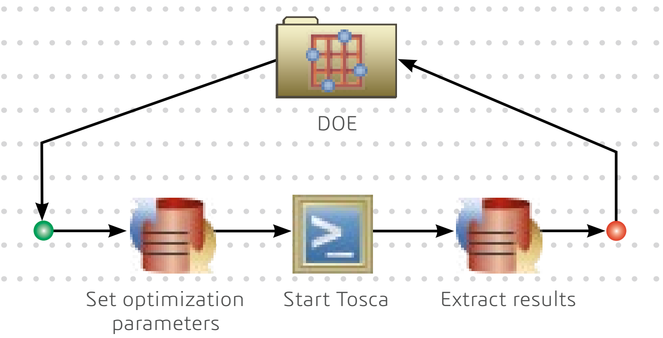
In specific cases you might want to expand workflows with non-parametric structural optimization capabilities and run Tosca Structure optimizations within Isight.

Possible applications are for example:

- Perform multiple structural optimizations with modified FEA input file parameters (e.g. material values, boundary condition, load scenario values)
- Perform multiple structural optimizations with modified optimization parameters or target values (e.g. amount of used material, displacement constraint values, demold control parameters)
- More advanced: Use Isight to modify the geometry (e.g. with Abaqus/CAE) of the topology optimization design space and then run the optimization

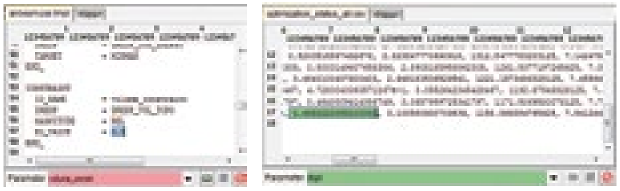
HOW TO SET UP ISIGHT TO RUN TOSCA STRUCTURE WITHIN AN ISIGHT WORKFLOW

As an example a simple Isight workflow is created to focus on Tosca Structure details. At first an optimization task parameter is modified, then Tosca Structure is started, and afterwards specific values are extracted to judge the performance of the obtained Tosca optimized design.



Here's What You Do:

1. Start with a default model in Isight Design Gateway. Drag and drop into the workflow an 'OS component' preceded and followed by 'Data Exchanger' components. Alternatively you could use one 'Simcode' component.
2. Configure the first Data Exchanger component to parse the Tosca optimization parameter file (*.par*) Define the input parameters to be varied and the substitution rules.
3. Configure the second Data Exchanger component to parse the output file ***optimization_status_all.csv*** which is located in the runtime Tosca subdirectory named as the Tosca parameter file. Define the output parameters that are of interest for your study and the extraction rules. Remember that you should have defined corresponding design responses in your Tosca parameter file to insure that that their values will be written in the csv file.



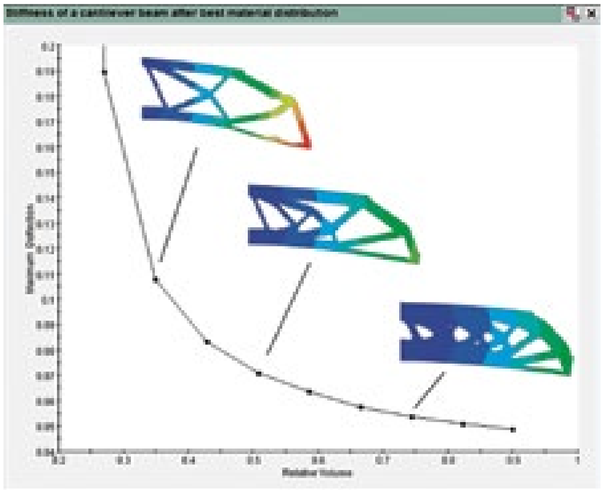
4. In the OS component Basic tab set the command to launch Tosca Structure optimizations:

   <tosca_install_dir>\bin\tosca.cmd -j <par_file>

   In the Required Files tab make sure to list both the Tosca parameter file and the input file containing your initial FE model as well as any other files that might be required for the Tosca optimization.
5. Feed the Isight workflow with an appropriate process driver (e.g. DOE or Optimization).

POSSIBLE APPLICATION

The above described workflow can be used, for example, to investigate how the spent material influences the stiffness of the design. Isight manages the amount of available material while the topology optimization with Tosca Structure efficiently locates the material to provide the stiffest structure possible.



WELCOME TO DASSAULT SYSTEMES NEW PROVIDENCE CAMPUS!

The building was designed in the spirit of collaboration, a hallmark of our work environment, and includes such features as a Customer Visit and Training Center, and state-of-the-art meeting and collaboration rooms. In addition to these features, we are proud that our building is seeking Gold LEED® Certification, meeting Dassault Systèmes' high standards for energy efficiency.

We welcome you to our new home and look forward to seeing you in Rhode Island!

For more information, visit www.3ds.com/simulia

DASSAULT SYSTEMES | The **3D**EXPERIENCE® Company