SIMULIA

Realistic Simulation News

May/June 2011
www.simulia.com

Javad Fatemi, Dutch Space

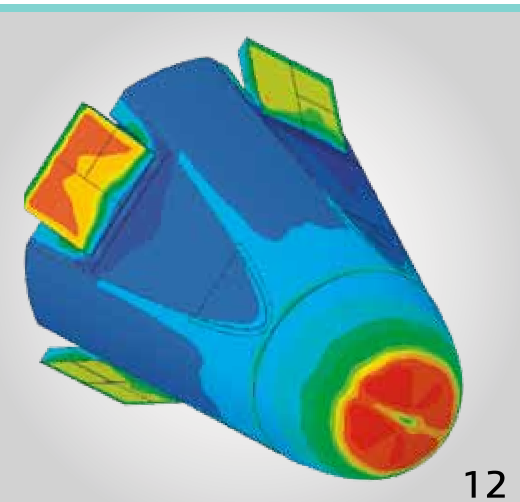
Dutch Space Uses Abaqus to Virtually Test a Thermal Protection System

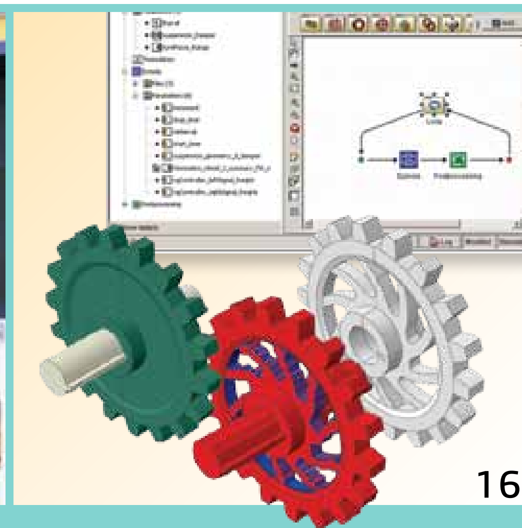
SIMULIA

SIMULIA

Realistic Simulation News

May/June 2011





Inside This Issue

In Each Issue

SIMULIA Realistic Simulation 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.simulia.com

Editor
Tim Webb

Associate Editor
Karen Curtis

Contributors
Javad Fatemi (Dutch Space), Pasquale Carlucci and Christopher Mougeotte (U.S. Army—ARDEC, Picatinny Arsenal), Laurence Young, Andrew Vechart and Rahul Goel (MIT, Man Vehicle Laboratory), Inés Lama (Applus IDIADA), David Hodgson (Rolls-Royce), Ivica Smojver (University of Zagreb), Silvestre Pinho (Imperial College of London), Parker Group, Rachel Callery, Matt Ladzinski, Jill DaPonte, and Jonathan Arata (SIMULIA)

Graphic Designer
Todd Sabelli

MAY_SIM_411_VOL 1

3DS SIMULIA

Advancing Simulation Technology and Methods—Together

Roger Keene
Vice President, World-Wide Operations

It occurred to me, while listening to the General Lectures at our 2011 international SIMULIA Customer Conference in Barcelona, just how far we have come in the development of new and innovative analysis technology in Abaqus. A slew of advances such as Topology Optimzation, CFD, Electromagnetics, Smoothed Particle Hydrodynamics, GPU support, XFEM and so much more (page 17) have been delivered even as we have expanded our entire solution portfolio to include Isight, the SIMULIA Execution Engine, and Simulation Lifecycle Management (page 16). Most intriguing though was the first glimpse of ExSight, our next generation analysis products built on the Dassault Systemes Version6 platform, which will offer unprecedented integration of Abaqus and Isight technology in an intuitive, collaborative and managed environment.

The highlight of the 2011 SCC though was the 95 customer papers illustrating the breadth of the application of our analysis, process automation and optimization technology to explore and improve the performance of materials and products. The first keynote speaker, J. (Siva) Sivakumar, the Engineering Group Manager for General Motors Powertrain, set the stage for the conference by pointing out that—although CAE vendors have made vast improvements in simulation software over the years—it is ultimately the users that are at the heart of driving realistic simulation technology forward and making it an invaluable tool in the product development process. Anita Bestelmeyer, CAE Manager for BD, opened the conference on day two. She illustrated the positive impact that a small group of dedicated users can have, not only on BD's success, but on improving the experience of the many patients that rely on BD's medical products.

This passion for exploring the boundaries of analysis technology can be seen in this issue of Realistic Simulation News. From pushing the limits in aerospace applications at Dutch Aerospace (page 12) to improving the protection of soldiers in the U.S. Army (page 4 and page 6), it is evident that the use of realistic simulation is a critical tool in accelerating the improvement of product performance in the Aerospace and Defense sectors.

As we continue to strive to bring new technology and methods to the industry, your participation at our international conference and Regional Users' Meetings (page 23) is more valuable than ever. These gatherings are not only your opportunity to share your requirements with us, but also to learn from your peers on how they are leveraging realistic simulation technology to their competitive advantage. I encourage you to attend the Regional Users' Meetings held around the world later this year where you can learn more about how to apply our current products more effectively and find out more about the future of simulation as we demonstrate and explain more about the ExSight products. I look forward to seeing many of you there!

Sincerely,

Download select 2011 SIMULIA Customer Conference papers that provide details on how our customers are applying realistic simulation in the Aerospace and Defense Industry.

Airbus Military—*Simulation of Residual Stresses due to Cold Expansion Bushing Installations: Application to Fatigue Life Evaluation*

EADS—*Bird Strike Simulations on Composite Aircraft Structures*

Fokker Landing Gear B.V.—*Virtual Testing of Composites Using Abaqus*

U.S. Army - ARDEC—*Analysis and Redesign of Re-sealable Ammunition Boxes Using Abaqus/Explicit CEL*

U.S. Army - ARDEC—*Modeling of a Tactical Spike Strip for Use at Checkpoints*

U.S. Army - ARDEC—*Using Co-simulation to Extend the Usage of XFEM*

For More Information
www.simulia.com/cust_ref

Making Army Helmets **Tougher and Safer**

MIT Man-Vehicle Laboratory uses Abaqus to explore and improve helmet liner design to prevent traumatic brain injury caused by blast events

Figure 1. The U.S. Army's Advanced Combat Helmet (ACH). (Image courtesy U.S. Army)

Simulating a blast event provides important, realistic data without the risk of involving test subjects

Andrew Vechart
Researcher, MIT, Man-Vehicle Laboratory

The U.S. Army helmet is an iconic piece of military equipment. The storied steel M1, first introduced in 1942 during WWII, served not only as head protection but as a seat, wash-basin, and soup bowl. It was standard issue to U.S. soldiers for the next four decades. As materials, ergonomic design, and ballistics protection evolved, the M1 was finally replaced by the 29-layer Kevlar PASGT (Personnel Armor System for Ground Troops) helmet in 1985, which in turn gave way to the lighter Kevlar/Twaron ACH (Advanced Combat Helmet) design in 2003 (Figure 1). Helmet liners progressed too, from compressed paper fibers, plastic, and rayon in the early days to more sophisticated suspension-webbing systems with chin straps constructed from stronger synthetics such as nylon.

Preventing head injury has become even more critical today with the development of advanced body armor, which is used routinely by most U.S. troops: Body armor has decreased the number of fatalities from explosions, but as a result survivors are experiencing an increase in non-fatal traumatic brain injuries (TBI). According to the Defense and Veterans Brain Injury Center (DVBIC), established in 1992, more than 150,000 U.S. military personnel have been medically diagnosed with TBI since 2001. Some experts believe that at least 30 percent of all troops who have spent four months in combat in Iraq or Afghanistan have been exposed to potential brain-injuring explosions.

TBI can be caused by land mines, mortar rounds, rocket-propelled grenades, suicide bombs, and most frequently, improvised explosive devices (IEDs), all of which generate a shock wave that travels 1,600 feet per second. The vast majority of brain injuries—ranging in severity from concussions to penetration injuries—are classified as mild to moderate (89 percent) and come with accompanying physical, cognitive, emotional, and behavioral symptoms. In past conflicts, TBI was labeled shell shock and battle fatigue and has even been linked to post-traumatic stress disorder (PTSD). Today, TBI is the signature wound of soldiers returning from combat.

To address this reality, the military has launched a program to develop a liner for the ACH that will reduce the frequency and severity of these debilitating injuries.

The challenges of helmet liner design

Since the 1970s, there has been no shortage of ideas about how to construct helmet liner systems. Countless patents and designs have emerged using air- and fluid-filled chambers. But the designs have primarily been sports related—for bike, ski, hockey, football, and horseback riding—and the protection systems focused on protecting against impact (striking an object) rather than blast (from a shock wave).

There's also been no shortage of hypotheses about what materials would be most effective at blast attenuation. Mechanical properties are the major contributor to the behavior of shock waves at material interfaces: acoustic impedance mismatches determine what portion of a wave is reflected and transmitted. Layered composites, cellular materials, expanded polystyrene, vinyl nitrate foams, and glycerin have all been suggested as candidates, while material properties such as porosity, density, and heat capacity have been proposed as factors contributing to blast mitigation.

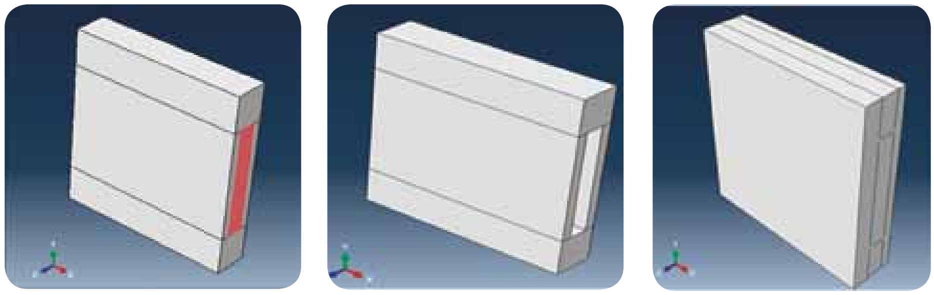


Figure 2. Simulation of a simplified flat-plate model tested the effectiveness of both solid and liquid filler materials for the helmet liner, with the blast force coming from the left in the z-direction. The solid filler materials (foam, glass beads, and aerogel) were modeled in red (left). A Coupled Eulerian-Lagrangian (CEL) analysis was used to simulate the fluid filler materials (water and glycerin): the Lagrangian frame of reference for the solid portion of the model (center); the Eulerian frame of reference for the fluid filler portion (right). Solutions were computed to two milliseconds after detonation when transmitted pressure values had stabilized.

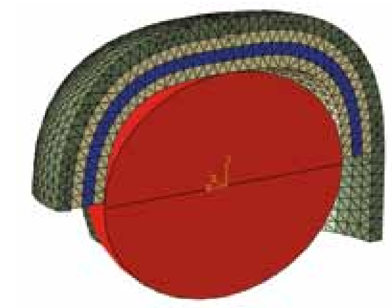


Figure 3. The helmet and head models are simplified to decrease the computational complexity of benchmarking helmet-liner filler materials.

Simulating liner materials and helmet-head contact during blasts

Dr. Laurence Young's (http://web.mit.edu/aeroastro/www/people/lry/) research on helmet and liner systems at MIT in the Man-Vehicle Laboratory started out, like most, with sporting applications. But the focus shifted in 2007 when his lab was awarded a three-year contract from the Office for Naval Research (ONR) to work on improvements for the ACH liner design. Finite Element Analysis (FEA) technology was identified as the primary tool for evaluating potential design solutions.

The MIT team was comprised primarily of graduate students—Andrew Vechart and Rahul Goel from late 2009 to early 2011—who would temporarily participate in the research before graduating or moving on to other projects. For this reason, according to Vechart, "We needed an FEA solution that had a short learning curve and was well-supported and documented." There were several other important requirements as well. "The software needed to handle the nonlinear complexity of the contact between the helmet, head, and air," says Vechart. "And it also had to be good at simulating the physics of the blast wave moving through air." Other design challenges included comfort, fit, and feel. The team chose Abaqus for its ability to meet these requirements, as well as the fact that it is frequently referenced in research literature for its blast modeling and analysis capabilities.

"Simulating a blast event provides important, realistic data without the risk of involving test subjects," says Vechart. "It also eliminates the need for special facilities and permissions required to handle explosives."

In order to evaluate the effectiveness of different filler materials during a blast event, the team decided to first analyze a simplified liner model—a flat sandwich-plate manufactured from high-energy-absorbing vinyl nitrile foam. A cavity in the plate was used to hold the different fillers including: fluids—water and glycerin; and solids—foam, glass beads, and aerogel (Figure 2). Results from these tests were compared against the benchmark case of a solid piece of foam with no cavity. The team used the CONWEP air blast capability in Abaqus to reduce computation times by eliminating consideration of the blast media (air) from the analysis. They then used the Coupled Eulerian-Lagrangian (CEL) feature in Abaqus to analyze the realistic behavior of fluid filler materials.

Next, the team created a simplified helmet-liner model and coupled it with a surrogate head model, also simplified (Figure 3). The team again used a CEL simulation, which effectively replicated the relatively complex fluid-structure interaction (FSI) of the air blast—high blast levels, short time spans, compressibility effects, and some nonlinearity.

For both analyses, Abaqus was used for the entire process: from modeling the geometry, to running the nonlinear and multiphysics simulation, to post-processing the results. The team used a Windows XP, 64 bit, 8 GB RAM system running on an Intel Core 2 Duo 3 GHz processor. Solutions were computed up to two milliseconds after detonation with peak transmitted pressure being of primary interest.

Benchmarking results and future simulations

To validate simulation results against physical experiments, collaborators at Purdue University's Zucrow Laboratory used a shock tube to create a controlled explosion equivalent to the one used in the simulation and an approximation of a typical IED explosion (50 pounds of TNT at a distance of 20 feet). The simulated blast had peak overpressures strong enough to cause TBI (50 psi) but not strong enough to be fatal (at 100 psi there is a one percent chance of fatality).

According to Young, the project was recursive, going from experiment to model to experiment, with each being refined based on input from the other. "The FEA model is an effective tool to plan and critique each series of experiments," says Young.

For the solid foam liner, there was reasonable agreement between simulation and test results for peak loading pressures and rise time as the shock wave impacted the liner, as well as for transmitted pressure on the back-side of the model. Comparing transmitted pressures for a variety of solid and fluid filler materials, the analyses indicated that glass beads, water, and glycerin had the lowest peak pressures—with glass beads having a value 80 percent less than solid foam. These results (in partial agreement with the experiments) also indicated that the rise time and pressure gradient for glass beads, glycerin, and water demonstrated the best characteristics for blast attenuation.

"For validation we used a relatively simple model," says Vechart. "In the future, we would like to create numerical models and use actual helmet and liner-channel geometries coupled with a realistic head model based on CT or MRI scan data." By using simulation, Vechart believes the engineering community will not only come up with a better liner and a more protective helmet, but the medical community will develop a deeper understanding of how to diagnose and treat traumatic brain injuries.

For More Information
www.mvl.mit.edu
www.simulia.com/cust_ref

Helping Keep Soldiers Safer with Realistic Simulation

Picatinny Arsenal uses Abaqus to analyze and improve strength of armor subjected to blast waves

The Objective Gunner Protection Kit mounted atop a combat vehicle.

When we talk of an Army and a mission, we usually think of soldiers in the field—patrolling through a combat zone in an armored vehicle, for instance. At the U.S. Armament Research, Development and Engineering Center (ARDEC) at Picatinny Arsenal in New Jersey, one critical mission behind the field operations is every bit as important: the analysis and testing of protective vehicle armor so that those soldiers return to base safely.

ARDEC's tradition of armament research at Picatinny stretches back a century. Due to the Center's expertise and vast amounts of accumulated physical data, it is occasionally tasked with proving out armor designs.

One such project was the structural assessment of an overhead cover add-on for the Objective Gunner Protection Kit (OGPK) used on the High-Mobility Multipurpose Wheeled Vehicle, or HMMWV. The OGPK, which received an Army Greatest Invention Award for 2007, is an integrated armor and ballistic glass cupola shield mounted on top of tactical and armored vehicles. It provides 360-degree protection from small arms fire and explosions while retaining visibility for the gunners. The mission for ARDEC was to ensure that the overhead cover provided effective protection during exposure to blast loading.

One of the best tools in their arsenal to accomplish this mission was finite element analysis (FEA) software, which they were already using extensively—but this time, engineers at Picatinny had a vision of how they could greatly increase the accuracy of their simulations and account for the effects of the shock interactions, including oblique blast waves on the OPGK due to reflections from the top of the HMMWV.

Achieving simulation accuracy

In the past, engineers at ARDEC had developed a standard process for structural analyses that involved using simplified blast parameters and manually applying pressure loads to a 3D model in Abaqus/Explicit to simulate a particular blast loading.

After weighing various simulation methods, the engineers at Picatinny decided to explore something new: conducting the blast load analysis using the fully Coupled Eulerian-Lagrangian (CEL) capability within Abaqus. The CEL capability enables the user to simulate a fluid or gas (the Eulerian domain) interacting with a structure (the Lagrangian domain). Here, the structure to undergo blast loading is surrounded by a volume of air. The blast wave that would exist at the inlet of the air domain is created through boundary conditions. The blast wave then propagates through the Eulerian domain and subsequently interacts with the Lagrangian structure located some distance from the inlet.

Pre-testing the analysis method

Before attempting a full-scale blast load analysis, the engineers at Picatinny validated the performance of the CEL method in Abaqus. They did this by verifying that the software could realistically model the reduction in strength of the shock wave as it propagated through the air (Eulerian domain) and that it could simulate normal and oblique reflections against the Lagrangian model accurately.

Simulating normal reflections: This was accomplished with a simple FEA one-dimensional (1D) domain CEL model of a "shock tube," much like the physical shock tubes used to study the behavior of gases under shock loading. A shock tube was used for the validation effort because the relevant analytical equations for the compressible flow are readily available. The shock tube consists of two chambers, one initially filled with gas under high pressure. The other was filled with room-temperature air at atmospheric pressure. The analysis is initiated in the state that would exist immediately after the burst diaphragm separating the two chambers ruptured. The expansion of the high-pressure gas into the low-pressure chamber creates a planar shock wave that propagates through the low-pressure chamber. Once the shock wave reaches the boundary at the end of the low-pressure chamber, a reflected wave is created. It is the pressure behind this normally reflected pressure wave that is compared against the theory to determine if the underlying code behind CEL is handling the compressible flow properly. Excellent correlation was observed between the shock tube model and the theory.

Spherical expansion: This validation test included a CEL domain representative of a 1D cut from a sphere (to account for spherical expansion). The Eulerian domain was assigned an air material at ambient temperature and pressure. At the narrow end (sphere center) of the domain, an explosive load is initiated and allowed to propagate through the domain. The load was defined using a velocity boundary condition with a triangular amplitude curve and a peak value equivalent to the initial particle velocity of the blast wave. As predicted by physical data, the shock wave decayed exponentially in strength, lengthened in duration and slowed down as distance increased from the source of the blast.

This simulation was also used for a mesh refinement study that was valuable for the final analysis model. When working with shock waves, the size of the mesh is quite important: a very fine mesh is needed to maximize the accuracy of the solution, though this increases the run time.

Simulating oblique reflections: From comparing the pressure decay of the 1D CEL model to data obtained empirically from exploding TNT, it was clear that accurately modeling the spherically expanding blast was an important factor in obtaining accurate results. In order to extend their methodology to a realistic problem it was therefore necessary to expand the validation of their modeling approach from 1D to 3D.

To accomplish this, the analysts modeled a 3D segment of a sphere for the Eulerian air domain with a Lagrangian modeled plate centered in it, tilted at a 45-degree angle. The blast wave used the same velocity boundary condition as the 1D CEL model and originated again from the inlet of the domain. This time, though, there were two results of analysis: peak reflected pressure at the surface of the plate, and incident overpressure in the air at the same distance from the inlet. The ratio of these pressures was compared to an empirical plot of reflected overpressure as a function of angle of incidence, and there was good correlation.

Now the engineers were ready to model and run a more detailed 3D CEL analysis.

CEL analysis

Structural parts of the analysis (the OGKP and a rigid part that represented the shape and angle of the HMMWV roof) were modeled with Lagrangian components. Most of the armor panels and brackets of the OGKP were meshed using SC8R 8-node continuum shell elements; the remainder of the brackets and the windows were meshed with C3D8R 8-node brick elements. Connector elements were used for all of the bolted joints to ensure that the structure was constrained properly and to enable monitoring of all the bolt forces. The mounting brackets of the HMMWV roof were modeled as elastic with linear strain hardening, while the armor panels were modeled using Johnson-Cook materials to capture plasticity and damage. For the baseline analysis, general contact was used to define the contact interactions between all of the armor panels and brackets.

The Objective Gunner Protection Kit shown with an overhead cover add-on.

The Eulerian domain represented the air around the structure that was the medium for the blast wave. The domain was modeled as a section of a sphere. To speed up the analysis while maintaining accuracy, a biased mesh was used in the Eulerian domain. In the region of interest near the Lagrangian structure the mesh consisted of 0.25” thick elements along the direction of initial blast wave propagation and coarser 1” thick elements everywhere else. This technique allowed for a reduced number of elements, even though the total was still relatively high at 2.6 million just for the Eulerian domain. The blast wave was defined using the same boundary conditions as described for the earlier simulations.

Realistic simulation guides future work

The simulated deflections on the armor panels were compared to previously conducted analyses that used simplified pressure loads on the surfaces exposed to the blast. Overall the comparison was favorable, with the CEL analysis providing more realistic results than previous analyses did.

Picatinny anticipates that CEL could be valuable in the design phase of new armor systems, since it provides an understanding of how a given armor system model responds to blast loading. This same analysis could be applied to any structure that might experience blast loading—for instance, an explosive test facility or buildings in a high-risk (combat) area.

One of the advantages of using the CEL approach is that Abaqus can execute all the shock interactions automatically, so that the analyst isn’t required to calculate the angle of incidence for each surface interaction in order to find the correct reflected pressures on each oblique surface. (A new alternative to the manual technique can be found in SIMULIA’s recent addition of ConWep to Abaqus, which can automatically calculate the correct distance and angles incidence in a blast model and assign the appropriate pressures.) Another important feature of CEL is that the Lagrangian structure can be easily reoriented within the Eulerian domain to analyze any angle of incidence that is required.

The ARDEC engineers concluded that the CEL approach for modeling blast loading shows great promise in its ability to provide valuable insight and realistic results. It also enables the analysis of very complex geometries that were previously impossible to solve accurately with more simplified methods. ARDEC intends to continue exploring Abaqus and CEL to thoroughly validate the new technique so that it may eventually be used to conduct predictive analyses.

For More Information
www.pica.army.mil
www.simulia.com/cust_ref

Aerospace & Defense Systems Require a "Re-boot"

Jonathan Arata, Industry Lead – Defense and Shipbuilding, SIMULIA Technical Marketing

Images courtesy of the U.S. Department of Defense

The rapid "operational tempo" of modern defense contingencies requires that products be delivered to the front-lines at speeds that would have been unthinkable only ten years ago. Innovators must adapt their design simulation strategies and engineering toolsets to meet the increased demands of time, quality, safety and cost.

The Defense segment of the Aerospace and Defense industry includes a wide range of products, large and small—from support vehicles, submarines, aircraft carriers and unmanned aerial vehicles to armored vests, helmets, artillery, and munitions. Many of these are highly complex "systems-of-systems" that must operate reliably in extremely demanding environments (sand, dust, salt water, extreme temperatures, etc.). They are also in development and manufacturing cycles for several years and have lifecycles that can be measured in decades. But the challenges of the modern battlespace, particularly in the age of unconventional warfare, also mean that "on-the-fly" modifications to existing systems and rapid development of new systems are becoming ever more commonplace.

In all new designs and product modifications, primary concerns for safety, survivability, reliability, and durability must be balanced against weight, efficiency, environmental impacts, and cost. In the past, most design and development in Defense has been accomplished by "make-and-break" approaches, whereby a large number of prototypes are built and tested physically. This is simply not at all practical for addressing the safety challenges and time constraints of the modern battlefield. The traditional lengthy and costly approach clearly demonstrates the need to evolve design and development strategies toward more aggressive use of "virtual testing"—meaning the replacement of physical testing, to the extent possible and practical, with realistic simulation. Therefore, the effective employment of robust and innovative realistic simulation methods and technology is absolutely essential to the success of any design/development/evaluation process for defense products and systems.

Technology and methods evolution

The simulation tasks necessary to design these products include many disciplines and domains, and the Defense industry was one of the earliest and most enthusiastic adopters of simulation-based-design strategies. In fact, many of the first Finite Element Analysis (FEA) codes are products of Government and Defense laboratories around the world. Partnering with forward-thinking universities, these laboratories created, what were then, state-of-the-art software tools that generally focused on specific design analysis workflows (such as underwater acoustics or nonlinear static shell analysis). These developments took place at a time when computational resources were expensive and offered limited capabilities. Today, many of these codes are still in heavy usage and in some state of active development. While major advances in computational power can help overcome some of the serious limitations associated with these codes, faster computers alone cannot compensate fully for the outdated simulation workflow processes, inability to manage data, and general user unfriendliness of most of these tools.

Clearly, modern simulation challenges such as those presented in this issue of *Realistic Simulation News* by Picatinny Arsenal (page 6) and MIT (page 4) as well as in papers by General Dynamics-Electric Boat (SCC 2010) and Beretta (SCC 2007) must be addressed by modern, user-friendly, robust, proven and supported simulation tools. Such tools, available as commercial-off-the-shelf (COTS) today, provide functionality and ease of use that is simply not available in Government and Defense developed software in spite of the decades of time and tens of millions of dollars invested in them. Indeed, the Rolls-Royce case study presented here on page 18, provides even more compelling evidence that it is far more cost- and time-effective for the Defense community to partner with capable commercial software vendors rather than develop and maintain these solutions themselves. In this regard, SIMULIA's scalable product suite for unified FEA, Multiphysics, Design Optimization, and Simulation Lifecycle Management is playing a critical role in helping Aerospace and Defense engineers solve this industry's most challenging design and engineering problems, rapidly and cost-effectively.

Unified FEA and multiphysics

SIMULIA's Abaqus Unified FEA platform offers best-in-class support for analysis problems both large and small. For workstation customers, our intuitive and easy-to-use Abaqus/CAE product enables you to develop geometry and meshes in a fraction of the time you might be used to. Our interfaces with popular CAD tools such as CATIA, SolidWorks, and Pro/E make those tasks even easier. The structural analyses capabilities in Abaqus enable you to leverage our advanced material models—including the capability for user-defined materials and elements—and general contact for both implicit and explicit dynamics. Abaqus also provides built-in multiphysics capabilities for computational fluid dynamics, new electromagnetic capabilities, and a new Abaqus/CAE Topology Optimization Module (ATOM) (see Abaqus 6.11 page 17). Our unified FEA and multiphysics solution provides an open platform that allows for coupling with Abaqus to existing analysis codes, whether commercially available or developed in-house.

The high-performance parallel solvers of Abaqus are optimized for cluster computing and provide excellent scalability for both implicit and explicit analyses on hundreds of processors, offering industrial and laboratory users the opportunity to solve large and complex problems efficiently and effectively. Improved model generation through new features in Abaqus/CAE, and accelerated solution times from our optimized and efficient parallel solvers, allow designers and analysts the chance to shorten their product development cycles and expand the design parameter spaces they can explore, which results in speeding optimized products to the battlespace in times that are simply not conceivable using legacy approaches.

Design exploration and optimization

Isight is used by defense and shipbuilding companies to connect a variety of applications, automate the execution of multiple simulations, and perform multidisciplinary design exploration and optimization. In a paper presented by General Dynamics–Electric Boat at the 2010 SIMULIA Customer Conference, Isight and Abaqus were used in a simple workflow to optimize the geometric properties of a 212-type submarine structure for structural stability. That is just one example of how Isight is helping our customers perform complex design exploration and optimization studies in a robust, automated and user-friendly environment. Isight is a key enabler for selecting the best design parameters to meet engineering targets, improve efficiencies, and reduce design cycles. (See Isight 5.5 page 16).

Many of the simulation process flows used in the virtual testing of defense and shipbuilding systems are robust, mature, and repeatable—some are even dictated by codes and standards. Isight makes it possible to capture the methods, deploy them to non-experts via a template-based interface, automate their execution, and share the results for collaborative decision-making.

Managing, securing simulation IP

Due to the complexity, field maintenance demands, and extended lifecycles of Aerospace and Defense products, it is critical that their related design analysis iterations be tracked, managed, and secured. These challenges become even more acute considering the highly diverse and distributed design and engineering workforce in the industry. The Simulation Lifecycle Management (SLM) solution from SIMULIA enables individuals, workgroups, and large enterprises to manage simulation processes, applications, data, and results. SIMULIA SLM also provides unique online collaboration capabilities that allow distributed engineering teams to share simulation methods, models, and results in order to make better-informed design decisions. These capabilities offer significant benefits to the Defense industry, where traceability of simulation results and their impact on design decisions are critical for accelerating product development and achieving customer acceptance. (See SLM V6R2012 page 16).

Customer-Focused Strategy

As our realistic simulation technology capabilities and product portfolio grow, it is vitally important that our solutions meet the demanding requirements of the Defense industry. We are closely engaged with our Aerospace, Defense, and research customers around the world to better understand their processes and simulation requirements in order to deliver the specific functionalities that enable them to solve engineering problems that lead to better performing and safer products. Customers in this industry can expect to see continued developments focused in the areas of advanced material modeling and multiphysics simulation, and high-performance computing, as well as improvements in ease-of-use. We are committed to working with progressive Defense contractors and laboratories to further define the role of Isight and SLM for their particular simulation process flows. In working with our customers, we are confident in our ability to deliver robust, realistic simulation solutions that enable designers and engineers to meet shortened time frames and tighter budgets, while delivering safer and more reliable products.

Jon Arata
Industry Lead, Defense and Shipbuilding, SIMULIA

Jon is responsible for developing and directing SIMULIA strategy for the Defense and Shipbuilding industries. He joined SIMULIA in July 2008 after a two-decade long career as an engineer, researcher, and business developer in the U.S. Defense industry: working for General Dynamics – Electric Boat, Foster-Miller (now QinetiQ North America) and as an onsite researcher at the U.S. Army Soldier Systems Center. He was awarded his Ph.D. in Solid Mechanics from Brown University in 2000, and has a Masters Degrees in Engineering and Mathematics.

For More Information
www.simulia.com/solutions/defense

What *is* that Noise in My Car?

Applus IDIADA uses Abaqus to help design-out vehicle squeaks and rattles

So you've finally bought your shiny, new car. But then one day the noise starts: an unidentifiable, repetitive, distracting sound coming from somewhere inside your vehicle. In no time at all, that annoying sound is driving you crazy.

The problem is called squeak and rattle (S&R) and it's been driving the automotive industry crazy, too. Paradoxically, while great progress has been made in other areas of noise and vibration (N&V), the fact that modern automobiles run more quietly than ever has made lingering S&R issues even more apparent. With engine and road noise diminished, smaller sounds that used to be hidden become magnified to the driver's ear. While specific S&R issues can be targeted, ongoing trends toward lighter cars and new materials continue to work against total elimination of the problem.

Squeaks and rattles are often located in the interior trim of a vehicle, such as the dashboard, but the exact source can be hard to pinpoint. Squeak happens when components periodically slip and stick together. Rattle occurs when parts hit each other intermittently. Both noises are usually due to inconsistent assembly tolerances or lack of stiffness. Some are more detectable at slower driving speeds, but others get worse as you accelerate.

When noticed during a test drive, S&R can be seen as a sign of poor quality and durability, putting-off potential customers. When S&R issues surface after purchase, they are difficult for car dealers to diagnose, expensive for them to fix, and the fixes may even lead to new S&R problems. Subsequent warranty claims can significantly impact vehicle manufacturers' reputations and profit margins.

A definitive paper on the subject, delivered to the Society of Automotive Engineers' Noise and Vibration Conference in 1999, exhaustively examined the problem, detailed existing methodologies, and concluded that more refined analysis methods were needed. Some years before, IDIADA was formed to support product development in the global automotive industry with design, engineering, testing and homologation services. Headquartered near its testing grounds in Barcelona, Spain, the global company now has branches throughout Europe, Asia and South America. In 2010, IDIADA won the "Automotive Testing Company of the Year" award from Automotive Testing Technology International magazine.

Car dashboard in test apparatus

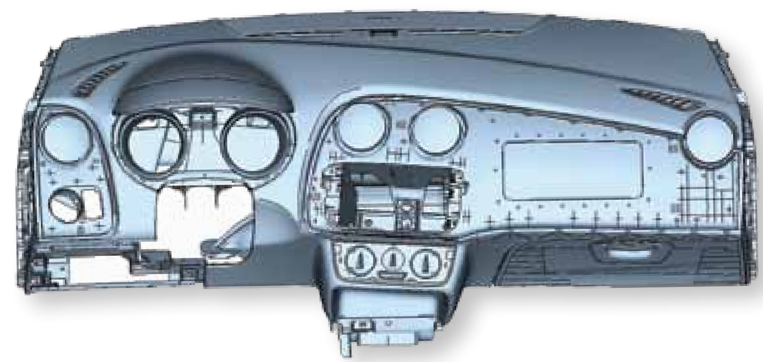


Figure 1. FE model of front of instrument panel used for the study, courtesy of SEAT.

Realistic simulation addresses S&R concerns

"Squeak and rattle have become of greater concern for us in recent years as more auto manufacturers come to us with these problems," says Inés Lama, project manager, design engineering, for IDIADA. "Our customers are asking us if it's possible to use simulation to identify the potential for squeak and rattle earlier in their design processes, rather than later when it is more costly and time consuming to solve."

As a mechanical engineer with a university concentration in vibration and noise, plus over a decade working with similar multiphysics issues at IDIADA, Lama is well-versed in using realistic simulation to visualize and predict many of the complex material responses that arise when motor vehicles meet the open road. She and her team have been using Abaqus Unified Finite Element Analysis (FEA) for years. "Since we'd already been using Abaqus in vehicle cockpit design and testing for thermal, impact and normal modal analyses, it made a lot of sense to simply develop a new load case for squeak and rattle inside Abaqus," she says.

Starting in 2008, the IDIADA team began developing an S&R-specific simulation protocol based on Abaqus. A paper delivered at the 2011 SIMULIA Customer Conference in Barcelona this May presents the latest improvements in this methodology, applied to rattle in a car instrument panel and correlated with real-world testing. The instrument panel (the physical and the virtual ones) were donated to IDIADA by Spanish car manufacturer SEAT (Figure 1) and used in the correlation process for validating the methodology. The FE model had been used in the normal development process of the component (behavior in crash, static and dynamic stiffness analysis and thermal analysis). But the test layout was designed specifically to provoke the rattle in the cockpit, because this phenomenon didn't appear in normal usage conditions.

Approximating a nonlinear rattle response inside a linear analysis

Rattle arises when parts collide and the relative movement between them generates noise if the surfaces adjacent to where the impact occurs radiate audible sound. Developing a new load case to simulate such an event required some creative thinking, the IDIADA team learned.

A classic automotive N&V analysis (of the effects of a car engine running, or tires rolling) uses modal theory to predict at what frequencies certain parts of the vehicle will begin to vibrate. "Modal theory is based on the hypothesis of linearity, without contact," says Lama. "But an S&R event, although it happens within a frequency-dependent, N&V-type setting, is also very nonlinear—the parts producing the squeaks and rattles are interacting with each other in three dimensions. A standard, eigenmode-based, N&V method alone can't model, or predict, the contact that will result in a rattle."

The engineers needed to come up with a simulation that would accommodate both frequency (N&V) and contact (S&R) behavior. "The unique connector element in Abaqus was particularly useful for us with this challenge," said Lama. Pau Cruz, an advanced Abaqus user of the CAE team, along with colleagues Jordi Viñas and Andreas Rousounelos, realized that the connector could be used as a "virtual sensor" for the detection of contact. Placing connector elements in the gaps between nodes in the FEA model of the instrument panel allowed for the measurement of the independent behavior of the model in three different directions. (To properly place the connector elements in relation to any two surfaces that might impact each other, IDIADA did a 'volumization' exercise. Figure 2.)

While the "virtual sensors" could provide a more accurate idea of the amplitude of movement ("interference") between two parts, amplitude alone didn't predict the possibility of rattle, the engineers discovered. The value of penetration also had to be determined, i.e. how much the parts were interfering with each other. And the amount of this interference was, in turn, affected by the frequency at which the car components were vibrating at various points on the instrument panel.

For example, low vibration frequencies actually tend to result in higher amplitudes of movement of parts, yet the kinetic impact of any collision would be low. At higher frequencies, parts would actually be displaced less, but the velocity of any impacts would be higher. Taking all this into account, the IDIADA team determined that their "rattle ratio" had to be calculated as the amount of interference detected scaled by the kinetic energy at the impact time. With this rattle ratio in hand, the engineers could more accurately track the possibility of significant penetration, and hence the chance of an actual rattle happening, between parts in the instrument panel.

"For a better visualization, you can sweep the frequencies within the whole range of study and see at what frequencies rattle appears with a 2D plot," says Lama. "Or you can analyze the most critical frequencies using a 3D plot and, with scripting, create a file with the vector representation of rattle issues."

This time it was the ducts' fault!

Since the detection of rattle potential in a model is based on an accurate simulation of the frequency response of areas susceptible to contact events, the team took particular care to improve the correlation of their FEA models and their real-world tests. They achieved this by using a Modal Assurance Criterion (MAC) system that looked for similarities and differences, in simulation and test values, that could be used to help make the models more robust.

By adding more geometric details to their models, including simulation of the masses of the radiators in the HVAC system, and fine-tuning the stiffness of different connection points, the engineers were then able to pinpoint three areas in the instrument panel that influenced the results the most. These were the two connections of the HVAC duct leading to the two side diffusers and the connection of the HVAC duct leading to the central diffuser (Figure 3).

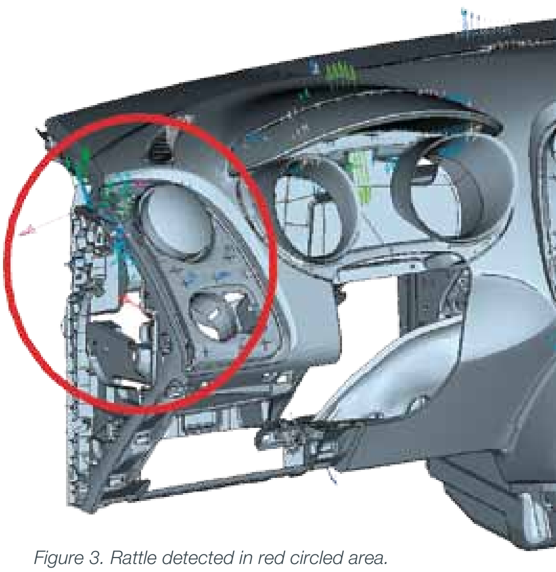


Figure 3. Rattle detected in red circled area.

Going even deeper with S&R analysis in the future

The first simulations of these three areas detected rattles with good correlation although, interestingly, the simulations detected many more rattling issues than the real-world tests. "In the future, we will be working on rattle detection criteria improvement to differentiate between rattles that can be heard and those that can't," says Lama. "We will also continue to refine our analyses to include those zones in the vehicle cockpit where there can be more problems with tolerances. At this point, we are beginning to succeed at speeding up the procedure for detecting movement and velocity of impact. Down the road, we will include some kind of tolerance criteria."

With IDIADA's advanced analysis methodologies continuing to evolve with the help of Abaqus, it looks like, "down the road," there will be a lot less squeak and rattle and many more happy automobile manufacturers—and drivers.

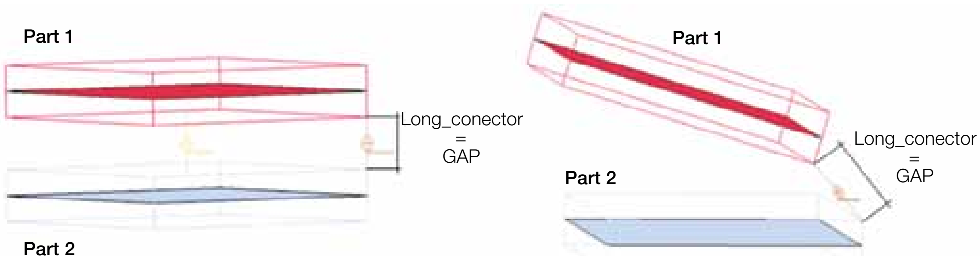


Figure 2. The use of Abaqus connector elements, between volumized, parallel (left) or angled (right) elements, enabled IDIADA to more accurately evaluate the gap between the elements and therefore the potential for rattle.

For More Information

www.idiada.com
www.simulia.com/cust_ref

Simulating Spacecraft Launch and Re-entry

Dutch Space uses Abaqus to virtually test a new metallic thermal protection system for hypersonic space vehicles

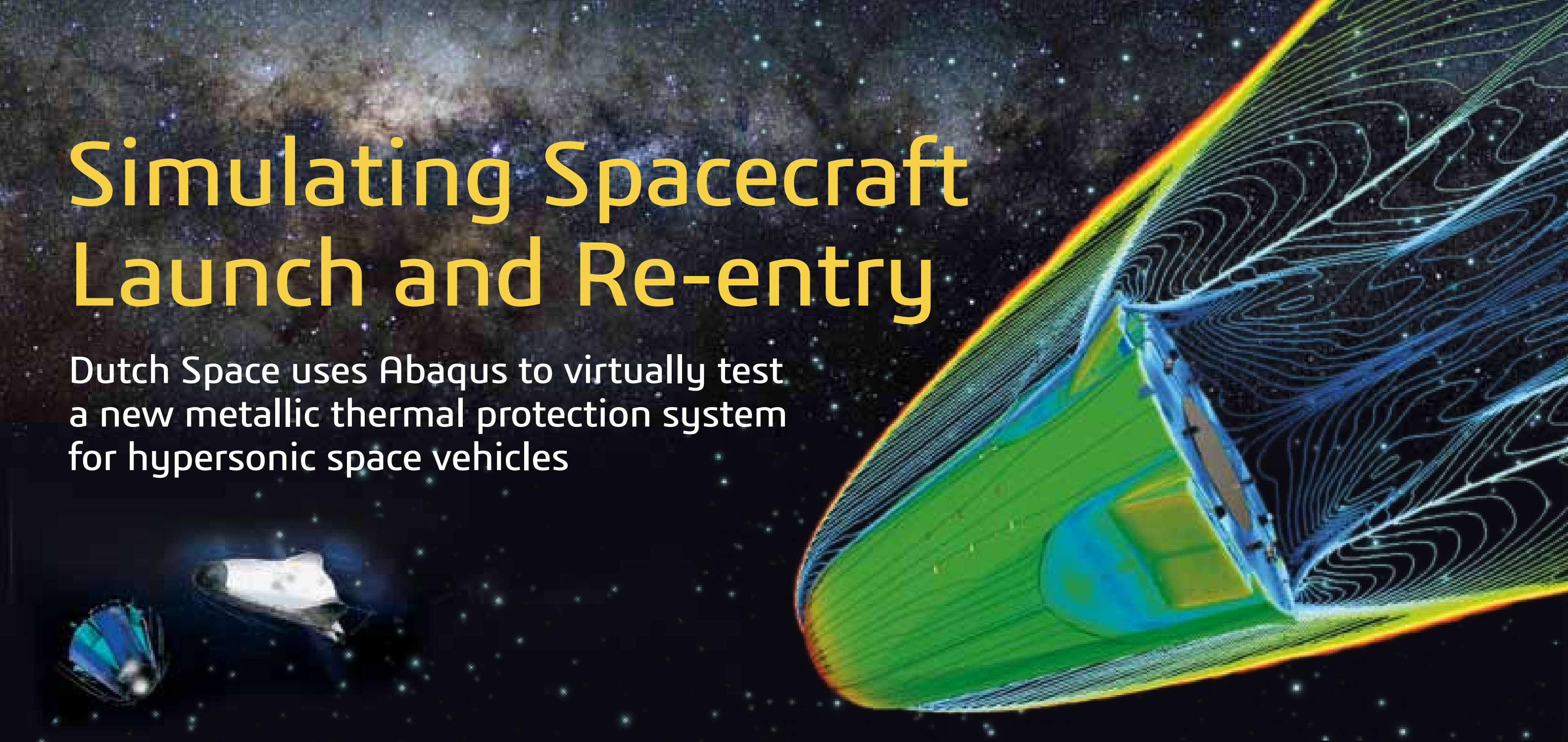


Figure 1. Simulation of the flow around EXPERT during atmospheric re-entry. Employing the CFD code, LORE, the heat flux over the vehicle's outer surface has been determined during the hypersonic phase (Courtesy L. Walpot, European Space Agency - AOES).

For a spacecraft to be reusable, it must first withstand a controlled explosion powerful enough to propel it through the earth's gravitational grip. Then to come home intact, it must survive the blast furnace of atmospheric re-entry with temperatures hot enough to melt steel. Acceleration, vibration, shock, skin friction, heat flux, and aerodynamic forces: These are not trivial design challenges. This *is* rocket science.

Thermal Protection Systems (TPS) have played a critical role in reusable vehicle designs since NASA engineers first began work on the Space Shuttle in 1972. Before that, with disposable space capsules (Mercury, Gemini, Apollo), ablative materials were used to absorb heat energy and then dissipate it through vaporization. To accommodate multiple missions, such as with the Shuttle, TPS designs took a different tack, utilizing a variety of materials (flexible insulation blankets, composites, ceramic tiles, and others) to protect the craft and its occupants from the extreme heat of re-entry. But since the Columbia accident in 2003, engineers have been particularly motivated to find more durable designs.

> We have confidence in the power of FEA to predict outcomes we can rely on.
>
> **Javad Fatemi**
> *Technical leader of the EXPERT's TPS project*

Since the mid-1990s, Dutch Space (Leiden, the Netherlands), a supplier of high-quality subsystems and products for the international space industry, has been a key player in the development of metallic TPS and hot structures for reusable craft. The metallic TPS of the European Experimental Re-entry Testbed (EXPERT), an unmanned capsule developed by the European Space Agency and the prime contractor, Thales Alenia Space of Italy, is one of Dutch Space's latest efforts (Figure 1). Its pending launch from a Russian submarine aboard a three-stage Volna rocket in the summer of 2011 promises validation, not only of new materials, but also of simulation as an integral tool in the development process.

New TPS technology: A tale of two materials

From the start, Dutch Space has focused on the use of metallic materials for TPS designs. According to Javad Fatemi, technical leader of the EXPERT's TPS project, metals have a lot of advantages over ceramics. "Metals are easy to use and maintain," says Fatemi. "They can handle impacts much better, and inspection is easier." But on the negative side, metals are heavier than other materials and can't cope with high temperatures as well as ceramics. So to take maximum advantage of material properties, Fatemi is betting on a TPS that uses a combination of metallic and ceramic parts.

The thermal protection system of the approximately five-foot-high, four-foot-diameter EXPERT is comprised of two major components: a nose cap made of a ceramic matrix composite (CMC), which takes the brunt of the thermal load during re-entry; and a conical metallic after-body that is manufactured from an oxide dispersion strengthened alloy, PM1000. These two materials were chosen for their high-temperature stability and weight efficiency. Four flaps that protrude from the surface at the base of the vehicle are also made of the more heat-tolerant CMC (Figure 2). "We couldn't make the whole craft out of metal," says Fatemi, "because the temperature is roughly 1900 degrees C at the nose and 2050 degrees C at the flaps, which are both higher than the 1200-1250 degrees C limit for the metal."

But while providing material advantages, a two-part, two-material design also introduced the project's greatest design challenge: The thermal expansion of the metal is eight times greater than that of the ceramic. "At the junction between the nose cap and the after-body, we had this huge thermal mismatch," says Fatemi. If the shape of the vehicle was to change during re-entry from the thermal mismatch—even just slightly—it could alter the flow from laminar to turbulent and, as a result, increase heat flux by a factor of two to three.

The solution was a structure that bridges the two components while allowing for differential heat expansion. "We needed to design the TPS so that we were sure it wouldn't trigger turbulence," says Fatemi. "Simulation was key to solving that problem."

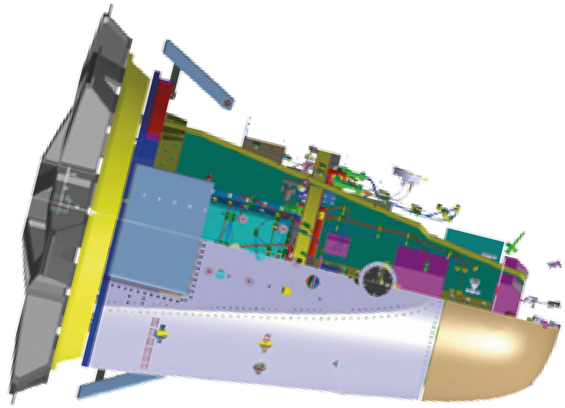


Figure 2. The European Experimental Re-entry Testbed (EXPERT), an unmanned capsule developed by the European Space Agency, utilizes a hybrid thermal protection system (TPS) comprised of a ceramic matrix composite (CMC) blunt nose (designed by DLR, Germany's national research center for aeronautics and space) and a metallic conical after-body (designed by Dutch Space). (Courtesy European Space Agency)

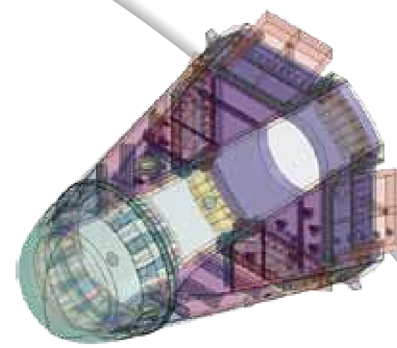


Figure 3. Thermal and structural models for the EXPERT TPS with integrated payloads.

Realistic simulation of launch and hypersonic re-entry

When Fatemi joined Dutch Space's EXPERT team in 2004, one of his first tasks was to compare commercially available finite element analysis (FEA) software that is used for simulating structural and thermal behavior. After conducting a tradeoff study, his reasons for selecting Abaqus were clear.

For one, says Fatemi, "With Abaqus, you have a unified finite element (FE) model. That means if you want to create both a thermal and a structural model, you can use the same mesh and easily change the element type and the boundary conditions."

Another reason was that the software could be easily coupled with CATIA. "We imported the CATIA model directly into Abaqus/CAE to make the FE model," says Fatemi. "Every time we made a revision based on our analysis, the CATIA model would automatically update."

Abaqus could also handle the complexity of the highly nonlinear TPS simulation with its robust interface between the pre- and post-processors and the solver. "The contact and specialized mechanisms, like special joints on the flaps, were all easy to model," says Fatemi. "With Abaqus, we are able to closely simulate reality."

The extreme in-flight forces that EXPERT encounters are a variety of thermal and mechanical loads: during the launch, it will experience acceleration, random vibration, and acoustic and shock loads; during re-entry, it will be subject to severe aerodynamic heating and aerodynamic forces; and there is, of course, the shock of impact when it comes back to earth, landing on the Kamchatka peninsula.

To accommodate the multiple physics that the vehicle is subjected to, Fatemi developed both 3D structural and 3D thermal models (Figure 3). He used the structural model to analyze the stiffness, strength, and thermo-mechanical behavior of the TPS when subjected to launch, aerodynamic heating, and pressure loads. In this model, all nonlinearities (geometrical, materials, and boundary conditions) were taken into account and the mechanical properties of all materials were treated as temperature dependent. The model used four-node linear quadrilateral shell elements (S4R), with each element having six degrees-of-freedom (DOF) per node (three translations and three rotations).

The thermal model was used to estimate the temperature histories for all TPS parts as well as payloads. This model was constructed from four-node, heat transfer quadrilateral shell elements (DS4) with DOF represented as temperature; it had a total of 112,994 elements and 128,358 nodes.

In the most recent analyses, Fatemi used CATIA V5 and Abaqus V6.10. Because of analysis complexity, he ran them on a high-performance Linux-based compute cluster for parallel computing.

Analyzing lift-off loads

The loads being analyzed during lift-off included quasi-static accelerations from the rocket, acoustic loads from the rocket engine, rapid depressurization at second-third stage separation, and random vibrations and shock each time a stage separates.

To ensure that the vibrations of launch do not occur at the vehicle's resonant frequencies, which can accentuate stresses, Fatemi carried out a normal mode analysis using the structural model. The requirement for minimum eigen frequency for the EXPERT was set at 80 Hz. The minimum value reached during analysis was 86 Hz, falling within specification.

A nonlinear static strength analysis was also performed to verify the structural performance of the TPS during launch. This analysis indicated that maximum loads occur during the separation of the second and third stages of the rocket and that during this event all EXPERT components had positive Margins of Safety (MS).

Re-entry challenges

While launch will generate harsh loads on the EXPERT, it is during re-entry that the vehicle is really tested by extreme thermal and mechanical loads. As the approximately half-ton vehicle slams into a wall of air at fourteen times the speed of sound, it generates tremendous kinetic energy. According to basic physical principles, this energy is translated into heat: when friction from the air slows the craft, its surface heats up, as governed by the relationship between air flow and heat flux. The 150-second re-entry window officially begins when the 436 kg (961 lbs) craft is at an altitude of 104 km (64.6 mi) and first encounters the atmosphere. Velocity at that time will be 5 km per second (11,184 mph), with an entry path angle of -5.5 degrees.

To simulate the aerodynamic heating of the EXPERT, a computational fluid dynamics (CFD) code, called Lore (developed by a student at Delft University of Technology), was used to calculate spatial distributions of heat flux over the vehicle's outer surface at five discrete speeds (between Mach 18 and Mach 10)

Continued

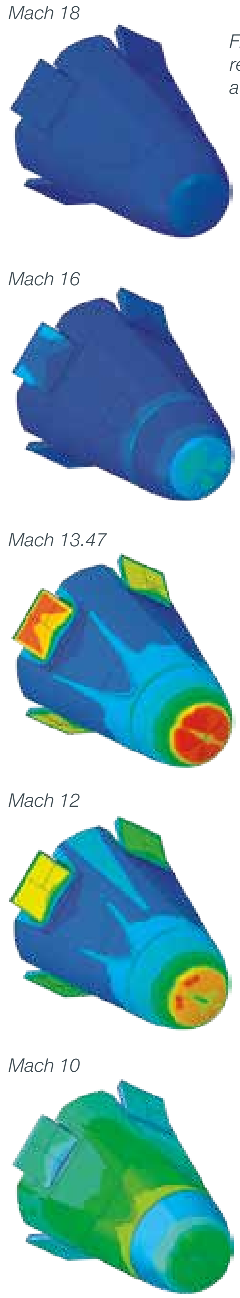


Figure 4. A series of simulation results for heat flux distributions at discrete Mach numbers.

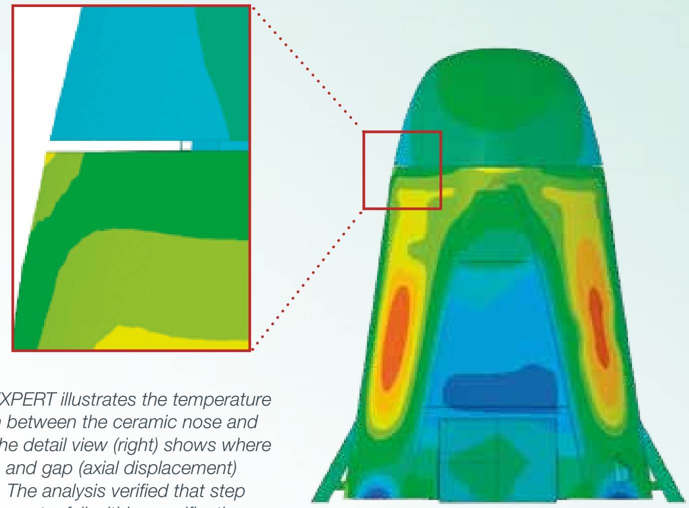


Figure 5. A side-view of EXPERT illustrates the temperature differences at the junction between the ceramic nose and the metallic after-body. The detail view (right) shows where step (radial displacement) and gap (axial displacement) measurements are made. The analysis verified that step and gap changes during re-entry fall within specifications.

on the re-entry trajectory. The heat flux distributions were then directly mapped from the CFD mesh to the thermal FE mesh in Abaqus using a dedicated sub-routine developed at Dutch Space (Figure 4). The stagnation heat flux profile was used to interpolate the spatial distribution of heat flux between different Mach numbers. Spatial distributions of dynamic pressure (at the same five trajectory points) were also calculated using CFD and mapped in the same way.

During re-entry, aerodynamic heating and dynamic pressure create severe loads on the TPS dependent on vehicle shape, atmospheric density, and speed. In addition, as the temperature increases, the mechanical properties of TPS materials degrade. To measure these effects, Fatemi used a sequential thermo-mechanical analysis, which predicted the temperature history in the EXPERT's TPS and payload, and generated maximum temperatures for each of the three critical EXPERT structures at specific times during re-entry: CMC nose (2165 K at 102 sec), CMC flaps (2328 K at 95 sec), and metallic after-body (1464 K at 112 sec).

The thermo-mechanical analysis also calculated the effects of the thermal mismatch between the composite nose and the metallic conical structure, generating gap (axial displacement) and step (radial displacement) measurements (Figure 5). Gap evolution is important in determining the design of the seals between the nose and the TPS, while the step values guarantee that changes in surface geometry will not trigger turbulent flow during re-entry. In both cases, analysis results fell within specified requirements. In addition, plastic stress/strain and MS (which vary as a function of loads and material temperature for specific parts) were calculated and determined to be within specification.

Confidence high in simulation results

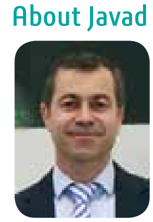
As the EXPERT project neared completion, the Dutch Space team took the final step of conducting physical tests to validate the predictions of analysis with actual measurements. In one scenario, Fatemi modeled and tested a highly nonlinear bolt joint at very high temperatures. In another, engineers conducted a dynamic test of the vehicle and compared it with simulation results.

Comparing test and analysis results, accuracy for the bolt joint was within three percent and for the dynamic test within five percent. "These comparisons gave everyone—the customer, the prime contractor, and Dutch Space—high confidence in the Abaqus FEA results," says Fatemi. "This is important since budgets were limited, full-scale thermo-mechanical tests are extremely expensive, and the EXPERT is going to fly without any full-scale thermo-mechanical tests."

The next project for Fatemi will be a complete vehicle analysis of EXPERT in which a CFD aerothermodynamic analysis is coupled with the Abaqus FE model (a first for non-military vehicles). He is also moving on to develop carbon fiber-reinforced polymer (CFRP) engine thrust frame technology for the European Space Agency's (ESA) next-generation launchers, with FEA slated to play a key role again.

"My objective is to build our capability for virtual testing and replace as much physical testing as possible," says Fatemi. "With EXPERT, we proved that simulation shortens the time-to-market and reduces costs. We have done all of our design verification using high-fidelity analysis and have confidence in the power of FEA to predict outcomes we can rely on."

About Javad Fatemi



Since arriving at Dutch Space seven years ago, Systems Engineer Javad Fatemi (Ph.D. mechanical engineering, University of Lyon 1, France) has lived and breathed every design detail and decision for the EXPERT's TPS, relying heavily on Abaqus and realistic simulation. He was honored by SIMULIA in 2009 (the bi-annual Ruben van Schalkwijk Award) for his technical achievements and enthusiasm, and continues to champion the use of simulation technology at Dutch Space. "I always wanted to be part of the space business," Fatemi says. "Every day, I love coming to work."

For More Information
www.dutchspace.nl

Aeroelastic Flutter: Validation of STAR-CCM+ and Abaqus Fully Coupled Co-simulation Model

From the bending of a tree's branch in the wind—to the flutter of an aircraft wing in flight, fluids and solids interact in harmony everywhere in the real world. However, in the virtual world of engineering simulation, the picture has rarely been quite so harmonious. Structural analysis and fluid dynamics, although intrinsically linked, have long been quite separate disciplines with the interaction between deforming structures and flowing fluids only being considered at a most basic level.

The challenges of fluid-structure interaction (FSI) are many: mapping data between the flow and structural solver, protocols and formats for data exchange between the solvers, coupling methods, dynamic fluid mesh evolution and lastly, proper validation of the model with experimental data. For these reasons, the numerical simulation of FSI problems was traditionally the preserve of research projects and academic studies, operating outside of the main engineering design process.

To satisfy the demand for a high-fidelity FSI solution that can be deployed in an industrial environment, CD-adapco and SIMULIA jointly developed a direct link between the STAR-CCM+ CFD package and Abaqus FEA, delivering coupled, two-way, fluid-structure interaction. Using direct co-simulation coupling provides efficiency and reduces overhead associated with things such as data transfer through file exchanges or use of external middleware software. CD-adapco's partnership with SIMULIA also means that setting up and running the problem may all be done within the easy-to-use STAR-CCM+ environment, with no need for writing scripts and input files or mapping data. The powerful physics of both codes may be leveraged in coupled FSI with STAR-CCM+'s full range of available models, providing the ability to study coupled, single and multiphase flows, chemical reaction and combustion as well as flow regimes from low speed to hypersonic. The options available in Abaqus are similarly broad, with coupled simulation supported for static stress/displacement, dynamics (implicit and explicit), heat transfer, temperature-displacement, thermal-electrical and piezoelectric analysis.

CD-adapco used the experimental flutter results on an AGARD 445.6 wing to validate the co-simulation close-coupling approach with Abaqus. The AGARD wing is one of the earliest and most studied experimental flutter test cases, and successful validation against this dynamic aeroelastic test case provides confidence in the accuracy and applicability of the STAR-CCM+/Abaqus FEA co-simulation approach.

The initial stage requires achieving a static aeroelastic state in which the fluid loads and the elastic restoring forces are in static equilibrium. Once achieved, an implicit unsteady aeroelastic simulation in the time domain was conducted and the Flutter Speed Index and Flutter Boundary Frequency were compared against the experimental data. For this simulation, a 2.2 million cell polyhedral mesh was created with prism layers growing out from the solid surfaces in order to accurately capture the boundary layer flow over the wing. The polyhedral mesh is illustrated in Figure 1 and Figure 2.

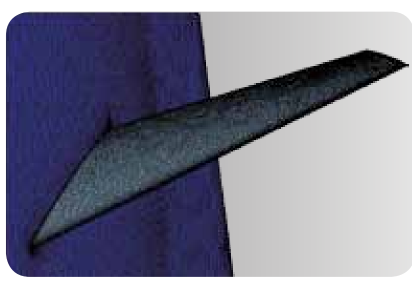


Figure 1. Polyhedral Mesh on AGARD 445.6 wing.

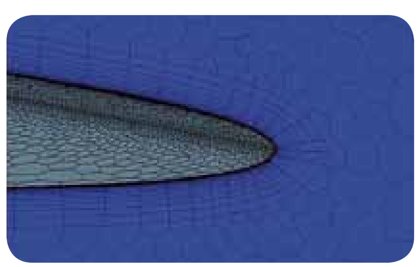


Figure 2. Close up of polyhedral mesh near wing tip with prism layers.

Using the coupled solver in STAR-CCM+, three Mach numbers, M=0.5, M=0.92 and M=1.14 were prescribed as freestream boundary conditions for which the aeroelastic solution was calculated. The multiquadric morpher capability of STAR-CCM+ was used to morph the fluid mesh around the wing due to its aeroelastic induced deformation.

In order to estimate the influence of calculation parameters on the results, several preliminary calculations were undertaken. These simulations were performed at M=0.5 for one full period of oscillation, using various time steps (from 0.1 to 0.4ms) and numbers of inner iterations (from 5 to 20), and the influence of these parameters on the damping effects on the wing tip displacement and calculation time were measured. The results show that the time step influences the results more significantly than the number of inner iterations, and allowed the prescription of optimal settings for step size and maximum inner iterations to give as little numerical damping as possible. The mesh morpher was tested in a standalone simulation by importing the Abaqus eigenmode solution and mapping to STAR-CCM+ mesh.

Four tests were conducted for the transient analysis at M=0.96 and FSI=0.32, one for each harmonic, over the period of the harmonic, using a time step of one-hundredth of the time period. The most important metrics for the validation study are the Flutter Speed Index which is a meaningful measure of the flutter and calculation of the Flutter Boundary. Flutter Speed Index as a function of Mach number can be compared to the experimental data to validate the co-simulation model.

For validation purposes, the numerical Flutter Boundary was calculated with the time domain method at M=0.5, M=0.96 and M=1.141. The results show that the the flutter boundary is well predicted across the range of Mach numbers and shows that the numerical results successfully capture the "sonic dip" near M=1.

The close-coupling between STAR-CCM+ and Abaqus brings the solution of a wide range of FSI problems within the easy reach of a typical engineer. In terms of both practicality and accuracy, co-simulation is the only way to tackle problems such as aerodynamic flutter, fluid induced bending, vortex induced vibration and galloping.

For More Information
www.cd-adapco.com

New Capabilities in SIMULIA's DesignSight and Simulation Lifecycle Management

The latest release of Dassault Systèmes V6 product suite, V6R2012, features new integrated design analysis capabilities within SIMULIA DesignSight and enhanced simulation processes and data management within SIMULIA SLM.

DesignSight enables CATIA V6 users to perform virtual tests during the design phase to quickly evaluate and improve the real-world performance of their design by integrating the robust linear and nonlinear Abaqus FEA technology in an easy and intuitive design environment. New features in the V6R2012 release include intuitive contextual balloons that simplify and accelerate the results review process by replacing the menu tool bars with an interactive selection capability. This provides users with easy access viewing the minimum and maximum results in a model, and the ability to select which regions of the model to show or remove.

SIMULIA SLM allows organizations to capture simulation knowledge, deploy approved methods, execute simulations on compute clusters, manage applications, and share simulation results. The latest release of SIMULIA SLM offers new template definition capabilities that improve control of simulations by enabling methods owners to specify mandatory fields, object substitution, and file import. This release also provides enhanced integration with ENOVIA Requirements Central by allowing simulation users to refer to product performance specifications, including test or use cases during the simulation process.

The V6R2012 release also introduces a new SLM product, SIMULIA Model Editor (MDL), which leverages the V6 open architecture by enabling third-party simulation data and models, such as FEA meshes, to be managed and edited directly on the V6 product structure. In addition, MDL allows analysts to work with their own managed edition of the V6 product structure independently, but fully aware of the in-work designer product structure, enabling efficient and high-quality concurrent design and analysis.

For More Information
www.3ds.com/products/simulia/portfolio/simulia-v6/

Latest Isight and SEE Releases Enable Optimization of Cyber-physical Systems

The Isight 5.5 release delivers additional application integration methods and enhanced optimization techniques, making it easier and more efficient for engineers to save time and improve product performance by optimizing them against performance or cost variables through statistical methods such as Design of Experiments (DOE) or Design for Six Sigma.

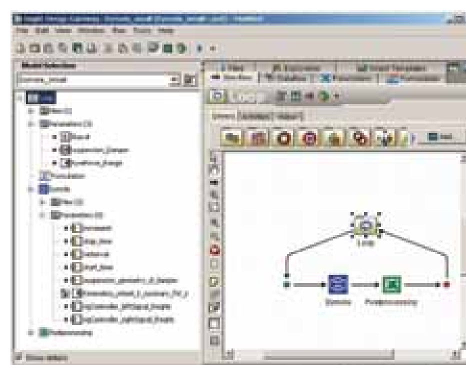
This latest Isight release continues to expand upon its leading open architecture by offering additional components that support the integration of modeling and simulation tools. Included in Isight 5.5 is a new component to support Dymola (also from Dassault Systèmes), which leverages the popular Modelica language for the modeling and simulation of dynamic behavior within complex integrated systems. The Dymola component in Isight allows our customers to optimize their products with a tight coupling between physical behavior and embedded computer systems.

Isight 5.5 also includes additional optimization capabilities, such as Mixed-Integer Sequential Quadratic Programming (MISQP) for solving problems with "real" and "integer" variables.

"Common design optimization problems that combine discrete and continuous parameters are often extremely difficult to solve," stated Dr. Klaus Schittkowski, professor for applied computer science at the University of Bayreuth in Germany. "For instance, when we try to find the smallest bolt possible, while not stripping the threads, only discrete combinations of bolt diameter and hole, pitch, flange diameter, thickness of the plate and nut height can be considered due to ISO regulations. The search space is no longer continuous and lacks derivatives, and has many isolated points and coupled parameters. The state-of-the-art Isight embedded MISQP code excels at solving these types of problems efficiently."

Key Features of Isight 5.5:

- Custom exploration strategy for creating custom search techniques at the application programming interface (API) level without having to program a custom optimization plug-in
- Support for deploying customized charts, including units on the axis and customization of an axis label, in a production environment



Isight simulation process flow utilizing the new Dymola component.

Key Features of the SIMULIA Execution Engine (SEE) 5.5:

- Secure credentials for simulation process flow data handlers and components
- Hot restart: In the event of an unplanned SEE shutdown or loss of network connectivity, any jobs that were executing at the time will be resumed when the SEE is restarted or network connection is restored
- Support for custom data handlers

For More Information
www.simulia.com/products/isight

Abaqus 6.11 Features Nonlinear Structural Optimization, Electromagnetic Analysis, and GPU-support

The Abaqus 6.11 release delivers a number of powerful, customer-requested enhancements for multiphysics, modeling, visualization, and performance.

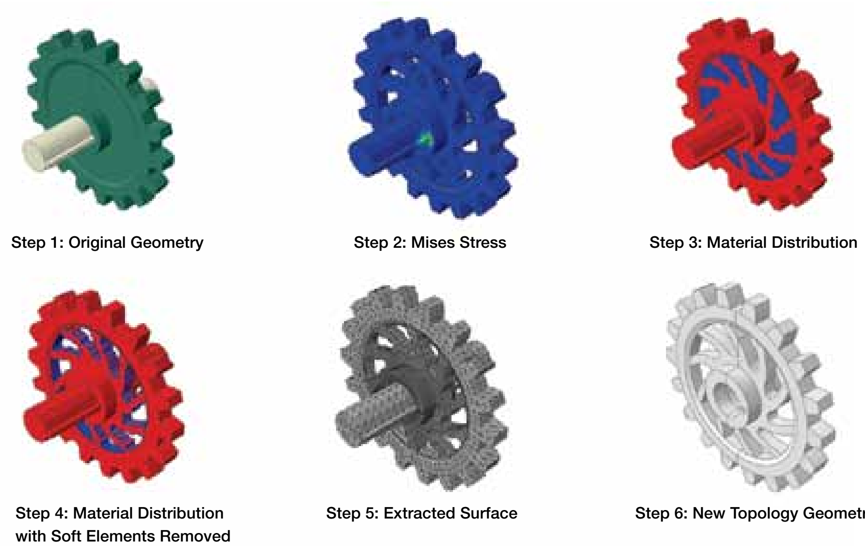
Among the key enhancements and new features, Abaqus 6.11 marks the availability of the Abaqus/CAE Topology Optimization Module (ATOM) as an add-on product, which enables users to perform topology and shape optimization that accounts for assemblies, large deformation, material nonlinearity and contact.

"We are exploring the possible uses of ATOM on different nonlinear models and customer cases," said Rob van Tol, program manager of Virtual Engineering at Sirris, a technological services company in Belgium. "We are impressed by the way ATOM has been integrated in Abaqus and we believe that nonlinear topology optimization will provide significant time and cost savings during our customer's product development process as well as enable them to be more innovative in finding the best design to meet their product's performance requirements."

Abaqus 6.11 also introduces a new electromagnetics solver to carry out problems requiring time-harmonic eddy current analysis, such as the hardening of a bearing surface due to induction. The release also provides a new smoothed particle hydrodynamics capability for modeling violent free-surface flows, such as fluid sloshing in tankers.

Customers will also have access to improved performance enhancements in Abaqus 6.11, including support for general-purpose computing on graphics processing units (GPGPU).

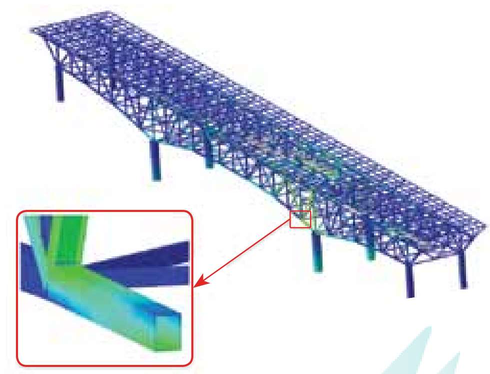
"The ability to run more design candidates, while shortening engineering cycles, is the future of computer-aided engineering," said Andrew Cresci, general manager of strategic alliances in NVIDIA's Professional Solutions Group. "CUDA-based GPU acceleration in Abaqus 6.11 can provide a 2x speed up for a range of models and industries, helping SIMULIA customers dramatically improve their workflow and deliver higher quality products to market, faster."



An example of the new Abaqus/CAE Topology Optimization Module, this image illustrates the setup and postprocessing of the topology optimization of a spur gear and shaft assembly.

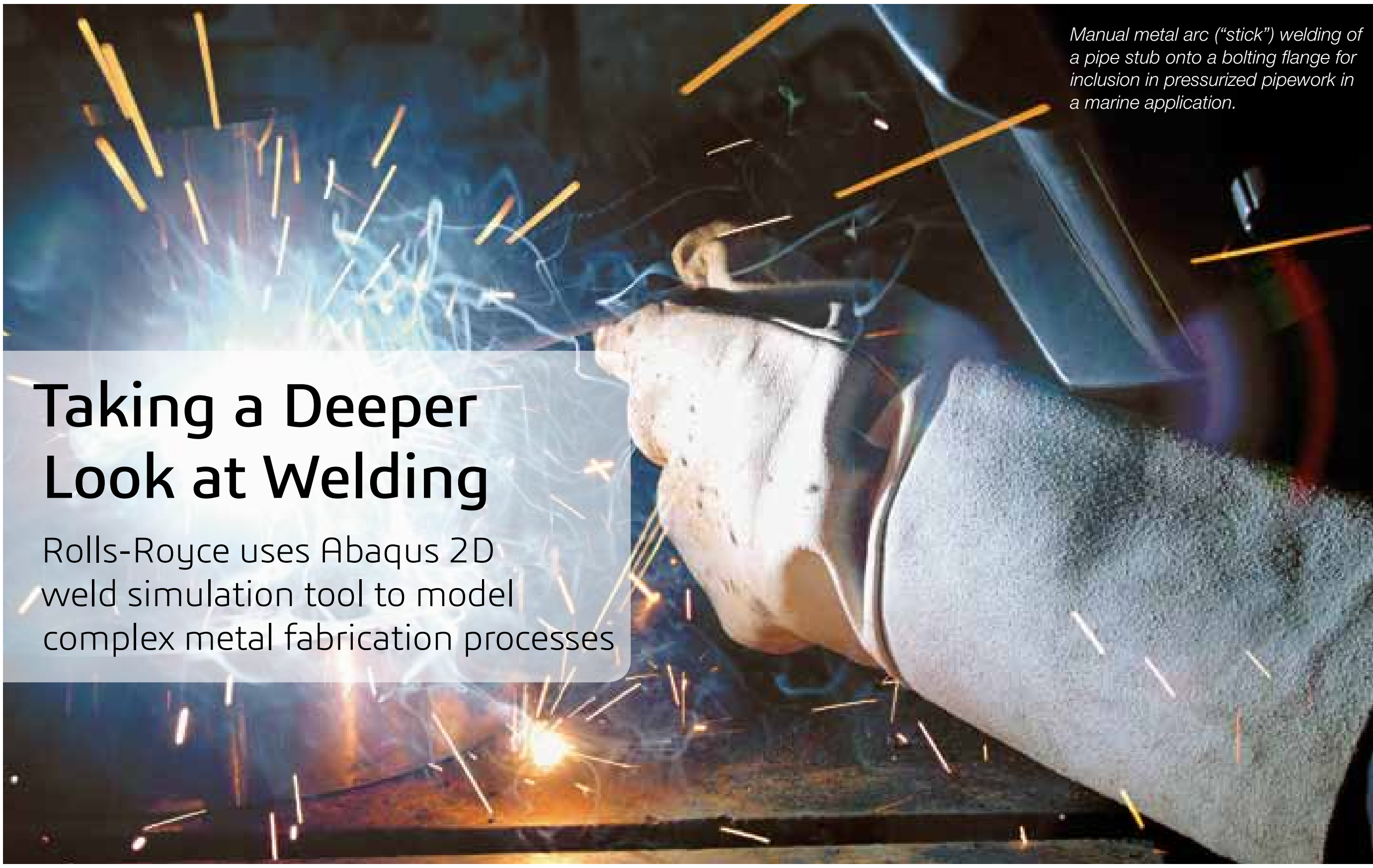
Key Features of Abaqus 6.11:

- Ability to define a spatially varying velocity when specifying inflow and outflow conditions and wall boundary conditions for a fluid dynamic analysis
- Enhanced CATIA V5 Bidirectional Associative Interface (AI) provides seamless parameter updating for rapid design analysis iterations
- Interactive mapping capability for defining spatially varying parameter values based on an external data source, such as pressure loads from a computational fluid dynamics (CFD) application
- More efficient large-scale fastener modeling for applications, such as rivets on an aircraft fuselage
- Ability to display stress variation on beam sections, such as structural members of an offshore platform subjected to wave loading
- Free body diagram (FBD) enhancements support analyses of force flow through a redundant structure
- Scalable thread-parallel execution capability of the parallel AMS eigensolver, for solving larger symmetric eigenvalues problems faster
- Parallel cavity radiation scheme for modeling heat transfer effects due to radiation in enclosures, such as heat transfer in an engine exhaust
- Parallel frequency response solver to support shared memory parallel (SMP) with up to 24-cores, providing class-leading performance



Abaqus 6.11 delivers the ability to display stress variation on beam sections, such as bridge beams subjected to dynamic loading.

For More Information
www.simulia.com/products/abaqus_fea



Manual metal arc ("stick") welding of a pipe stub onto a bolting flange for inclusion in pressurized pipework in a marine application.

Taking a Deeper Look at Welding

Rolls-Royce uses Abaqus 2D weld simulation tool to model complex metal fabrication processes

The idea of joining metals together dates back to the Bronze Age. Forge welding advanced in the Middle Ages, when village blacksmiths hammered heated metal until bonding occurred. The discovery of the electric arc in the 19th century sparked the development of many welding techniques in use today, with advances in electron beams and lasers further refining the technology in more recent decades.

Many industries—including shipbuilding, aerospace, defense, offshore energy, automotive, and nuclear—continue to rely heavily on welding to build their core products (a single automobile can contain 5,000 to 10,000 welds). Yet the practice of welding is still considered to be more "art" than "science," highly dependent on the skill and competence of the individual welder, or the quality of robotic welder programming.

In a process that involves high temperatures, material phase transformation, and the deposition of material, it's a challenge to produce the perfect weld every time. So engineers are looking for ways to study and predict the effects of different welding techniques on the behavior of the materials being fused. This knowledge can help avoid design guesswork, speed up the product development process, and contribute to higher quality of finished products. What's more, by identifying the best welding methodologies, the information can be incorporated into improving the training of human welders, and the programming of robotic ones as well.

Autogenously welded plate sample.

Realistic simulation deepens knowledge of welding process

Realistic simulation with finite element analysis (FEA) is a key tool for the design engineers in Rolls-Royce's Marine division, who are currently quantifying and verifying the many parameters involved in welds used in marine power plants and propulsion. "Welds are a complex modeling problem requiring both thermal and structural solutions," says David Hodgson, stress engineer, primary components, Rolls-Royce (Derby, U.K.). "We are looking to predict the distortion of components during manufacture, the position and magnitude of peak residual stresses, and the effects of the welding process on the metals involved."

Hodgson's group has an ongoing research partnership with the Materials Science Center of The University of Manchester, and Serco Assurance. Rolls-Royce has been evaluating several FEA packages for years, including SYSWELD, VFT, and Abaqus FEA from SIMULIA, the Dassault Systèmes brand for realistic simulation. Recently, as part of their strategy to improve their capabilities to accurately perform weld analysis, the company added the Abaqus Welding Interface (AWI). An extension to Abaqus/CAE, the product streamlines the generation of two-dimensional welding simulations by providing a graphical user interface (GUI) for defining all aspects of the weld model such as weld beads, weld passes, film loads, and radiation loads."The weld simulation tool is helpful because it allows us to carry out further analysis of both thermal and structural models directly within the Abaqus unified FEA environment," says Hodgson. "This has helped us reduce data translation issues, training time and expense."

Simulation goes under the torch

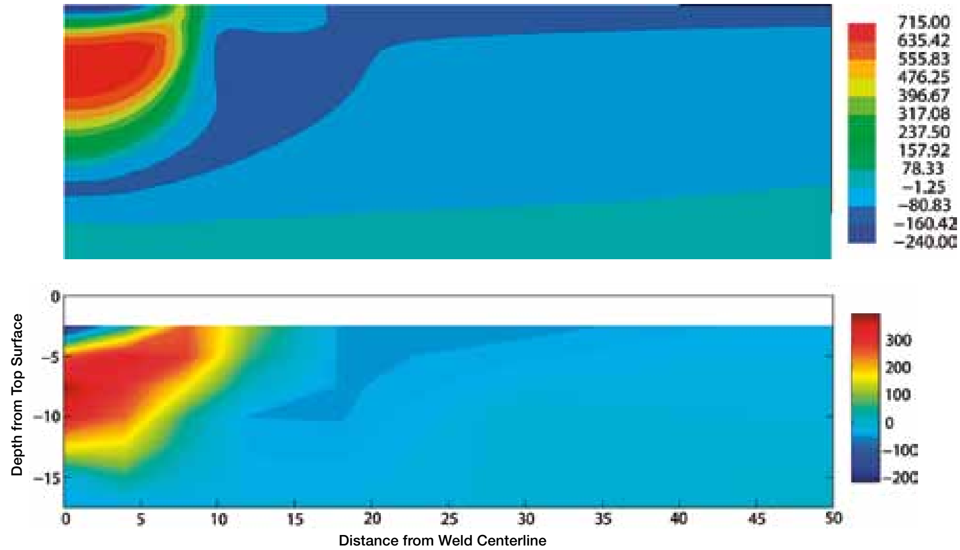
The research team decided to use the AWI tool to examine three different welding scenarios: autogenously (self-material) welded plates, an eight-pass groove-welded plate and a seven-pass ring weld disk. The autogenously welded plate model simulated a single plate of pressure-vessel steel melted by one or two passes of a welding torch. The more complex eight-pass and ring-weld steel models involved multiple torch passes and a ferritic steel filler material—which might be used in an offshore oil installation, or a nuclear reactor. Both quadrilateral and triangular heat transfer elements were used for the thermal models, while a generalized plane strain model was used for the structural analysis.

Applied heat, and the metal's structural response to that heat, are intimately intertwined in the welding process. But from the simulation point of view, each of these separate parameters needs to be defined so they can be included in a multiphysics analysis of their coupled interaction. Through the AWI, the temperature history calculated in the thermal model provides the temperature input (i.e. the load) for the structural model. This allows the structural model to analyze the thermal expansion and contraction of the metals being welded that result from changes to the materials' mechanical properties.

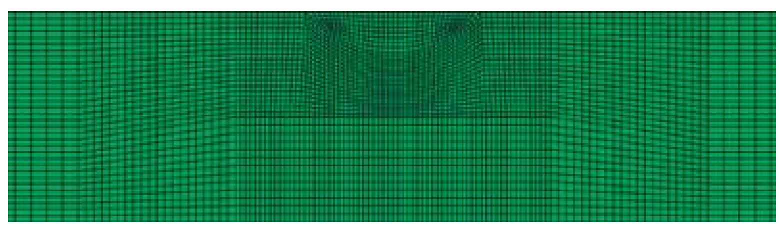
Automating surface definitions speeds up modeling

To set up such a model, the AWI imports a basic meshed part (with no boundary conditions, loads or interactions) with its materials and sections defined, and allows the user to create weld beads along the area to be welded. The modeler then automatically defines weld passes based on the weld bead order, and assigns surface film and radiation heat-transfer properties.

"One of the most time-consuming aspects of weld modeling is the surface definitions for heat transfer coefficients," says Hodgson. "These definitions have to be constantly updated as the weld build-up modeling progresses. The automation of this by the AWI helps speed the process considerably." The method used for controlling the heat flux within the thermal model involves setting a fusion boundary: Sensor nodes within the mesh, defined at specific depths, are used to end the heating step when their average temperature reaches a predetermined limit (1500 degrees Celsius in this case).



FEA image (top) and Neutron diffraction (bottom) stress measurements in one-pass test sample.



FEA model of autogenously welded plate.

Real-world welding shows stress results

To verify their models against real-world results, the engineering team created small welding test specimens of the pressure-vessel steel and measured the residual stresses after passes by a mechanized Tungsten Inert Gas welding head. The measurement process involved neutron diffraction (ND), which (in a manner similar to X-rays) involves placing a sample in a beam of neutrons and recording the diffraction intensity pattern that results from changes in the structure of the crystalline solids in the steel. The pattern records the crystal lattice spacing within the steel that can be compared with a stress-free spacing value to calculate a strain measurement. This strain measurement can then be used to infer the stress present within the metal.

The autogenously welded FEA models were compared with the ND test results and the one-pass results showed a particularly close correlation. Work continues on refining the more complex models for accurate simulation of the many intertwined processes involved in welding. "The 2D weld modeling interface is a useful tool to combine with Abaqus/CAE," says Hodgson. "Its evolution into a full 3D moving heat source is desirable and should be conducted in conjunction with advanced weld material modeling tools."

For More Information
www.rolls-royce.com/marine
www.simulia.com/products/welding

Award-Winning Research Advances Composite Analysis Methods

Silvestre Pinho with Aeronautics engineering students at Imperial College London.

Silvestre Pinho, a senior lecturer in Aero-Structures at Imperial College London, Department of Aeronautics, was the recent winner of the 2010 Albert Cardon Award for "Best Under 35-Years-Old European Researcher in Composite Materials," given once every two years by the European Community in Composite Materials. To those who have met him, it's evident that Pinho brings enthusiasm and innovative methods to his research.

"My research is focused on multiscale structural models for advanced composite materials," Pinho commented. "The importance of this area in aeronautics is easy to understand: While modern commercial aircraft look very similar to those from a few decades ago, the structural materials inside are completely different. And again, in a few decades, aircraft will be made of different materials, and the evolution is only possible because significant research is being devoted to it."

Pinho's workplace and alma mater, Imperial College London, is consistently ranked in the world's top 10 in the (*Times Higher Education)* World University Ranking. Its Department of Aeronautics was rated with the top classification (5*) in the last Research Assessment Exercise, undertaken on behalf of U.K. higher education funding councils to evaluate the quality of research conducted by British higher education institutions. Pinho's group currently has 10 Ph.D. students and two Post Doctoral Research Assistants. To date, his group's main contributions have been the development of micromechanical models for different failure mechanisms for carbon fibre reinforced composites and the development of mode I translaminar fracture toughness tests, a numerical smeared crack formulation, cohesive elements and multiscale modeling.

In 2009, Pinho collaborated with SIMULIA in the 3rd World-Wide Failure Exercise (WWFE), where accurate modeling of crack propagation in composite materials is essential. Pinho later prepared a framework for user-defined failure criteria suitable for composites to be used in conjunction with the XFEM feature in Abaqus. Through collaboration with SIMULIA R&D this technique—which allows users to define the onset of crack propagation, the orientation of the fracture plane and the fracture toughness as a function of the damage mode predicted by user stress- and strain-based criteria—has been included in Abaqus starting with the pre-release version of Abaqus 6.10-EF.

"Abaqus is an excellent platform for structural research, as it allows researchers to develop and test their own research models, while benefiting from a state-of-the-art finite element code, including complementary models and user-friendly pre- and post-processing," says Pinho. "SIMULIA staff members are very open to collaboration with model developers. They are supportive and responsive to ideas and suggestions. They also have a positive, strategic view of the importance of academic model developers for Abaqus."

Beginning in 2011, SIMULIA will be involved with Pinho on a new research project that his group has outlined as: "Bridging the Scales; from the toughness of small specimens to the damage tolerance of large aerospace panels." Airbus will also be involved in this three-year project, sponsored by the Engineering and Physical Sciences Research Council. The goal of this research project is to develop methods to accurately predict the strength and damage tolerance of large composite components by gaining a deep understanding of 1) the failure process at the micromechanical level, 2) the corresponding energy absorbing mechanisms, and 3) the different scales involved in the fracture process.

"The demands of new aircraft developments require a significant improvement in the modeling capabilities for characterizing and simulating the behavior of laminated composite materials," says Alan Prior, SIMULIA's aerospace lead in Europe. "This research, to work in parallel on both testing and modeling, is very important and distinct. It is critical for our industrial customers that new and innovative numerical methods are supported with real physical data rather than being theoretically isolated. Silvestre's team research proposal is very timely and will help SIMULIA address major requirements for the realistic simulation of composite structures."

Silvestre Traveira Pinho studied Mechanical Engineering at the University of Porto and graduated with an M.Sc. in Mechanical Engineering in 2002. During his Masters degree, numerical modeling stood out as a very active and interesting research area. His research is focused on multiscale structural models for advanced composite materials. He obtained his Ph.D. in Aeronautics from Imperial College London in 2005.

For More Information
www3.imperial.ac.uk/people/silvestre.pinho

Bird Strike Damage Analysis Using Coupled Eulerian-Lagrangian Method in Abaqus

Bird strike analysis of aircraft components is an important step in the process of flight certification and validation. Due to the advancements in numerical simulation technologies, it is now possible to obtain analysis results that successfully replicate the physics of the real event. The Department of Aeronautical Engineering, Faculty of Mechanical Engineering and Naval Architecture at the University of Zagreb, has developed a numerical damage predication procedure to gain insight into complex structural behavior of impacted structures without the need for time consuming and costly physical experiments. The accuracy of numerically, as well as experimentally, predicted responses greatly depends on the physically correct modeling of bird replacement models. Due to the fact that the stiffness of the impacting bodies is much lower compared to the impacted structures, bird strikes are classified as soft body impacts. During the impact, the bird is subjected to stresses that greatly exceed the material's strength. In addition, large deformation of material during soft body impacts has traditionally presented a source of numerical challenges in the finite element analyses.

The Coupled Eulerian-Lagrangian (CEL) capability of Abaqus/Explicit is a very powerful tool in overcoming these challenges. It has been applied to efficiently capture the fluid-like bird behavior upon impact. The great advantage of CEL analyses is that most of the problems associated with extensive bird mesh distortion are eliminated, as the Eulerian description allows finite elements to be fixed in space and the material to flow through these elements. On the other hand, the impacted structures are discretised by traditional Lagrange finite element formulation. The impacting forces are transferred to the Lagrangian structure through Eulerian-Lagrangian contact, which is based on the penalty contact algorithm.

The numerical bird strike simulation has been applied on a very detailed airliner flap model consisting of composite laminates, sandwich structures and metallic structural items. The model is assembled of solid, conventional and continuum shell and beam elements, while tie surface-based constraints have been employed in order to connect meshes of elements having dissimilar numbers of degrees of freedom. The Abaqus built-in progressive damage and failure model completely fulfills the requirements of the simulation and has been applied to model impact damage on the composite part of the structure.

The great advantage of CEL analyses is that most of the problems associated with extensive bird mesh distortion are eliminated, as the Eulerian description allows finite elements to be fixed in space and the material to flow through these elements.

The composite components of the model use Hashin's failure initiation criterion and accounts for the following failure modes: fiber rupture in tension, fiber buckling and kinking in compression, matrix cracking under transverse tension and shearing, matrix crushing under transverse compression and shearing. Damage is modeled employing continuum damage mechanics principles using damage parameters which modify the initial undamaged elasticity matrix. Failure of metallic structural items has been modeled by shear failure criterion, which is based on the accumulated equivalent plastic strain and suitable for high-strain rate dynamic problems. The shear failure criterion has also been employed to predict crushing of the Nomex honeycomb core.

Bird material has been replaced with an equal mass of water, as birds mostly consist of water and air trapped in the bones and lungs. To take the trapped air into account the density has been reduced to 938 kg/m3, while the Mie-Grüneisen equation of state has been used to model constitutive behavior. The bird geometry has been replaced with a cylinder having hemispherical ends and a length-to-radius ratio equal to two, as this geometry best resembles pressure time histories of real birds during impact tests and corresponds to the geometry used in the experimental work.

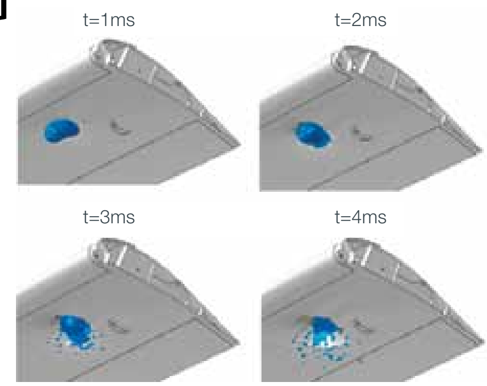


Figure 1. Deformation of a 1.81 kg bird during the impact.

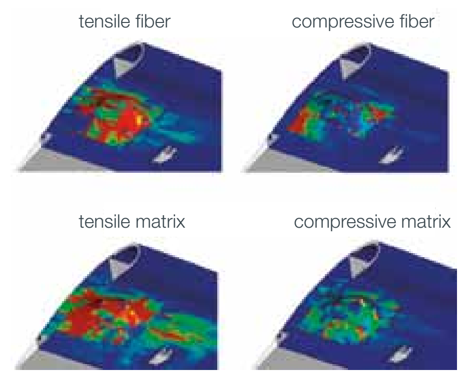


Figure 2. Contour plots of Hashin's failure criterions of lower flap skin (cross-sectional view) at an impact of a 1.81 kg bird.

The results shown in this contribution demonstrate the damage prediction capability in a simulation of a bird strike on an inboard airliner flap involving a 1.81 kg bird at 100 m/s. The kinetic energy of the impact event (Figure 1) is sufficient to cause severe damage on the flap as the bird penetrates the lower flap skin and impacts the upper skin. The dominant failure modes of the composite flap skins are tensile fiber and matrix failure (Figure 2). Although the impact in this case resulted in complete failure of the lower skin and serious damage of the upper skin, the main load carrying structural items were not affected by the impact.

Abaqus/Explicit has proven to be a very efficient and numerically stable tool. As the software has great potential in implementing new constitutive and damage models, the ongoing research will couple the effects of aerodynamic loading as well as implement other failure modes.

For More Information

http://aerodamagelab.fsb.hr/
Prof. Dr. Sc. Ivica Smojver
ismojver@fsb.hr

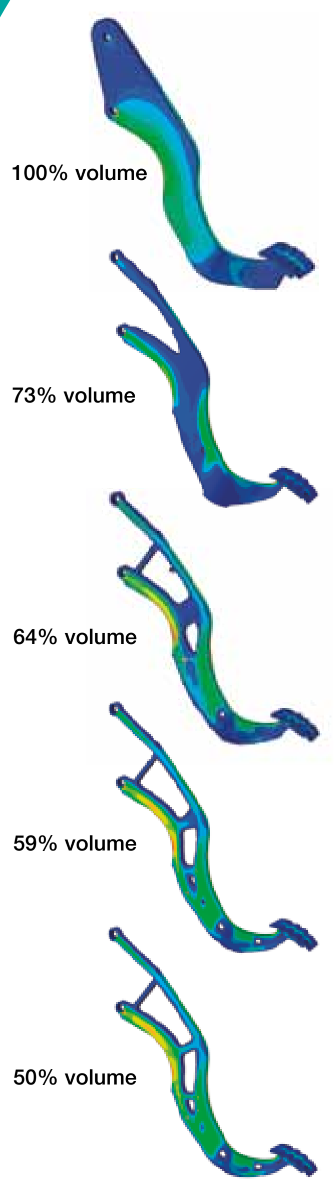
Abaqus/CAE Topology Optimization Module (ATOM)

How to reduce a part's volume while maintaining its structural integrity using ATOM.

ATOM delivers advanced capabilities for nonlinear structural optimization and provides important benefits to engineers and product designers by identifying the optimized topology and shape of a structure.

Designed with ease-of-use in mind, ATOM is a seamless extension of Abaqus/CAE and enables nonlinear topology and shape optimization within the familiar Abaqus product interface. It is focused on enabling engineers to determine an optimum topology and/or shape for a part or assembly. Once the analysis has completed, the final shape can be exported in a CAD neutral format to facilitate an update to the original geometry.

Topology and shape optimization allows the engineer to explore a variety of designs to improve performance, while reducing development time and product costs.



Nonlinear optimization performed on a brake pedal using ATOM resulted in a 50% reduction in mass, more torsional stiffness, and addressed out-of-plane warping effects, which would otherwise not be considered using a linear optimization tool.

TIP: HOW TO REDUCE THE WEIGHT OF A PART

ATOM is an add-on product for Abaqus/CAE that enables engineers to optimize a design under real-life loading conditions. This gives the designer confidence that the final part will perform its function correctly without additional wasted material or incident.

HERE'S WHAT YOU DO:

1. Build the model in Abaqus/CAE and run a test job to ensure the model runs successfully
2. Change to the Abaqus optimization module
3. Create a new topology optimization task and select the region for optimization
4. Create design responses for volume and strain energy
5. Create an objective function to minimize the strain energy design response
6. Create a constraint to reduce the volume design response
7. Optionally add manufacturing restrictions or other constraints or objectives to the design
8. Change to the job module in Abaqus/CAE and create a new optimization process
9. Run the newly created optimization process
10. Postprocess the results
11. Optionally, you can export the results in a CAD neutral format

For More Information
www.simulia.com/products/atom

2011 Regional Users' Meetings

Our Regional Users' Meetings (RUMs) provide a valuable forum for discovering and understanding how experts in engineering and academia are applying the latest simulation technology and methods to accelerate and improve product development. Attend the upcoming RUM in your area to learn about the latest enhancements to our products and the ongoing strategy of SIMULIA. For additional information, visit www.simulia.com/events/rums.

Americas

September 28–29	Chicago, IL
October 11–12	Canada
October 24	Houston, TX
October 27	San Diego, CA
October 27	Ohio
November 1–2	Plymouth, MI

Europe/Middle East/South Africa

September 27	Italy
September 19–20	Bamberg, Germany
September 22–23	Copenhagen, Denmark
September 29–30	Paris, France
October 11–12	Czech Republic
October 18–19	United Kingdom
October 27–28	Poland
November 14–15	Austria
November 17–18	The Netherlands

Asia Pacific

October 13–14	Seoul, Korea
October 18–19	Baengaluru, India
October 20–21	Singapore
October 26–27	Suzhou, China
October TBA	Taiwan
November 28–29	Yokohama, Japan

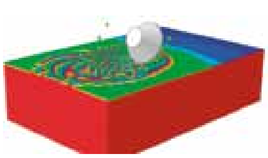
Benelux 2010

China 2010

Czech Republic 2010

Technology Briefs

Inspired by real-world projects, Technology Briefs provide detailed application examples on the use of SIMULIA solutions in a wide range of industries. Over 40 Technology Briefs are available at our website. Below are the new additions for 2011:

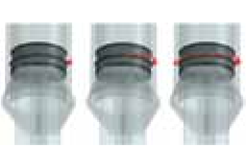


Water Landing of Space Flight Re-entry Vehicles Using Abaqus/Explicit

Space flight re-entry vehicles impart highly dynamic loads on the crew and/or payload during a water landing. This Technology Brief demonstrates how the Coupled Eulerian-Lagrangian (CEL) method in Abaqus/Explicit which provides the means for analyzing these complex physical phenomena in the water landing of a re-entry vehicle.

Simulation of the Quasi-static Crushing of a Fabric Composite Plate

Simulating the crushing response of composite structures can significantly shorten the product development cycle and reduce cost in the Aerospace, Automotive, and Railway industries. This Technology Brief describes a methodology for modeling this behavior using Abaqus/Explicit, which shows very good quantitative and qualitative agreement between the numerical results and experimental data.



Pre-filled Syringe Failure Analysis Using Abaqus/Standard

During its working life, a pre-filled syringe experiences stresses that may result in material damage. Specifically, the syringe barrel may develop microcracks that coalesce and propagate, causing the syringe to fracture and its contents to lose sterility. Abaqus/Standard offers the capabilities necessary to include fracture and failure in the syringe design. This Technology Brief demonstrates the application of the extended finite element method(XFEM) to analyze crack initiation and propagation along an arbitrary, solution-dependent path without the need for remeshing.

For More Information
www.simulia.com/techbriefs