

SIMULIA

Realistic Simulation News

September/October 2011
www.simulia.com



**Rolls-Royce:
Speeding the
Route to Quality
with Isight**

U.S. Army-ARDEC
Simulating Fracture
with Abaqus
Page 6

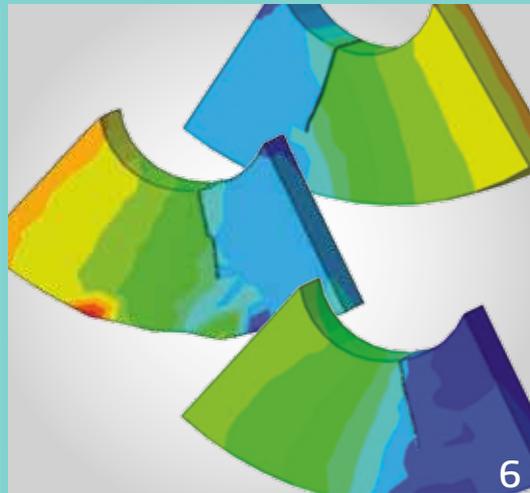
TÜV NORD
Managing Energy
Ups and Downs
Page 16

Lenovo
Designing out Flex,
Engineering in Stiffness
Page 18

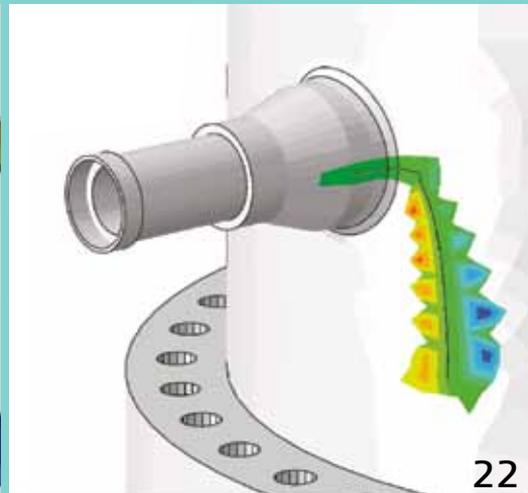
 **SIMULIA**



12



6



22

Inside This Issue

12 Cover Story

Rolls-Royce: Speeding the Route to Quality with Isight

On the cover: Dierk Otto

6 Customer Case Study

U.S. Army–ARDEC Simulates Fracture with Abaqus

22 Tips & Tricks

eXtended Finite Element Method (XFEM): How to Estimate the Safe Operating Life of Structural Components

In Each Issue

3 Executive Letter

Bruce Engelmann, Chief Technology Officer

4 Customer Spotlight

Auburn University Leverages XFEM for Survivability Assessment of Fine-Pitch Electronics

8 Strategy Overview

Zhen-zhong Du, technology lead for fracture & failure technology

11 Alliances

e-Xstream engineering: Fatigue and Failure of Short Fiber Reinforced Plastics
Component Central Enhanced

14 Customer Case Study

Structural Engineers Use Abaqus to Satisfy Stringent Fire Safety Roofing Standards

16 Customer Case Study

TÜV NORD Uses Realistic Simulation to Help Power Providers Manage Energy Ups and Downs

18 Customer Spotlight

Lenovo Designs out Flex, Engineers in Stiffness with Realistic Simulation

20 Academic Update

University of Girona: Virtual Testing of a Composite Cylindrical Lattice Structure

University of Iowa: Fracture Modeling of Ceramic Liners for Total Hip Arthroplasty

23 News and Events

- 2012 SCC—Save the Date!
- New SLM Whitepaper Available
- V5R21 Release Update
- SIMULIA Online Resource Center

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

Karen Curtis

Associate Editors

Rachel Callery
Tim Webb

Contributors

Lyonel Reinhardt (U.S. Army–ARDEC, Picatinny Arsenal), Axel Schulz (TÜV NORD), QingJuan Zhen (Lenovo), Dierk Otto (Rolls-Royce), Zhongcheng Ma (Z.Ma Research and Consulting), Jordi Torres (Universitat de Girona), Jacob Elkins (University of Iowa), Pradeep Lall (Auburn University), Elodie Maurer (e-Xstream engineering), Parker Group, Bruce Engelmann, Dale Berry, Zhenzhong Du, Matt Ladzinski, Jill DaPonte, and April Alfieri (SIMULIA)

Graphic Designer

Todd Sabelli

The 3DS logo, SIMULIA, CATIA, 3DVIA, DELMIA, ENOVIA, SolidWorks, Abaqus, Isight, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners. Copyright Dassault Systèmes, 2011.

Advanced Mechanics Technology—the Foundation of Realistic Simulation Methods Evolution



Bruce Englemann
Chief Technology Officer

Our global team of researchers and developers is at the forefront of the evolution to develop new mechanics simulation technology and methods to achieve deeper understanding of real-world behavior. I purposely use the word *evolution*, as we have been on this journey of exploration and discovery with our customers and partners for more than three decades.

Today, our long-term commitment to innovation and meeting industry requirements is paying off. Improvements in computing power and proactive customer engagements are helping to accelerate our delivery of advanced FEA, multiphysics, and optimization technology. At the same time, we are experiencing growing customer demand to migrate their legacy simulation tools and methods to our robust, scalable, and higher-fidelity solutions. Our customer's involvement in this evolution is illustrated through our Fracture Customer Review Team, featured on page 8 in this issue. This team, along with other customer review teams, meets regularly with our R&D teams to share their requirements which drive new developments to help solve specific industry challenges.

With the release of Abaqus 6.11, we made major strides in our goal of making the modeling of fracture and failure as common as including the effects of Mises plasticity. You can read about customers in this issue of *Realistic Simulation News* who are using the innovative eXtended Finite Element Method (XFEM) technology in Abaqus to simulate stationary and propagating cracks in their 3D models; whether it be the fracture of a pressure vessel due to operational thermal loads (TÜV NORD, page 16), co-simulation with XFEM to safely model equipment drop test (U.S. Army – ARDEC, page 6), or the analysis of ceramic hip bearing fracture (University of Iowa, page 21).

In the coming months, you will be able to take advantage of the advances of our solver technology within a new paradigm of user experience, collaboration, and simulation process and data management. Our next-generation Realistic Simulation solutions are rapidly being fully-integrated into Dassault Systèmes Version6 platform. These new products will enable your company to leverage simulation at the very beginning of the product development process and throughout your product development cycle—from concept, to detailed design, to manufacturing, to consumer-use, and even maintenance and recycling.

You will be learning more about our new products at the user meetings and customer conferences being held around the world this fall. I invite you to come to Providence, RI, to meet with our R&D management at our brand's headquarters and to attend the 2012 SIMULIA Customer Conference next May (see page 23). By being an active member of our global community, you will be able to share your requirements for this on-going technology and methods evolution and ensure that we deliver market-leading solutions that help you solve your engineering challenges.

I look forward to meeting you in Providence soon.



Download select 2011 SIMULIA Customer Conference Papers that provide details on how our customers are applying realistic simulation to better understand fracture and failure.

Fiat Research Centre—*Mixed Lagrangian Eulerian Method for Tire/Road Interaction in Finite Element Vehicle Dynamic Simulation*

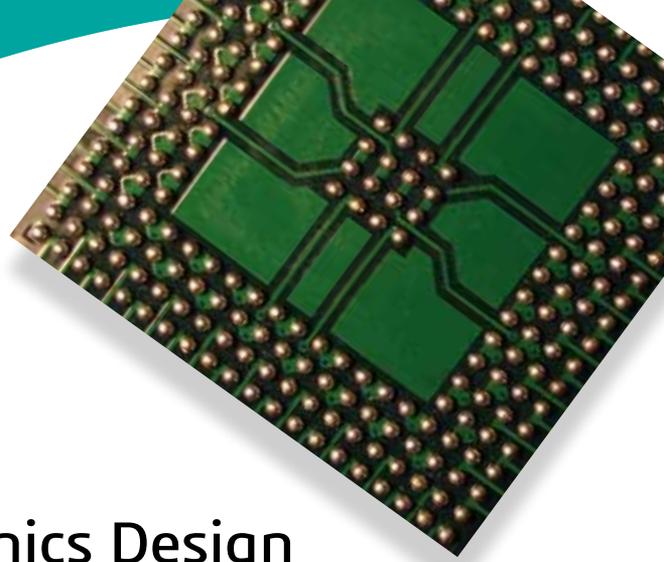
Malaysian Rubber Board—*FEA Modeling of Lateral and Vertical Stresses Distributions in Concrete Blocks Compressed by Rubber Pad*

Medtronic—*Interaction between Short Surface Cracks and Residual Stress Field in Shot Peened Titanium Samples*

Rhombus Consultants Group, Inc.—*Simulation of Composite Fatigue Delamination in a Mixed-Mode Setting*

For More Information

www.simulia.com/XFEM



XFEM Impacts Electronics Design

Auburn University studies shock and drop in circuit board packages

By Pradeep Lall, Mandar Kulkarni, and Arjun Angral
Auburn University, Department of Mechanical Engineering
NSF Center for Advanced Vehicle Electronics (CAVE³)

Rugged, damage-tolerant, and indestructible—this doesn't normally describe how we think of electronic products, but it does describe the way of the future for electronics. Specifically, handheld portable electronics have become more complex and smaller due to advancements in technology. Yet their size can also contribute to their vulnerability during rough use. The loss of functionality due to shock damage during everyday usage can be a source of disruption—from possible loss of time in restoring normalcy to loss of data or valuable information.

In the past, researchers have addressed the drop reliability of electronics at the product level using various experimental and analytical techniques. The transient dynamic behavior of lead-free and leaded solder-interconnects has been studied in ball-grid array (BGA) and

copper-reinforced solder column package architectures using advanced finite element techniques such as cohesive element modeling for both ceramic (CBGA) and plastic (PBGA) packages.

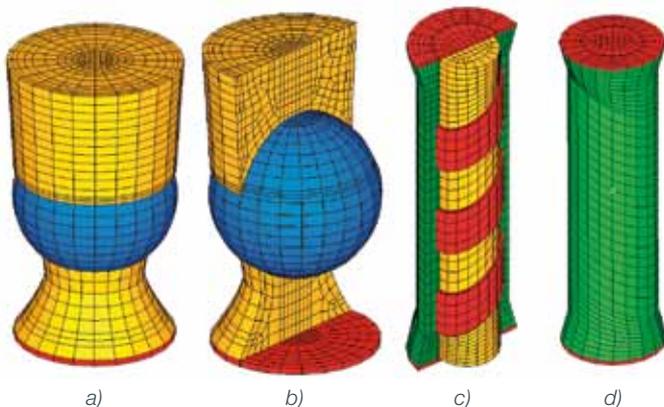
Previous modeling methods used to overcome the length-scale issues between individual interconnects and the package for shock and drop simulation include smeared property models, beam models with conventional and continuum shells, and global-local sub-models. Cohesive element models have been used to study failure at the intermetallic compound (IMC) layer of packages. But now the use of the eXtended Finite Element Method (XFEM) for the study of cracking in solder interconnects under impact loading is being explored.

XFEM was originally introduced to solve problems involving crack growth without the need for re-meshing. The method

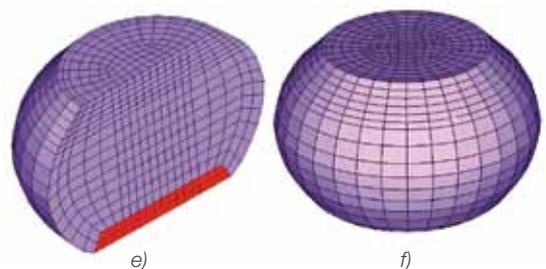
adds a discontinuous enrichment function to regular finite elements to capture the effect of the crack. In classical finite element analysis (FEA), a crack needs to be meshed accurately, and if cohesive elements are used, then the crack can propagate along pre-defined element boundaries. If cohesive elements are not used, then crack propagation would require remeshing at each step, which affects the accuracy of the results and is inefficient.

XFEM provides more realistic crack analysis

By contrast, XFEM allows the crack to grow along an arbitrary, solution-driven path with no remeshing required. For solder interconnects in portable electronics that are subjected to shock and impact, XFEM enrichment functions are added to the elements in all the corner solder interconnects, since these joints are the most vulnerable to failure during a drop.



(a), (b) 90Pb10Sn Solder Joint on CBGA (c), (d) Cu-Reinforced Solder Column on CCGA (e), (f) Sn3Ag0.5Cu Interconnect on CBGA (g), (h) Sn3Ag0.5Cu Interconnect on PBGA324.





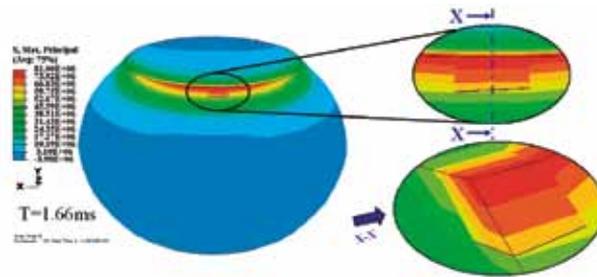
XFEM uses cohesive constitutive relationships to govern debonding, enabling nucleation and the growth of the crack. Instead of embedding a crack tip in the bulk solder, this method automatically introduces a new cohesive segment when the critical cohesive traction is reached. Cracks are introduced as jumps in the displacement fields, with their magnitude governed by the cohesive traction separation constitutive law. Similar constitutive failure laws have been used with cohesive elements to address time-dependent dynamic interface fracture from drop and shock in solder interconnects. The question was—could engineers achieve more accurate simulation by using these cohesive constitutive laws to model bulk solder failure with XFEM?

To find out, researchers at Auburn University performed simulations of drop impacts on BGA and copper-reinforced solder column package architectures using the XFEM capabilities in Abaqus FEA. Results were compared with physical drop tests to validate the analysis method.

How to model a drop

The FEA models for this project reflected the complexity of populated printed circuit boards. Because the corner solder interconnects are highly vulnerable to failure during a drop, refined meshes were required in these areas. XFEM enrichment functions were added to the regular elements to allow for crack nucleation and then propagation, as driven by the solution.

The drop analyses used three separate versions of the explicit global model, reflecting previous methods of representing solder joints in a drop analysis: a beam model, a smeared property model, and a cohesive element model. For all global models, the package was modeled with



Correlation of model predictions with failure modes from experimental data—Sn3Ag0.5Cu CBGA.

C3D8R elements, the PCB was modeled with S4R shell elements, and we used R3D4 elements to model the rigid floor.

For the XFEM sub-model, the corner interconnects were modeled with fully-integrated continuum elements (C3D8), while the remaining solder interconnects were created with beam (B31) elements. The rest of the package, including the chip, mold compound, substrate, and printed board, was modeled with C3D8 elements.

Node-based sub-modeling provided displacement boundary conditions to the XFEM sub-models. The displacements at the driven nodes in the sub-model were defined by output from digital image correlation (DIC) locations on a physical speckle-coated PCB.

The XFEM enrichment functions were only added to areas where previous empirical data suggested that cracking could be expected. For the CCGA and Hi-Pb solders, enrichment was defined for only the eutectic phase at the top and bottom of the corner interconnect columns. For all the other interconnect types, enrichment was defined for the entire corner joints.

Correlation of model predictions with experimental data

Physical board assemblies were cross-sectioned after failure, to observe the failure modes. There was excellent correlation between the Abaqus model predictions and the location of failure modes in board assemblies after physical shock tests. As predicted, copper columns fail close to the package-interconnect interface and Sn3Ag0.5 Cu interconnects on the CBGA package fail close to the package-solder interface, while those on the PBGA package fail near the board interface. 90Pb10Sn interconnects fail close to the board interface in the eutectic solder. The results extracted from the models also showed good correlation, in both

magnitude and phase, with experimental data taken from high-speed measurement of strains from DIC.

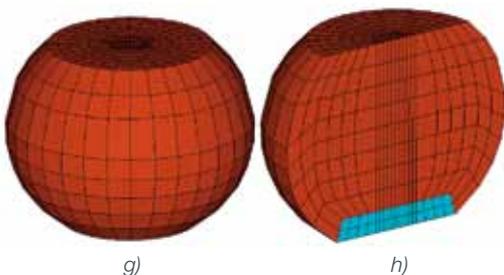
The potential is great for XFEM to increase the accuracy of simulating impacts and cracking in electronic components. That, in turn, will aid engineers to make their products more durable and able to withstand greater impacts.

And that is a big goal for small electronics.

Center for Advanced Vehicle and Extreme Environment Electronics

CAVE³ Electronics Research Center is a National Science Foundation Industry-University Cooperative Research Center at Auburn University. The Center focuses on research related to electronics design, reliability, and prognostics in harsh environment applications such as automobiles, military, defense, and aerospace. Electronics in harsh environments may be subjected to high-g loads, vibration, and very high and low ambient temperatures; extreme temperature changes; moisture and high humidity; exposure to dirt, contaminants, chemicals, and radiation. Typical office electronics may not experience such extreme environments or face the reliability and life-cycle requirements needed for critical applications. These themes provide the motivation for the Center's strategic directions related to technology development and research. The Center is supported by the NSF and member companies.

For More Information
<http://cave.auburn.edu/>



Simulating Fracture with Abaqus

Co-simulation with XFEM technology evaluates safety of hand grenade drop

It's a common enough moment. One person sighs after a lost game or a missed pitch, "It was close," and another will reply, "'Close' only counts in horseshoes and hand grenades."

But 'close' doesn't count at all for designing a hand grenade. The weapon must have proven reliability and predictable performance even when subject to rough handling or accidental dropping. Realistic simulation with finite element analysis (FEA) can be a powerful tool for helping establish those safeguards.

At the U.S. Army Armament Research, Development and Engineering Center (ARDEC) at Picatinny Arsenal in New Jersey, design engineers are constantly exploring ways to refine their use of FEA to verify design strength. They recently evaluated the eXtended Finite Element Method (XFEM) technology in Abaqus. The XFEM enriched environment can be used to look closely at fracture failure (a mode of material breakage under a load), even when the cracks don't follow element boundaries. No matter how finely meshed, a standard FEA model only simulates cracks as they propagate along element boundaries. But XFEM is a powerful tool for accurately modeling "the cracks that fall in the cracks" between elements.

ARDEC engineers were already using Abaqus to model design behavior, since many of their loading scenarios involve highly dynamic, transient events. As such, ARDEC engineers are well versed in the use of Abaqus/Explicit, SIMULIA's best-in-class transient simulation software. However, XFEM works only in Abaqus/Standard, so for



Figure 1. M67 hand grenade

the most complete picture of fracture failure, the designers turned to a technique called co-simulation—simultaneously running two different solvers on the same model. This gave them the best of both worlds: XFEM accuracy in an implicit analysis, and simulation over time in an explicit environment.

To prove out their analysis approach, the engineers simulated a drop test of an M67 fragmentation grenade—the type currently used by the U.S. military (Figure 1). In physical tests, the grenade is dropped in several orientations to ensure safety and functionality (Figure 2).

Looking for a worst-case scenario, the ARDEC engineers ran an explicit analysis and discovered that the highest stress occurred when the grenade hit the ground on the upper corner of the safety handle. Also the material failure model was modified

to simulate the effect of substandard material. The stress concentration in the notched region of the handle, near the safety pin, was selected as an area of interest.

For this analysis, engineers modeled the handle with an elastic-plastic material model using the properties of ASTM A109 Steel (Table 1). All other grenade materials were modeled as linear elastic.

They then formulated a material failure simulation (Table 2) using the maximum principal stress criteria ("Maxps Damage" in Abaqus). Starting the analysis at the beginning of the drop, with a downward velocity of zero and continuous acceleration, would have been complex and time-consuming. It was also unnecessary; during the downward fall, nothing important happens to the grenade. Instead, the engineers modeled the grenade at the moment before impact and used its final velocity. The grenade hit with a damage energy of 5.3 kiloNewtons/meter (almost 1200 lb./meter) after being dropped from a height of 1.219 meters—roughly the height of an object held at arm's length. The grenade assembly was deformable, but the impact surface was rigid and constrained in all directions.

To ensure accuracy, the designers meshed the handle model very finely (four elements thick) and carefully lined up the mesh in the implicit and explicit boundary regions. The notched area on the handle (where fracture occurred) was the XFEM enriched zone. It was kept small to eliminate the possibility of XFEM elements near the co-simulation boundary.

Yield Strength, MPa	305.3
Ultimate Tensile Strength, MPa	437.7
Young's Modulus, GPa	204.8
Strain at Ultimate Failure, percent	18.0
Poisson's ratio	0.29
Density kg/m ³	7,823.0

Table 1. Material Properties, M67 Handle

ASTM A109 Steel

Maxps Damage	Maximum Principal Stress: 345 MPa
Damage Evolution	Type: Energy
	Softening: Linear
	Degradation: Maximum
	Mixed mode behavior: Mode-Independent
	Mode mix ratio: Energy
	Fracture Energy: 5.3 kN/m

Table 2. Material Failure Model

The run time for the co-simulation was dramatically reduced from the implicit dynamic analysis: 10 minutes for co-simulation versus 6 hours for the implicit analysis.

Three analyses were used to validate the co-simulation method (Figures 2-3):

- In the first, Abaqus/Standard and XFEM were used to analyze a continuously meshed model of the handle.
- The second split the model in two, running dynamic analyses in Abaqus/Standard with tied regions to an enhanced XFEM region. The two-part structure would later facilitate co-simulation.
- The final analysis was the co-simulated drop test. Most of the model was analyzed in Abaqus/Explicit. The area of interest on the handle was still modeled in implicit so that the XFEM technique could be applied.

The run time for the co-simulation was dramatically reduced from the implicit dynamic analysis: 10 minutes for co-simulation versus 6 hours for the implicit analysis.

Afterward, the ARDEC engineers compared simulation results by plotting averages for eight elements in the area of concern on the grenade handle. Charts of the averaged results for plastic equivalent strain (Peeq) and Von Mises stress showed that the stress values between the tied implicit model and the co-simulation model matched well. Crack growth in the second and third analyses was similar as well.

Subsequently, the engineers were able to fine-tune the co-simulation analysis in a number of ways. They used matching meshes at the co-simulation boundaries because, if the nodes were not nearly coincident, no loads would be applied to them. Starting both analyses with the same initial time step improved the convergence of the implicit XFEM analysis. And keeping the XFEM enriched area away from the co-simulation interaction boundary also promoted convergence and prevented cracks from starting right at the boundary.

Co-simulation enabled the ARDEC engineers to retain their preferred explicit

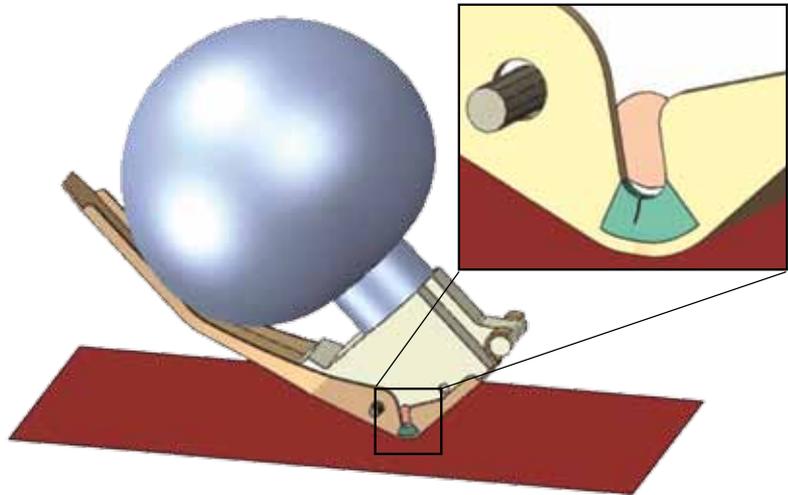


Figure 2. Abaqus simulation of a drop test showing notched area of the safety handle (in green), analyzed using XFEM technique and showing crack formation

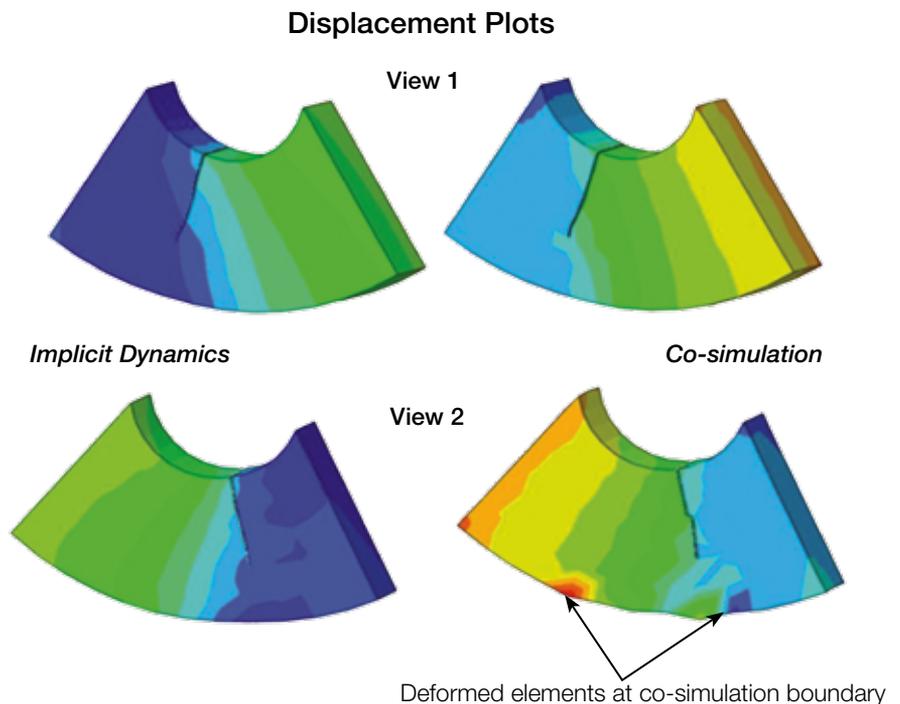


Figure 3. Side-by-side comparisons of the implicit dynamics and the co-simulation analyses, showing the areas of element deformation at the co-simulation boundary.

environment for transient analyses and augment it with the fracture failure capabilities of XFEM in implicit. The significant run-time savings will help ARDEC continue to explore the potential of using co-simulation in the future. In addition to simulating drop tests with XFEM, ARDEC engineers are interested in using it for concrete penetration, gun launch, recoil, and interaction of gun supports with the ground.

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

Improving Our Understanding of Fracture and Failure

Zhen-zhong Du, Technology Lead—Fracture & Failure Technology

Today, simulation has become an integral business practice across a wide spectrum of industries, whether you are building an airplane made of composites, designing a small hand-held electronic component, or developing a medical device. Predicting and designing against the failure of such products by fracture or fatigue is an essential, yet challenging part of the design and simulation process.

As SIMULIA's technology leader for fracture & failure technology, my professional involvement in fracture mechanics and Abaqus can be traced back nearly 25 years. I first used Abaqus (Version 4) as a graduate student working for my Ph.D. on fracture of tubular joints in an offshore structure at the University of Glasgow, Scotland. I joined SIMULIA in 1996 and have been focused on our fracture and failure technology from the beginning. When I started, fracture and failure simulation was performed only by the researchers. It required very deep knowledge of fracture mechanics and numerical methods and was used by only a small subset of the Abaqus customer base.

Over the past 15 years, I have been one of the key contributors within our team of talented developers on advancing the fracture and failure technology in Abaqus. We have a well-defined R&D strategy in place to help our customers achieve their failure analysis objectives. Our long-term goal is to make the modeling of fracture and failure as common and as easy as including the effects of plasticity.

Customer-driven— industry-focused capabilities

To provide the best solutions to our customers, it is crucial that we clearly understand their fracture and failure requirements. To this end, we formed the SIMULIA Fracture Customer Review Team (FCRT) in 2003, which is composed of leading industry and academic experts. The FCRT meets annually with our developers and product managers to help

identify and prioritize the development of new capabilities and validate the new functionality.

The input from this team, as well as other customers, has led directly to the release of substantial new fracture capabilities, including cohesive elements, Virtual Crack Closure Technique (VCCT), material damage models, J-integral contour modeling, low cycle fatigue with direct cyclic, and many others over the past eight years, beginning with the Abaqus 6.4 release. Additionally, a significant number

of innovative capabilities were developed through customer-requested enhancements and direct engagements with our customers and partners. One such partnership, with Boeing Commercial Aircraft Group, enabled us to deliver the VCCT technique within Abaqus/Standard in version 6.5. This technology has made a positive impact on the use of composite materials in fabricated structures today.

Another R&D philosophy we've adopted is to pay close attention to applications in other industries when working with

Formed in 2003, SIMULIA's Fracture Customer Review Team is composed of 62 leading industry and academic experts (12 are shown below).



Michel Fouinneteau,
AIRBUS



Stephen Hallet,
Bristol University



Yao Yao,
ExxonMobil



Scott Finn,
GE Global Research



Silvestre Pinho,
Imperial College
of London



DM Hoyt,
NSE Composites



Thomas Siegmund,
Purdue University



Ronald Krueger,
National Institute
of Aerospace



Tom Walker,
NSE Composites

customers on a primary industry application. One example is a cost-shared development project with PSA Peugeot Citroën to develop direct cyclic analysis technique on a cylinder head and exhaust manifold in the Automotive industry. The direct cyclic solution method is a unique capability in Abaqus to automatically calculate the stabilized, steady-state response of a structure to cyclic loading, thus increasing solution accuracy and reducing solution time compared to traditional methods. The direct cyclic analysis technique has also found many applications in Aerospace, High-Tech, and Life Science industries. In two IMECE papers, published in 2006 and 2007 respectively, Intel engineers described how they applied the direct cyclic solution

method in Abaqus for thermal fatigue analysis of electronic packages.

By working closely with our customers and being cognizant of different industry needs, we are able to understand the various industry-specific processes and simulation requirements and align our development efforts towards solving real engineering problems.

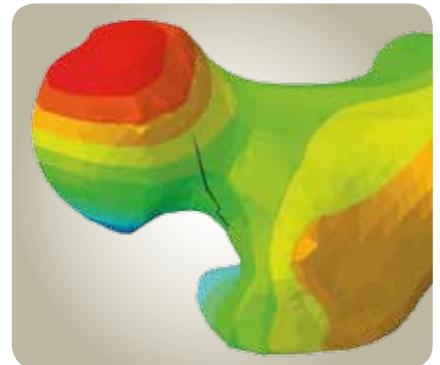
Innovative technology for fracture & failure

In May 2009, the Abaqus 6.9 release continued to deliver our strategy with the introduction of a framework for applying the eXtended Finite Element Method (XFEM). This technology is an extension of the conventional finite element method based on the concept of partition of unity, which allows local enrichment functions to be easily incorporated into a finite element approximation. The presence of discontinuities is handled by the special enrichment functions in conjunction with additional degrees of freedom. A distinguishing feature of XFEM is that the model geometry does not have to conform to the crack—this leads to significant simplifications, especially when it comes to creating a model and simulation of crack propagation. In other words, with XFEM cracks can propagate in any direction across a mesh, *even through existing elements*.

XFEM, as a formalized approach, has been in the technical literature since the first paper was published by Belytschko and Black in 1999. It has gained acceptance in the academic world as a viable approach to modeling fracture. However, the state-of-the-art ideas found in the open literature are not general or robust enough to be used in a high-quality commercial product such as Abaqus. Many technical and architectural issues needed to be resolved by the SIMULIA XFEM development team. More importantly, Abaqus is a general purpose code and developers have to write a method that can be used for a multitude of physical problems. The challenge is perhaps at least as big as coming up with an idea that works for only a few specific problems in academia. It has to be robust, accurate, and easy-to-use. SIMULIA has recognized that completely developing the XFEM capability is a multi-year development team effort and has invested significant R&D resources into this endeavor over the years.

Since XFEM was first included in the Abaqus 6.9 release, we have rapidly added and enhanced the functionalities to support our customer's growing interest in the solution. Such improvements include support for:

- Contour integral evaluations for an arbitrary stationary crack
- The implicit dynamic option for transient analysis
- The Linear Elastic Fracture Mechanics (LEFM) approach to complement the cohesive segments approach
- Low cycle fatigue analysis within the framework of direct cyclic analysis and user-defined damage initiation criteria
- Second-order tetrahedron elements to go mainstream with XFEM and contour integral evaluations with residual stress field.



Simulated crack formation along a femur neck during an in-vitro impact test modeled using XFEM. Courtesy of Mayo Clinic.

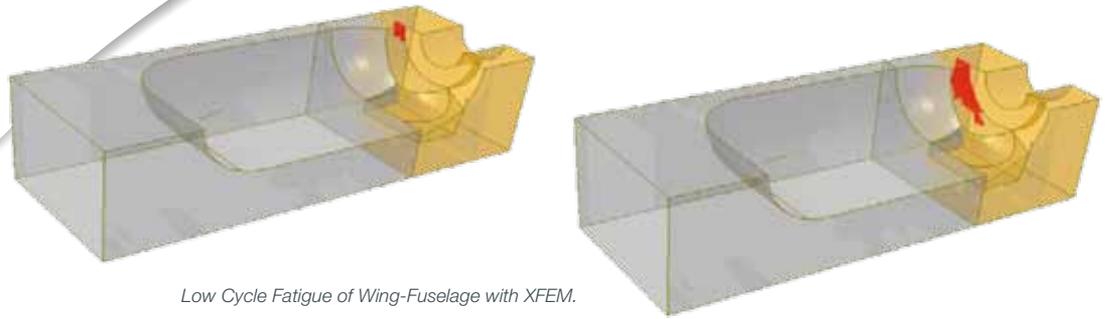
Measuring success

At the 2011 SIMULIA Customer Conference (SCC), I moderated the Special Interest Group (SIG) on fracture and failure which was well-received by approximately 60 customers. The goals of the SIG are to interact more closely with attendees on industry-focused topics, provide value through close interaction with SIMULIA R&D, provide a review of our historical progress, current status, and a peek at our future strategy, and to allow feedback on our products, company, and strategy. Our customers are very energized by this topic and the fact that we have made excellent progress on delivering innovative applications of the XFEM technology in each release (with Abaqus 6.11 being the fifth release).

Continued



Strategy Overview



Low Cycle Fatigue of Wing-Fuselage with XFEM.



In this pressure vessel model, XFEM in Abaqus/Standard allows for the prediction of arbitrary, solution-dependent crack growth independent of the finite element mesh.

Our customers are particularly happy that our fracture and failure development is guided by the FCRT and some of them have expressed great interest in joining the team. At the same time, we greatly appreciate the feedback and the enhancement requests made during the Q&A.

The SCC provides an excellent forum to directly interact with many customers about fracture and failure. Out of 95 customer presentations, there were seven customer papers on XFEM, in addition to a half dozen on fracture and failure with conventional finite elements. These papers were on applications across a wide spectrum of industries, including Industrial Equipment (Tenaris Dalmine on pipe rupture), Energy (TÜV NORD on fracture of pressure vessel due to operational thermal loads using XFEM), Defense (U.S. Army-ARDEC on fracture of equipment, such as a hand grenade drop test or gun breech, using co-simulation in conjunction with XFEM), and Life Sciences (University of Iowa on ceramic total hip bearing fracture using XFEM). It is always good to see our customers putting Abaqus XFEM technology into action! It's evident that the

efforts by the XFEM team over the past four years have certainly paid off.

Future endeavors with fracture & failure technology

The FCRT will continue to help SIMULIA align our strategy in fracture/failure/fatigue simulation with real industry needs, fill in any missing functionality in the first generation of enhancements, and prioritize work on these enhancements. We are fully committed to developing complete and robust XFEM functionality across a wide section of our customer base in all industries. Our customers are expected to see continued developments focused in the area of multiphysics simulation, as well as improvements in ease-of-use. Significant challenges remain to deploy the XFEM method to non-experts and to integrate this technology into the next generation of products built on Dassault Systèmes Version 6 platforms. We are proud of what the FCRT has accomplished so far and excited about where XFEM, and all the other fracture and failure technology we have today, will lead us in the coming years.



Zhen-zhong Du,
Technology lead
for fracture & failure
technology

Zhen-zhong's professional involvement in fracture mechanics and Abaqus can be traced back nearly 25 years. He first used Abaqus as a graduate student working for his PhD on fracture mechanics at the University of Glasgow, Scotland. Working with his advisor, he co-authored a paper on constraint effects that was published in the *Journal of Mechanics and Physics of Solids** in 1991. He then worked at Cambridge University and at the University of California at Santa Barbara prior to joining SIMULIA in 1996. He has been one of the key contributors within SIMULIA's talented team of developers on advancing the fracture and failure technology and other mechanics technology in Abaqus.

*Z-Z Du and J.W. Hancock, "The effect of non-singular stresses on crack-tip constraint", *J. Mech. Phys. Solids*, Vol. 39, No. 4, pp. 555-567, 1991

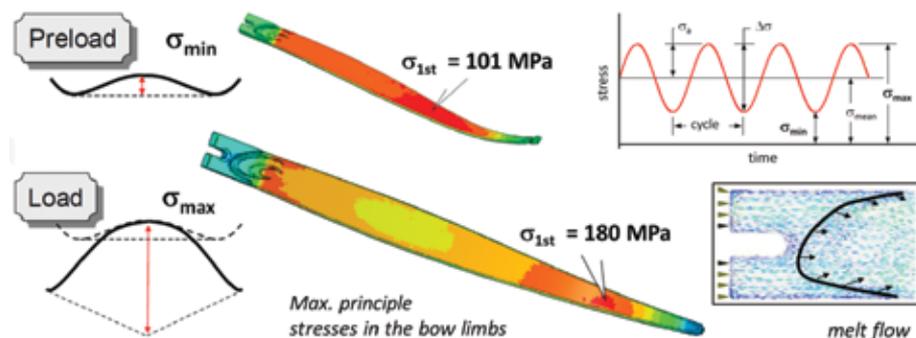
For More Information
www.simulia.com/XFEM

Fatigue and Failure of Short Fiber Reinforced Plastics

The primary goals of designing reliable product parts are to be able to predict performance and lifetime duration. The development of composite concepts for short fiber reinforced plastics poses new challenges that differ from the design of metal parts due to the direct influence of composite material properties, which affects performance prediction and fatigue behavior.

Solvay Specialty Polymers, a material supplier headquartered in Belgium, uses DIGIMAT, e-Xstream's nonlinear multi-scale material and structure modeling platform, with Abaqus to support their customers in the investigation of new composite concepts. One of Solvay Specialty Polymers' customers is investigating the use of Solvay Specialty Polymers materials to produce the best recreational archery bows on the market. However, upon testing a new and apparently improved design of the bow limbs, it turned out that the design showed a reduced fatigue lifetime compared to the existing geometry. Was this premature failure directly connected to the change in the processing of the bow limbs?

Solvay Specialty Polymers used the Digimat-CAE interface for Abaqus, with fiber orientation data coming from Moldflow, to compute first principles stresses for the



Prediction of lifetime for injection molded bow limbs. Two load cases are taken into account, the preload due to the bow string and the maximum load when performing archery. The fiber orientation of the 50% GF SOLVAY material is taken into account by performing computations of the 1st principle stress via the Digimat-CAE/Abaqus interface based on Moldflow results. The fiber orientation in the limb is affected by two converging flows in the melt.

preload and the load case of the bow. The bow's lifetime was then predicted based on a Haigh diagram and it was found that the fatigue behavior of the original design was superior to that of the new design. The Abaqus analysis of the new design predicted higher stresses based on the fiber orientations. The higher local stiffness in the critical region of the bow limb could be explained by a higher orientation of the fibers due to a change in the melt flow in the processing step.

"By using DIGIMAT, we could clearly demonstrate the tight link between geometry, flow of material, fiber orientation and mechanical behavior," stated Laurent

Hazard, CAE senior specialist at Solvay Specialty Polymers. "We successfully used this approach to evaluate the endurance of a number of different bow limb designs."

In the upcoming DIGIMAT release, Solvay Specialty Polymers will be able to account for the anisotropy of SN data, which is a function of the local fiber orientation. Future DIGIMAT releases will facilitate the lifetime prediction of short fiber reinforced plastics with new interfaces for LMS, FEMFAT and NCODE.

For More Information

www.e-Xstream.com

Component Central Enhanced

Peruse the listing of Isight components created by SIMULIA Component Integration Program partners

SIMULIA Component Central provides Isight users with a single-source directory to view, understand, and download SIMULIA's Isight components and plug-ins. Now with an enhanced partner components section, users are able to quickly view partner-developed, value-added components and plug-ins.

Partner components and plug-ins further extend SIMULIA's library of components and solutions for the rapid creation of powerful, customizable workflows within Isight that incorporate many widely utilized simulation tools. Could you leverage any of these components to streamline your simulation environment?

See how you can get best fit models automatically from model modification

and analysis using the Isight/SolidWorks component from Myrtos Corp., study optimization problems which have a huge number of variables using the intelligent Design Advisor (iDA) stochastic optimization component from Exemplar, or seamlessly incorporate and exchange data with Safe Technology's fe-safe™, AVL's EXCITE, or CD-adapco's STAR-CCM+ among other tools.

Partner components are distributed and supported by CIP partners. If you're interested in joining the CIP or would like to explore the possibility of making your component more broadly accessible to Isight users, please contact us at alliances@3ds.com.



For More Information

<http://components.simulia.com/partner>

Speeding the Route to Quality

Rolls-Royce uses Isight for simulation process automation and design optimization of aircraft engine turbomachinery

Designing a jet engine is one of the most difficult engineering challenges there is. The intake, fan, compressor, combustion chamber, turbines and exhaust must all operate in tandem throughout a vast range of altitude, weather and temperature conditions. Further complicating the design task is that important, invisible player: aerodynamics. How do you get power efficiently out of the system by controlling the speed and direction of the air that moves through such a complex structure?

The engineers at Rolls-Royce, whose core product is the gas turbine engine, understand the intricacies of this challenge well. The company is a world-leading provider of power systems and has established a strong position in global markets—civil aerospace, defense aerospace, marine and energy. In the civil arena alone, their engines can be found on 30 types of commercial aircraft, with more than 13,000 engines in service for 650 airlines. A Rolls-Royce powered aircraft takes off or lands every 2.5 seconds.

While it once took up to ten years to develop a new aircraft engine, the industry average has now shrunk to about two. And Rolls-Royce is working to condense that time even further. “Our customers expect consistent performance, fuel-efficiency, and short delivery cycles,” says Dierk Otto, design systems engineer at Rolls-Royce

Dierk Otto,
Design Systems Engineer,
Rolls-Royce



We have noticeably less redesign work now, which leads to better control over manufacturing costs.

Deutschland Ltd & Co KG. “On our side of the equation we are looking to ensure quality and reliability while keeping design, manufacturing and maintenance costs low.”

The solution for Rolls-Royce has been its “robust design” program which emphasizes the leading role of design as the entry point into the company’s Six Sigma program. This focus on the impact of early design on quality has led Rolls-Royce to employ powerful engineering resources from the fields of CAD, CAE, CFD, FEA and more, in the pursuit of optimum product performance. “We are continuously evaluating new design concepts by integrating analysis tools from different disciplines,” says Otto. “Our experts

and teams rely heavily on Isight software for process integration and automation, as well as optimization, to accelerate problem solving in this complex design environment.”

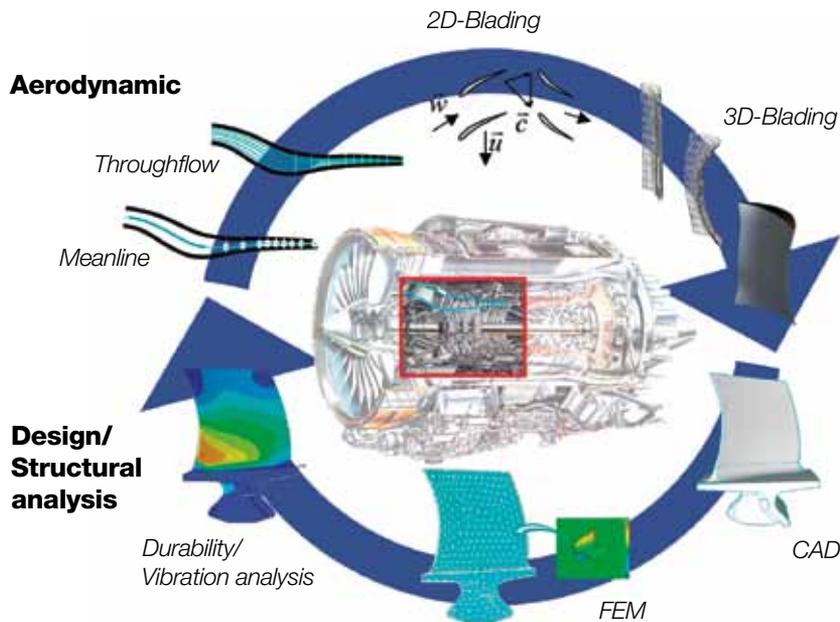
Isight was introduced at Rolls-Royce more than a decade ago and is now employed throughout the company. The software’s drag-and-drop capability for creating simulation process flows (known internally at Rolls-Royce as ‘workflows’) lets engineers link—and automate—all the steps in a particular design process, allowing the simultaneous integration of multidisciplinary simulations (like aerodynamics plus stress plus weight plus cost). Design exploration with DOE or Monte Carlo methods or a variety of powerful optimization techniques (like genetic, gradient-based, or deterministic algorithms) can be performed automatically, and then linked back into an ongoing workflow. Any number of design parameters and analysis types can be included in the workflow, and the design can be further optimized through material trade-off studies, tolerance reviews and manufacturability assessments.

Robust design of an engine compressor

As an example of the Isight toolkit in action, consider the design of an aircraft engine compressor. It all begins with the aforementioned challenge of aerodynamics. A large aircraft wing and an individual engine blade are actually both airfoils—their shape and orientation affect the direction, lift and thrust of the air that passes by them. But a wing is a passive airfoil, while a blade is an active part of the propulsion system. Air entering through the engine fan into the low-, intermediate- (in a three-shaft engine), and then high-pressure compressors is pulled through a gauntlet of hundreds of blades—some spinning (rotors), some not (stators). The airfoil cross-section, number and layout of these blades are determined by how much pressure is required at each stage of the compressor.

The first compressor task is meanline prediction. Starting from an existing design, the engineer must find the optimum form of the annulus—the donut-shaped area of rotating “blade exits” through which the air flows—for the desired new configuration. The meanline is calculated halfway between the hub (base of the blades) and the tip (outer end of blades) of the annulus. The total area of the annulus determines exit velocity and pressure rise for each stage. This is the principal determinant of the size and cross-sectional layout of the compressor, how many stages it should have and what





The design iteration loop for an aircraft engine compressor blade. Process automation and optimization with Isight can be applied at any stage of the cycle.

the inner and outer diameters of each stage will be.

At this point the design engineer is not thinking about blade shape; he is only conducting a thermodynamic and fluid flow exercise to determine the overall flow of air that results from different diameters of annulus at different compressor stages. Yet it's a very complex exercise already: Aerodynamic parameters that have to be taken into account in meanline prediction include pressure ratios, efficiency, surge margins, form factors and so forth. Running these analyses manually would be immensely time-consuming. But by using Isight, the engineer can integrate all the meanline tasks into an automatic process flow that works through each task sequentially, evaluating the data and applying any relevant external programs, to arrive at the optimum solution.

Isight components decrease design process complexity

Although the typical Isight user at Rolls-Royce is an expert who can set up such complex simulation flows quickly, not everyone on the design team needs to work at these deepest levels of expertise. For example, a CFD analyst might want to use an Isight workflow of a meanline prediction to measure the feasibility and consequences of his own design ideas and changes. To support these more casual users of Isight, Otto and his colleagues created customized Isight components of calculation routines, data mapping, program controls and bundles, and dynamic link libraries. The casual users can

now call upon whatever component they need and drop it into their own simulation flows to speed up and simplify their work, generating the same solutions that an expert would without having to work through all the subroutines.

"This user-friendly component approach gives our team a number of advantages," says Otto. "Information can be shared easily—even with worksites in other locations—and we can standardize our process build-ups more readily as well as speed up our runtimes."

Multidisciplinary optimization drives design

The next step in the development of the engine compressor requires the engineering team to move from one-dimensional meanline analysis to throughflow (optimizing streamlines of predicted air movement inside the annulus). The final step is actual blade design. Starting with two-dimensional blade geometries (profile sections) that produce the flow angles and conditions predicted by the meanline and throughflow analyses, designers use CFD solvers within Isight to automatically optimize the geometry of every single blade, the cross-section profile of which is the familiar airfoil shape.

By stacking multiple airfoil profile sections on top of each other, connecting them with linear filaments, adjusting the lean of the resulting structure and giving it a root and a platform to sit on, the engineers arrive at the first 3D shape of a blade. Using an in-house Rolls-Royce blade generator tool, these blade profiles can be modified via

different design parameters like maximum thickness, blade angles, camber style, etc. The goal is to find the airfoil design that best meets performance requirements and structural criteria as well as the previously determined, all-important aerodynamics.

In aircraft engines, lighter weight is always at a premium. Arriving at a CAD model of a blade where the geometry and orientation have been optimized for weight, the designers still need to prove that the blade will survive under real-world conditions. This is where static and dynamic (FEA) are brought into the optimization loop to perform stress analysis, study vibration and resonance behavior, and examine material creep and lifetime wear. When critical issues are identified, the engineers can modify the sections and rerun the stress analysis. Here Isight is an integral tool for accelerating the identification of the best design solution.

This automatic blading design process has a great deal of external input-output data flow, so selecting the correct design parameters is very important to avoid error. Taking a page from their experience with creating custom components for meanline prediction, the Rolls-Royce team decided to create a series of templates for blading as well. Again, the casual user (perhaps an FEA analyst this time) can now select just the required design parameters, set upper and lower boundaries for the desired parameters, and include DOE or optimization runs to refine constraints against performance goals. "While templates help speed up our process build-ups and reduce error-sources, they are also designed to preserve process flexibility," says Otto.

Evaluating more design options—faster

Using components and templates together in a single design iteration loop—modify airfoil, run 2D CFD calculations, do 3D optimization with FEA, and finally evaluate results—now takes only 13 minutes, whereas previously it would need about a day. "We have noticeably less redesign work now, which leads to better control over manufacturing costs," says Otto. "Overall this allows faster response to our customers' demands while adhering to the strictest quality standards of the aircraft industry."

For More Information

www.rolls-royce.com
www.simulia.com/cust_ref

Fire, Fasteners, and Finite Element Analysis

Structural engineers use Abaqus to investigate fastener strength and help satisfy stringent fire safety roofing standards

From the first sign of smoke or scream of sirens as a building goes up in flames, there may only be minutes that separate a safe evacuation from a more serious scenario. The physical composition of construction materials and the building methods used to erect, fasten, and anchor the components are all factors in determining this time-safety window.

Building codes in the European Union require that roofing systems remain structurally sound for 15 minutes when a fire breaks out to give occupants time to evacuate. For example, commercial, industrial, and residential sheet metal roofs—popular in Scandinavia for their snow-shedding properties—must be able to withstand normal weights and loads at temperatures as high as 600°C (approximately 1100°F), close to the temperature at which aluminum starts to melt. Critical to their structural integrity is the performance of the powder-actuated fasteners (PAFs) commonly used to connect and install these roofing systems.

PAFs are basically nails, of numerous shapes and sizes, made of high-quality hardened steel with a tensile strength four to five times that of the base material. Using a small explosive charge, they are “shot” like a bullet from a nail gun at 600 meters per second—a speed and force powerful enough to drive the fastener into the building’s steel or concrete superstructure—displacing the substrate material and anchoring the fastener.

First developed for underwater ship repair in 1915, PAFs are still widely used today in the shipbuilding industry and also in roof and composite floor-frame-system construction. A simple low-tech device, surprisingly similar to their century old precursors, PAFs are the key to keeping modern metal roofs structurally secure under any conditions—including fire.

Putting PAFs under the finite element analysis microscope

While PAFs are relatively simple, much is still unknown about the characteristics and installation methods that contribute to their strength and durability. For instance, numerous tests have been conducted on pull-out forces, but there has been very little research on shear resistance, which can be important in scenarios such as fire. As a result, current design recommendations are based primarily on the information readily available for bolts and rivets.

To advance the knowledge of structural and mechanical engineers in the construction industry, a group of engineering researchers in Finland collaborated to study the load-bearing and lap-shear behavior of PAFs under ambient and fire conditions. The group included: Ruukki Construction (a major supplier of sheet metal and load-bearing steel sheets to the construction industry), Aalto University’s Department of Civil and Structural Engineering (a group involved in research on the fire-engineering design of steel structures), and Z.Ma Research and Consulting (a structures and strength consultant specializing in using complex simulation technologies).

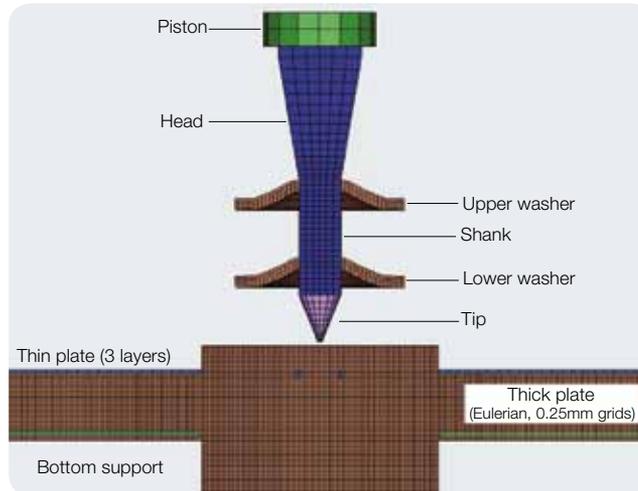


Figure 1. Abaqus finite element model of a typical power-actuated fastener (PAF) showing the structure of the nail and the sheet-metal roofing system.

For a deeper understanding of the real-world physics behind the seemingly simple PAF lifecycle—from nail driving, to extreme heating, to shear loading—Dr. Ma and the Aalto University research team chose Abaqus. “Abaqus provides very good functionality for solving a highly nonlinear problem under complex dynamic, transient dynamic, and thermal loading,” says Dr. Ma.

The FEA models for the PAF consisted of a thin plate on top (equivalent to the sheet metal roofing), a thicker plate on the bottom (corresponding to the building’s steel substrate), and a standard fastener with its three-part, head-shank-tip shaft structure separated by two washers (Figure 1). The thick plate

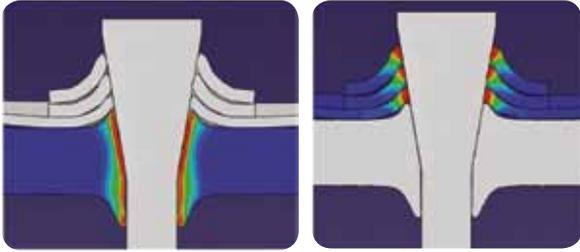


Figure 2. These Abaqus FEA models illustrate the plasticity in the thin plate and washers (left) and the equivalent plastic strain in the thick plate (right) after the fastener has been installed. Note that the red color shows the regions where the equivalent plastic strains are greater than 0.45, indicating significant amounts of plastic deformation.

was modeled using an Eulerian, or fixed, frame of reference (EC3D8R elements), while the thin plate and fastener used the Lagrangian or moving frame (C3D8R elements).

Relying on Abaqus capabilities, Dr. Ma divided the PAF analysis into a sequence of separate steps that examined the strength of the nail for all phases of the installation process including before driving, after driving, and after “springback” (a step that takes into account the elastic rebound of the materials and the “welding” phenomenon between the nail shank and the thick plate). He also used the model to simulate the residual stresses, deformation, and plasticity in the PAF, thick plate, and thin plate (Figure 2), the heating of the fastener-plate samples to increasingly hotter fire temperatures (200, 400, and 600°C), and the addition of lap-shear displacement loads to analyze the load-bearing characteristics of the fastener assembly.

“The Coupled Eulerian-Lagrangian formulation in Abaqus/Explicit was especially well suited for such an analysis since it can account for material nonlinearity, contact nonlinearity, and progressive failure prediction while still handling significant deformations,” says Dr. Ma. “Abaqus was able to carry out the multi-step simulation efficiently with its parallel computing capabilities.”

Simulation correlates well with experimental testing results

For the thin plate, the simulation identified the four major components responsible for the shear resistance load: the bearing force between the thin plate and nail shank; the frictional force between the washer and the thin plate; the frictional force between the thin and thick plates; and the bearing force of the two washers due to the tilting of the thin plate as a load is applied.

The simulation also showed that as the fastener penetrated the substrates, the material of the thick plate flowed up and outward around the nail, forming a protuberance feature that grew in height as the nail went deeper. When the nail shaft reached the qualified installation stand-off height of 8-11 mm above the surface, all layers of the sample (thin plate, thick plate, and washers) contacted each other tightly and the protuberance was its tallest (1.5 mm high at an 8 mm stand-off height) (Figure 3).

The study determined that at qualified stand-off heights, the protuberance feature provided a 30- to 60-percent increase in the PAF’s load bearing resistance: basically, the deeper the nail is installed (within specification), the taller the protuberance becomes

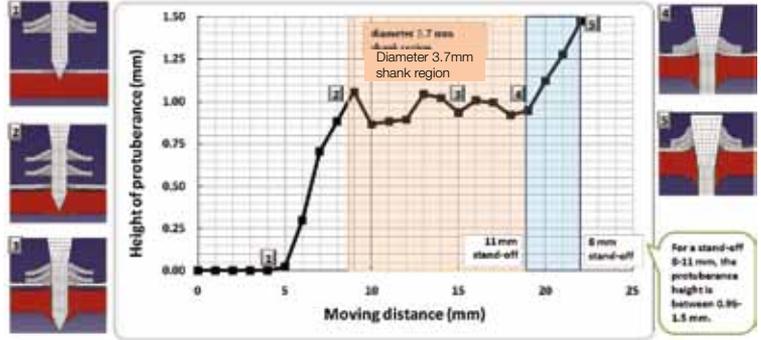


Figure 3. This graph shows the height of the protuberance feature as related to the depth of fastener penetration during the nail driving process. Between 11 and 8 mm stand-off (the height of the nail above the substrate qualified by the manufacturer for proper installation) the protuberance reaches its maximum height.

and the more durable the connection is at ambient and high-heat conditions when subject to lap-shear loading.

To validate the studies, the team at Aalto University conducted physical testing on a series of specimens representative of a typical cold-formed sheet metal roof fastened with PAFs and identical to those used in the computer model. Comparisons of Abaqus FEA results with measured load-displacement at both ambient and elevated temperatures showed good correlation for the assembly.

In the past, physical testing was the only way to evaluate the effectiveness of PAFs. Now with finite element modeling and analysis methods in Abaqus, designs of fasteners as well as best practices and specifications for installation can be evaluated outside the lab, before reaching the construction site.

“Using finite element analysis, we were able to determine the contribution of each factor in PAF assembly and roof strength,” says Dr. Ma. “The results of our study may indicate that fewer fasteners can be used to keep roofs structurally sound.”

That’s a finding that should get the attention of anyone responsible for keeping the roof over our heads safe, even when there is a fire.

Profile/Bio



Zhongcheng Ma is a senior consultant in product and civil structures under complex environmental loads. He has a Ph.D. in structural mechanics from Harbin Institute of Technology (China) and a D.Sc. in technology, specializing in steel structures, from Helsinki University of Technology (Finland). He has used Abaqus for simulations involving impact, crush, drop, fire, and earthquakes.

For More Information
www.simulia.com/XFEM

Helping Power Providers Manage Energy Ups and Downs

TÜV NORD uses Abaqus to predict fatigue and fracture and to optimize power plant cycling and operations



An aging chemical manufacturing process vessel, with cracks that had developed from extreme temperature shifts, is removed and analyzed by engineers from TÜV NORD.

Conventional coal and gas power plants were not designed to navigate the production peaks and valleys of today's energy landscape. Originally built to produce continuous power, they are increasingly being used as backup to supplement sustainable forms of energy, such as wind and solar, which are playing a more important role in the power mix. But since the wind doesn't always blow and the sun doesn't always shine, energy providers need to constantly toggle steam generating plants from off to on and back again.

Such starts and stops are quickly becoming the new standard for energy companies. Operating under these conditions means greater stresses on the entire power plant, but especially on the thick-walled, main steam-line components through which steam is transported from the pressure vessel to the turbine and where the greatest temperature shifts occur. Plant start-ups, when most extreme, can involve a temperature increase in these components of almost 500 degrees centigrade in an hour. A shut-down is almost identical, but in reverse. To ensure that overall power generation remains constant enough to meet varying demand, the number of start-stop cycles will rapidly increase in the future.

With frequent steep temperature cycling, system components—such as the vessel, valves, and header—are subject to fatigue and fracture. As a result, their lifespan

may be shortened, negatively affecting operational efficiency, system maintenance, and energy production.

For Axel Schulz, the coordinating engineer on the stress calculation and design team at TÜV NORD SysTec in Hamburg, Germany, evaluating and solving the engineering challenges of temperature cycling for the power and chemical industries is a primary focus. These projects have taken on even more importance recently following the passage of a new energy program in Germany.

"The German government has established an ambitious plan to replace nuclear energy with wind and solar power by 2022," said Schulz. "To compensate for the fluctuating power production of the newer technologies, we have a critical need for fast-cycling power plants." As the energy industry gradually becomes greener, it's likely that this trend will be repeated worldwide.

FEA guides improvement of power plant operations

In response to such changing conditions, TÜV NORD created a Cycle Optimized Operation (COOP) program designed to improve facility standards, performance, and management. "When evaluation and testing closely reflect reality," said Schulz, "plant operation and cycling can be made much more efficient and costs can be controlled."

Realistic simulation with Abaqus finite element analysis (FEA) software from Dassault Systèmes' SIMULIA brand was chosen as the core solution for the COOP concept. "Abaqus has very sophisticated capabilities for analyzing fluid structure interaction, fracture, and fatigue," said Schulz. "It also has advanced tools for calculating crack propagation."

German regulations for power plant operation are not only strict, they are extremely conservative and based on over-simplified assumptions: currently, start-up and shut-down processes are calculated as approximately quasi-static operating procedures, and heat transfer coefficients are not used (the steam temperature is set equal to the wall temperature). Due to these simplifications, according to Schulz, fatigue and damage to parts and components can be significantly overestimated.

In the first module of the COOP's two-part process, TÜV NORD engineers use Abaqus fluid structure interaction (FSI) capabilities to simulate real-world heat transfer conditions within plant components. This results in more realistic assessments of component stresses and less conservative design codes.

In the COOP's second module, the Abaqus FSI simulations enable optimization of the plant component's interior shape to reduce pressure losses, stresses, and noise emissions. This module also uses eXtended Finite Element Methods (XFEM) and crack propagation analyses—both native to Abaqus—to calculate crack geometry and growth. These analyses allow the engineering team to determine new safety standards and more accurately evaluate service life.

"Using simulation, we can recommend better regulations and suggest more accurate inspection intervals, eliminating unplanned maintenance and costly repairs," said Schulz.

Putting the COOP program into action

To evaluate the COOP program methodology, the TÜV NORD team carried out an analysis of two aging chemical process vessels (components used in chemical manufacturing that also undergo frequent cycling and extreme temperature shifts). "We chose to study these vessels because we provide services to the chemical industry, and these process vessels have operating conditions that are a good match for main steam-line components in fast-cycling power plants,"

said Schulz. The vessels selected for the study also had developed some cracks, so the engineering team could compare FEA results with physical measurements.

The process vessels were constructed with skirt support expansion joints manufactured out of creep-resistant steel and designed to minimize thermal loading during cycling. During start-up, a 490° C medium was added to the vessel, elevating the temperature 135° C in only 15 minutes. During shut-down, cold water at 50° C was fed in, cooling the vessel's contents by 250° C in 45 minutes. The vessel was subjected to the stresses resulting from these extreme temperature changes approximately 200 times per year, or once every 33 hours.

As a result of this frequent and extreme cycling, cracks had developed in all 82 expansion joints, all at the same position, varying in length and depth depending on time-of-service. For the simulation, TÜV NORD engineers were able to model just a 15-degree sector of the process vessel, because of its symmetrical shape and relatively constant operational load.

As part of the COOP's first module, the team carried out a sequentially coupled temperature-stress analysis. (In this special case, an advanced FSI-calculation was not necessary.) They set up this simulation using real-world operating values for temperature (heat transfer), mass flow (speed), and pressure, as well as actual component geometry. The results indicated that a stress maximum of approximately 1,150 MPa occurred on both heat-up and cool-down when the temperature transient was steepest (Figure 1). Using fatigue curves of test rods (made of materials similar to the process vessel) subjected to load cycles, the lifespan of the component was calculated to be 400 cycles (or approximately two years).

In the COOP's second module, the engineering team used the results of the real-world stress calculation as a starting point and performed a shape optimization by changing the contours of the expansion joints (the same technique can be used on a vessel contour). They determined that minor changes in shape can deliver significant improvements in fatigue strength and that as geometries are modified to improve flow, the resulting pressure losses and local bending stresses can be reduced.

To further understand the interactions between vessel stresses, fatigue, crack geometry, and crack propagation, the

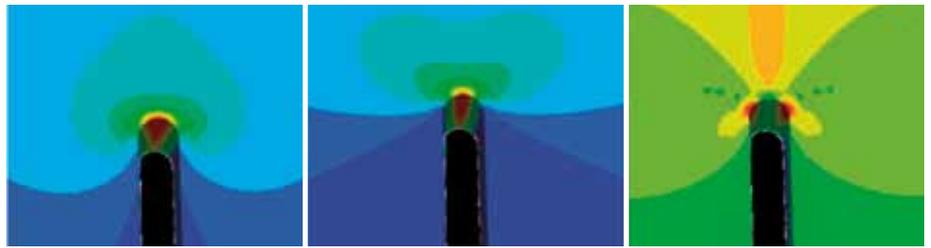


Figure 1. FEA of the stress distribution around the expansion joint of the process vessel skirt support shows equivalent stress (left), circumferential stress (center), and axial bending stress (right).

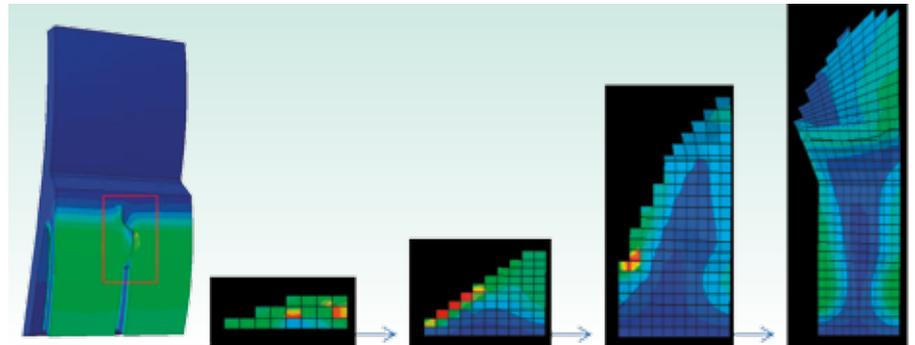


Figure 2. Abaqus eXtended Finite Element Method (XFEM) shows a progression of increasing crack growth (left to right), starting in the area of interest on the process vessel's skirt support expansion joint. Dominant circumferential stresses cause a crack to form at the end of the expansion joint. Due to uneven stress in the frame connector above the joint, the crack grows more rapidly on the outside than on the inside of the connector.

team used the XFEM capability in Abaqus coupled with contour integral calculations. "The XFEM method makes it possible to study crack growth across elements in a way not possible before," said Schulz. "The contour integral calculation gives us the ability to analyze stress intensity values for a series of crack depths and shapes." These analyses showed that the greatest stress occurs when the vessel is most rapidly heating up, the stress decreases with increasing crack depth, and the crack grows more rapidly on the outside than the inside of the frame connector (Figure 2).

The team then compared their simulations with physical testing using strain gauge and crack measurements on the vessels themselves to verify the Abaqus FEA results. "With both stress distribution and crack growth, we found very close correlation between Abaqus results and the measurements," said Schulz. "It became apparent that, at a high number of cycles, crack growth was being overestimated."

Simulation provides a clearer view

For TÜV NORD, realistic simulation has created a window into the black box of power plant operations. With a better view and enhanced understanding of the stresses of thermal cycling, the engineering team can make recommendations to power

providers about optimizing power plant cycling processes and system component designs.

According to Schulz, improvements include the nuances of ramp-up and ramp-down rates, as well as more precise timing of maintenance, service, and inspection cycles. Also, component designs can be modified for greater lifespan. For example, lowering the stress and fatigue level of a main steam-line valve could save as much as €200,000 in replacement costs. The COOP program is designed to minimize and manage unplanned equipment repair and reinvestment such as this.

"Since early 2011, we have been using the COOP process in Germany, and we'll soon be using it with both energy and chemical industry customers worldwide," said Schulz. "By incorporating advanced FEA tools into our methodologies, we are helping create more reliable and flexible power plants."

Such flexibility will likely become a required capability for energy companies seeking to navigate the peaks and valleys of the future power grid.

For More Information

www.tuv-nord.com/en
www.simulia.com/XFEM

Lenovo ThinkPads Get Tougher and More Sensitive with Abaqus FEA

SIMULIA's FEA software helps Lenovo design out flex, engineer in stiffness and improve keystroke feel

The Lenovo ThinkPad line of laptops is known for its striking aesthetics. The shape of the original IBM design was inspired by Japanese *bento* lunch boxes (a form still used in ThinkPads today). The 1995 ThinkPad 701, with its unique “butterfly” expanding keyboard, is in the collection of New York’s Museum of Modern Art.

But most laptops don’t sit safely in display cases, or even stay in one place. Consider a typical morning in a coffee shop: A college student strides to his table, bangs a laptop down, flips it open, and begins typing forcefully and enthusiastically. Nearby, another student slams her computer shut, picks it up with a one-handed squeeze, and dashes out at caffeine-induced speeds, bumping it against the doorframe.

At Lenovo, a great deal of design effort goes into ensuring that “portable” doesn’t mean “breakable.” It’s a measure of how seriously engineers take this task that two recent designs for the ThinkPad X300 and the T400 were code-named Kodachi and Shinai respectively—both the names of Japanese martial arts swords. Like those legendary weapons, ThinkPads are

widely known for their performance and durability as well as looks—a legacy that Lenovo has continued since it purchased the laptop brand from IBM in 2005. “Our aim is to create a high-quality, reliably strong notebook while keeping cost and product development time down,” says Dr. Zhifeng Xin, senior manager at the Lenovo Innovation Design Center (IDC) in Beijing.

The crucial phase of proving out new laptops is done by Lenovo’s award-winning design center. In the past few years, the IDC has received a US Industrial Design Excellence Award (IDEA), Germany’s Red Dot Best and iF Design Awards, Japan’s international G-Mark Good Design Award for industrial design, and the Intel Innovation PC Award, among others.

Engineers at the IDC analyze all of Lenovo’s products, ranging from PCs and notebooks to cell phones and servers. Finite element analysis (FEA) takes place at the concept, development, and failure analysis stages. FEA simulations at Lenovo include modeling drop, shock, vibration, static pressure, and motherboard strain.

The IDC also performs computational fluid dynamics thermal and airflow acoustic simulations.

Realistic simulation of notebook computers like the ThinkPad began at the IDC in 2007 and became a formal checkpoint in the product development process shortly thereafter. Abaqus has been an essential tool for FEA since the IDC first opened its simulation technology center. “With Abaqus, we’re able to be more innovative, more quickly, and produce higher-quality products,” Dr. Xin says.

Engineers use Abaqus early in design process to verify product strength, choose between different versions, and identify and improve problem areas. The process involves pre-processing from CAD to meshed model; establishing loads, boundaries, and part interactions; running the analysis; and creating the reports.

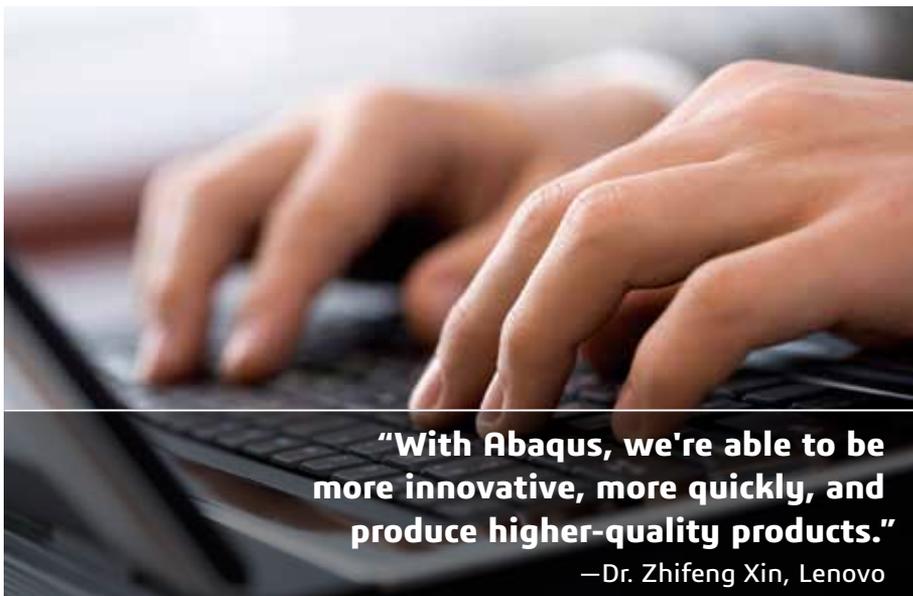
Flexible analysis, rigid results

FEA is especially central to eliminating flex, an objective vital to making the ThinkPad more reliable. A number of forces—such as carrying a closed notebook one-handed—can potentially deform the laptop frame and bend the motherboard, which could damage the soldered connections or the liquid crystal display (LCD). “Notebooks face demanding loads and forces even when they’re just being carried from place to place,” says Dr. Xin. “It’s important for the covers to be stiff enough, with minimal deformation, so that the rear cover will protect the display, while the base cover protects the motherboard.”

During recent work on a new model, simulation of flex was instrumental to the goal of reducing weight without loss in strength or rigidity. A base cover that used carbon fiber reinforced plastic (CFRP) with an aluminum shield would offer support to the motherboard. The LCD screen would be protected by a rear cover with CFRP and graphite-fiber reinforced plastic (GFRP).

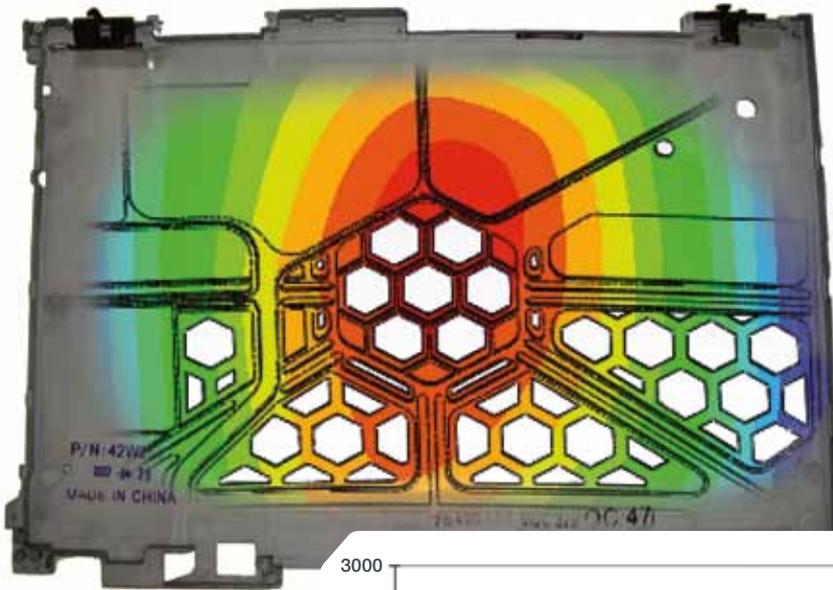
Analysts simulated the physical tests that were performed on a magnesium LCD frame, an LCD panel assembly, a rear cover made of PC/ABS, and a CFRP base cover with its aluminum shield in place. “We applied loads of 400 to 3200 grams, and we compared the resulting deformation to physical test results,” says Dr. Xin. “In each case, simulation had a close correlation with real-world behavior during physical testing.”

In a whole machine analysis, engineers evaluated the performance of laying-up composite CFRP for the rear cover; this kind of composite material could enhance much of the strength for the entire



“With Abaqus, we’re able to be more innovative, more quickly, and produce higher-quality products.”

—Dr. Zhifeng Xin, Lenovo



Abaqus FEA simulation of a Lenovo ThinkPad magnesium frame (above). Test and simulation correlation, Lenovo ThinkPad magnesium frame (right).

machine. “In this instance,” Dr. Xin says, “the nonlinear capabilities of Abaqus were valuable to us because of the complex performance properties of the materials, complex geometry deformation and complex contact behavior.” Engineers also evaluated two alternatives for the array of CFRP: horizontally versus vertically orienting the fibers. The analysis found that horizontal lay-up of the fibers resulted in smaller deformation. The engineers were also able to prove out the strength of a (now-patented) joint line for combining CFRP and GFRP. Due to composite CFRP being limited by part geometry shape, the rear cover is made of both CFRP and GFRP through the patented joint method.

Analysis weighs alternatives

The Kodachi (ThinkPad X300) project proved that the simulation tool has the ability to provide accurate results, allowing engineers to predict the strength of the structure and offer suggestions to CAD designers, on the project codenamed Shinai (ThinkPad T400s), for example. Analyzing deformation of the closed notebooks under a press load revealed that the Light Emitting Display (LED) stress was much lower than in the Kodachi design. During press load on the rear cover of the whole machine, the Kodachi LED

contacted the track-point—the cursor tracking button in the ThinkPad keyboard—potentially creating as much as 8.4 MPa of stress. By contrast, the LED on the Shinai contacted the palm rest on the base, yielding a much lower stress of 2.2 MPa.

The analyses also showed that four areas on the magnesium frame of the Kodachi design exceeded specifications for material yield strength and created deformation of the rear cover of 5.8 mm, resulting in higher overall deformation than on the Shinai, which only exceeded specs of material yield strength in one area. The data from the simulation guided revisions to the Shinai frame, helping the development team speed up the project schedule.

“Because of our work with Abaqus,” Dr. Xin says, “we were able to fix the Shinai design quickly and easily.” In both cases, FEA results were quite similar to physical test results.

FEA gets in touch with keystroke feel

Keyboard design is a strong selling point for the ThinkPad. The notebooks have long been known for their sturdiness and their pleasing signature “feel”—the physical response of keys being pressed. In the case of a new keyboard, the engineers wanted to reduce the thickness of the

assembly (thereby making the overall notebook slimmer) but still keep their traditional keystroke feel.

“‘Feel’ may sound like a vague term,” Dr. Xin says, “but with FEA, it is actually quantifiable.” To confirm that the feel remained the same from the old keyboard to the new, engineers simulated the effects of keystroke pressure on the original dome-shaped rubber spring, the central component that provides each key with its resistance and spring-back. Because the rubber dome was axisymmetric, they were able to model a 2D half-section of the spring and run the analysis on that. The nonlinear simulation enabled the engineers to establish values—on a feeling chart—for how far the rubber dome traveled downward (compressed) as keystroke force increased. (Typical typing force is between 0.6 to 1.2 Newtons.)

The engineers then used Abaqus to evaluate a new rubber dome design for the thinner keyboard in order to match the keystroke behavior of the old rubber dome. “The feeling chart of the new simulation showed close correlation with the old one,” Dr. Xin says, “and both simulations matched up well with data from physical testing.”

These are just some of the simulations a Lenovo ThinkPad undergoes on its way from design to reality. The motherboard itself is also analyzed (and physically tested) for flex, and the Ball Grid Array (BGA) components are modeled undergoing strain to confirm that the solder joints will perform well. “As always, nothing validates a simulation like its close conformity to real-world testing,” Dr. Xin notes. “The push-point tests of the motherboard, and the strain tests for the BGA, were based closely on our physical prototype testing, and they yielded similar results.”

All of the simulations at the Lenovo IDC yield the same result: high-performance, quality products that undergo less prototyping, reach production faster at a smaller cost, and perform reliably for users, from museum curators to coffee-fueled college students.

For More Information
www.lenovo.com

Virtual Testing of a Composite Cylindrical Lattice Structure

Composite grid structures are a promising solution for replacing some of the current aluminium and traditional composite parts in aerospace applications because they achieve a significant weight reduction. Traditional composites are characterized by their strength and stiffness along fiber direction, while mechanical properties in other directions are not favorable. In contrast, the principal load-bearing elements of grid composite structures are unidirectional composite ribs, maximizing their specific structural contribution. The optimum weight-saving solution is a lattice structure, which is a grid structure formed by a lattice pattern without skin.

Researchers Jordi Torres and Norbert Blanco at Universitat de Girona and Encarna del Olmo and Eugenio Grande at EADS CASA Espacio developed a virtual testing tool implemented within Abaqus/Standard to assist during the design of a cylindrical lattice composite structure for aerospace applications. This design will be manufactured by EADS CASA Espacio following the Advanced Fiber Placement (AFP) composite layering method within the frame of the CENIT-ICARO Spanish research project. The AFP manufacturing method is a well-consolidated technology at EADS CASA Espacio, which has been



Figure 1. First buckling mode calculated with a beam-based model used to assess the structural response.

selected to build a new generation of ultralight structures based on grid and lattice designs.

The virtual testing tool was implemented in two steps. First, a parametric analysis with a preliminary model was considered to determine the best design solution for the cylindrical lattice structure in terms of natural frequency, buckling load and weight. Due to the nature of lattice structures, the

geometry of all the lattice configurations was completely parameterized and analyzed with beam-type element simulations.

The second part of the virtual testing procedure consisted of the lattice structure testing simulation to predict the mechanical behavior of the real structure. Due to the size differences between the large-scale structure and the local interactions, the virtual testing was divided into two analyses. A Finite Element (FE) model composed with beam elements (Figure 1) assessed the global structural response—natural frequency and buckling load, while the local failure was obtained with a FE model including solid elements only. In this case, a global model provided the displacements to a local submodel that assessed the failure criteria (Figure 2). The intralaminar failure was calculated with LaRC-04 failure criteria, which account for different composite failure mechanisms and consider nonlinear shear behavior and in-situ effects. The possible adhesive peel-off between the lattice part of the structure and the extreme load interfaces was also analyzed through the following procedure: 1) Determine critical zones using a stress-based criterion, 2) Apply the Virtual Crack Closure Technique (VCCT) with a pre-crack with the measure of the minimum detectable defect and 3) Run a nonlinear analysis involving cohesive elements.

The results of the composite cylindrical lattice virtual testing procedure showed an optimum ratio between weight and load capacity of the structure. As desired, fiber compressive failure in the ribs of the lattice part far from the load interfaces was predicted before the general collapse of the structure under buckling conditions. Moreover, the stability and stiffness requirements could be achieved with a significant weight reduction in comparison with current designs.

J. Torres¹, N. Blanco¹, E. Del Olmo², E. Grande²

¹ AMADE, Mechanical Engineering and Industrial Construction Department, Universitat de Girona, Girona, Spain.

² EADS CASA Espacio, Technology and Innovation, Madrid, Spain.

The first author acknowledges the Ph.D. scholarship BES-2010-036295 funded by the Spanish Government in the VLANCO project (MAT2009-07918).

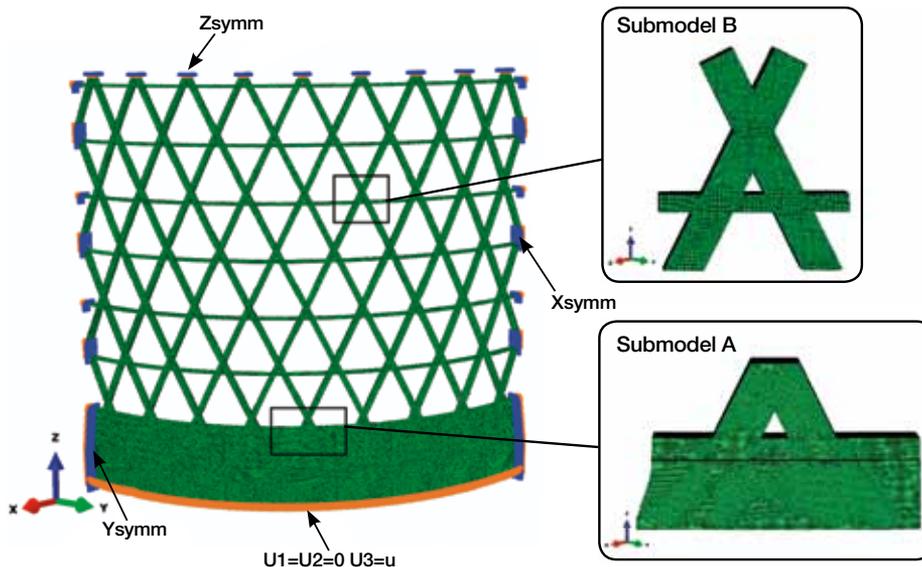


Figure 2. Solid model and submodels to analyze the failure mechanisms of the cylindrical lattice.

For More Information

amade.udg.edu/eng
www.simulia.com/XFEM

Fracture Modeling of Ceramic Liners for Total Hip Arthroplasty

Today, total hip arthroplasty (THA) is the treatment of choice to relieve joint pain and loss of mobility as a result of end-stage osteoarthritis or other severe hip pathologies. THA is one of the most successful surgical interventions in medical history. Currently, more than 250,000 cases are performed per year in the U.S., a figure expected to double in the next 20 years.

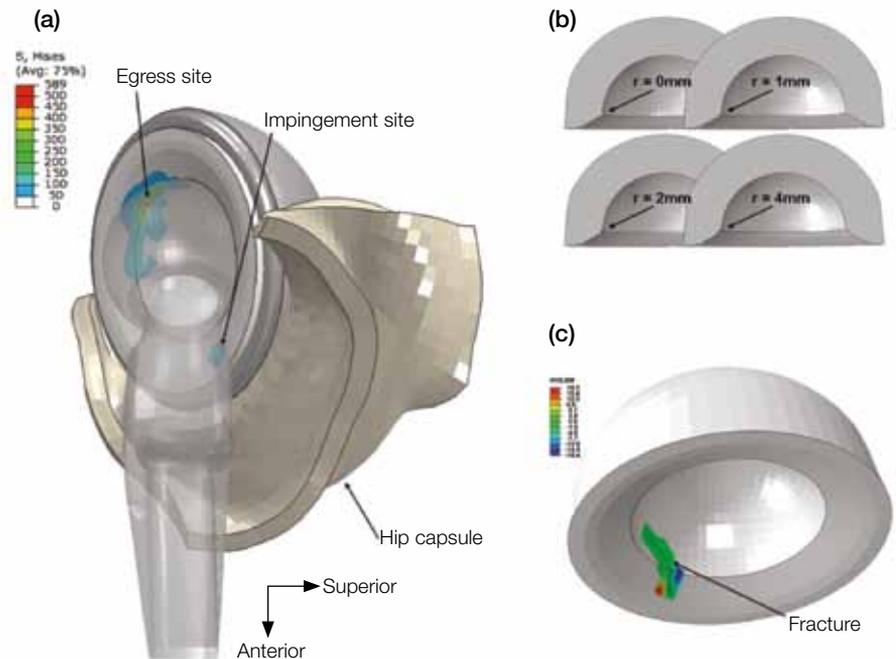
While highly successful, THAs do sometimes fail. Conventional THA bearings, consisting of a metallic femoral component articulating with a polyethylene acetabular cup, are challenged by accumulative wear debris generated over the lifespan of the implant, which can elicit adverse biological reaction. Alumina ceramics for THA were introduced nearly four decades ago to improve long-term results in younger and more active THA patients. Ceramic-on-ceramic (CoC) bearings offer several advantages over contemporary bearings, such as excellent compressive strength and lower wear. However, because of the ceramic material's brittle nature concerns persist regarding implant failure due to catastrophic fracture. In general, both components of a CoC implant are prone to fracture. Fracture of the ceramic head is a well-recognized problem historically, and extensive investigation has led to several design-specific improvements.

While it is well-established clinically that impingement between the femoral neck and liner can predispose to fracture, to date, little quantitative information exists regarding fracture propensity for liners. To help close this knowledge gap, an eXtended Finite Element Model (XFEM) of THA impingement was developed by University of Iowa and SIMULIA to investigate fracture risk and crack propagation for various cup designs and surgical orientations.

Method

A previously developed and physically validated nonlinear dynamic FE model of THA impingement (Figure a) was used to determine stresses developed during various impingement scenarios for a CoC implant. The FE model consisted of THA hardware (28-mm head, 46-mm liner and cup backing). The hip capsule, which acts to stabilize the joint, was assigned a fiber-based anisotropic hyperelastic constitutive model (Holzapfel-Gasser-Ogden).

Four separate models were created, by varying the cup-lip fillet radius between 0 mm and 4 mm at a constant cup inclination of 40°



(a) Dynamic FE model of THA impingement demonstrating stress concentrations arising at both the impingement site and head egress site during head subluxation. (b) Four distinct cup designs of various edge-chamfer radii were investigated. (c) Fracture initiation and propagation corresponding to egress site during the XFEM analysis.

(Figure b), to simulate different edge profiles in contemporary THA use. Four additional models were generated to investigate surgical orientation, by varying the cup inclination between 30° and 60°, each with a constant 10° of anteversion.

Boundary conditions for driving the impingement model consisted of an input sequence of prescribed joint rotation and loading, determined from optical motion capture of subjects performing a stooping motion.

These impingement models were executed in Abaqus/Explicit. Stresses occurring during the simulations were passed (node-based) to the XFEM model of liner fracture in Abaqus/Standard. The XFEM submodel contained two separate enrichment regions, corresponding to the impingement and egress sites. Damage initiation criteria were specified at 300MPa maximum principal stress, with literature-based mixed mode (power-law) damage evolution. A separate analysis assumed a pre-existing flawed liner with correspondingly reduced fracture parameters.

Results

Fracture initiation was tracked at both the egress and impingement sites. Fractures,

when they occurred, typically developed at the egress site (Figure c). Fracture initiation was demonstrated to be sensitive to both cup orientation and cup edge radius, with fracture risk increased for sharper edges at higher values of cup inclination. Substantially higher occurrence of fracture was observed for the (assumed flawed) reduced fracture criteria analyses.

The use of XFEM has proven to be a valuable technique to investigate crack initiation and propagation in ceramic THA. This technique can be used in THA design and surgical planning analysis to reduce ceramic THA fracture risk.

Jacob Elkins¹, Xiangyi (Cheryl) Liu², Xiaoliang Qin², Zhenzhong Du², and Thomas Brown¹

¹Dept. of Orthopaedics and Rehabilitation, The University of Iowa, Iowa City, Iowa.

²Dassault Systèmes Simulia Corp, Providence, Rhode Island.

For More Information

www.uiortho.com
www.simulia.com/XFEM

eXtended Finite Element Method (XFEM)

How to Estimate the Safe Operating Life of Structural Components

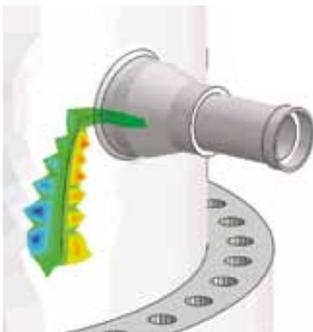
Fracture and failure modeling allows for product designs that maximize the safe operating life of structural components. The state-of-the-art XFEM capability for modeling fracture and failure in Abaqus alleviates the shortcomings associated with traditional approaches.

XFEM provides the ability to model discontinuities, such as cracks, along an arbitrary, solution-dependent path. The method can simulate both the initiation and propagation of a discrete crack. Because the mesh is not required to conform to the geometry of the discontinuity when studying crack growth, it is not necessary to remesh the bulk materials during the solution to account for changes in crack size or orientation.

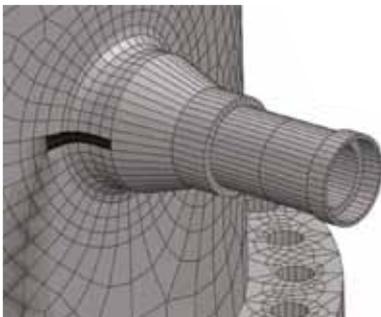
XFEM allows the engineer to model the presence of discontinuities in an element by enriching degrees of freedom with special displacement functions. In addition, XFEM enables the modeling of discontinuities in the fluid pressure field, as well as fluid flow within the cracked element surfaces, such as hydraulically driven fracture. XFEM can also be used for low-cycle fatigue analysis using the direct cyclic procedure.



Mises stress distribution in the pressure vessel



Contour plot of PHILSM near the nozzle



STATUSXFEM showing progressive damage and failure

TIP: How to model crack propagation in a structural component

Here's What You Do:

1. Build the model in Abaqus/CAE with no XFEM specification and run a test job to ensure the loads and boundary conditions are correct
2. To model the initial flaw, create a separate part representing the crack surface or line and assemble it along with the model you created in step 1
3. Specify the damage initiation and evolution for the traction-separation law as the material parameters for the enriched regions
4. Provide a small value of stabilization to regularize the analysis and improve convergence behavior
5. Define the enrichment region and assign the interaction property using the crack editor
6. Assign the location of the initial crack as an inside edge or a surface belonging to the same instance as the enriched region or to a different instance (preferred)
7. Specify frictionless small-sliding contact as the contact interaction property
8. Specify minimum and maximum increment sizes, typically with an increased number of increments to improve convergence behavior
9. Specify additional solution controls (i.e., discontinuous analysis settings) and increase the number of cutback attempts to aid convergence
10. Request XFEM-related output variables STATUSXFEM, PHILSM and PSILSM to aid crack visualization
11. Change to the Job module and create a new job to run the XFEM analysis
12. Use the STATUSXFEM and PHILSM and PSILSM to view the progressing crack front

For More Information

www.simulia.com/XFEM

2012 SCC—Save the Date!

The 2012 SIMULIA Customer Conference (SCC) will be held May 15–17 in Providence, RI, and will provide equally interesting and pertinent topics encompassing the use of Abaqus, Isight, and SIMULIA SLM across a wide range of industries. Visit our website to see what customer papers have been submitted.

www.simulia.com/SCC2012

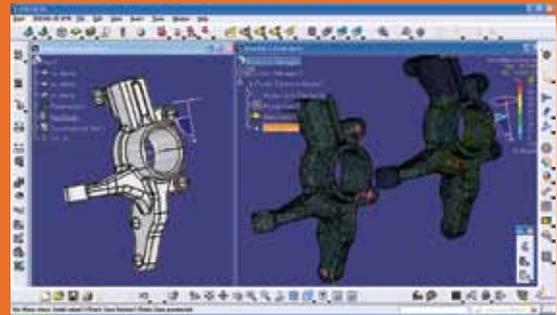


2011 SCC

V5R21 Release

The V5R21 release enhances CATIA Generative Part Structural Analysis by providing surface selection that enables users to save time during model creation, making surface selection more reliable and intuitive. It is now possible to import composite properties from the CATIA Composite Design workbench into an analysis model. V5R21 also provides a simpler, more efficient approach to contact modeling, using the Abaqus “general contact” technology.

www.simulia.com/v5r21

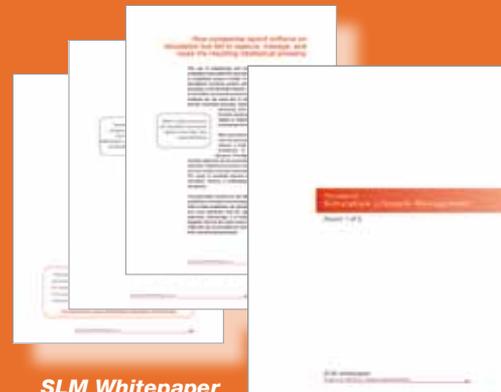


V5R21

SLM Whitepaper Available

Manufacturers are increasing their use of analysis and simulation tools due to continuous pressure to deliver innovative products to market faster. Integrating the management of these tools, data and processes with the product development cycle is crucial. This paper presents the challenges associated with deploying simulation lifecycle management (SLM) environments, required elements, benefits, and explains how Dassault Systèmes SIMULIA SLM products respond to those needs.

www.simulia.com/slm_whitepaper



SLM Whitepaper

SIMULIA Resource Center

Learn more about SIMULIA’s product lines, download technology briefs on detailed application examples, access select conference papers and view webinars on-demand of SIMULIA solutions—all in one central location!

www.simulia.com/resources



SIMULIA Resource Center



**See your way to a
better design.**

Explore, Discover, Improve...

In today's competitive business environment, improving product efficiency, safety, and reliability—while reducing time and costs—are mission critical goals. Our customers are achieving these goals by leveraging Isight for simulation automation and design optimization.

Isight enables the integration of CAD, CAE, test data, and other applications into consistent and repeatable simulation process flows. By automating simulation execution, Isight enables you to use powerful design of experiments and optimization tools to quickly—and affordably—explore and improve product performance.

Download Customer Papers on Isight at
www.simulia.com/cust_ref

 **SIMULIA**