ABAQUS INSIGHTS

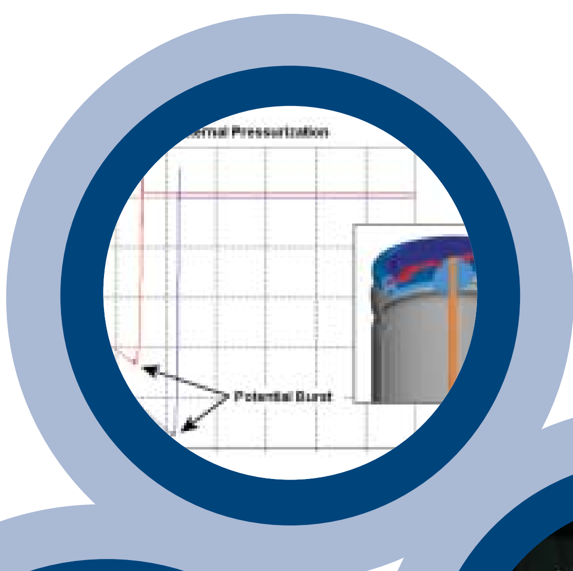
Summer 2005

Regional Users' Meetings

Optimizing the Design of Rayovac Batteries

Pedestrian Protection Impactors

Technology Briefs



ABAQUS

ABAQUS INSIGHTS

In this issue:

ABAQUS INSIGHTS is published by

ABAQUS

Rising Sun Mills
166 Valley Street
Providence, RI 02909-2499
Tel. +1 401 276-4400 Fax. +1 401 276-4408
info@abaqus.com

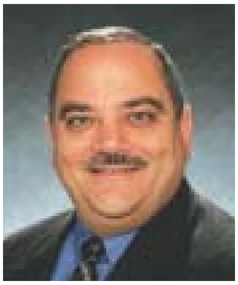
WWW.ABAQUS.COM

Editor:
Jeff Rankin

Contributors:
Charlie Chin
Pablo Cruz
Karen Curtis
Karen Donovan
Karl D'Souza
Mark Goldstein
Stacy Hart
Asif Khan
Charles Lu
Carles Mitjans
Mark Monaghan
Steve Morse
David Palmer
Jon Wiening
Stephanie Wood

Production Coordinator:
Laura Wistow

Our Focus is the Same as ABAQUS Evolves



Since 1978 the ABAQUS organization has stayed true to its original strategy of developing and supporting advanced finite element analysis software. Keeping focused has allowed us to improve the ability of ABAQUS to predict real world behavior and to reduce the time it takes to get useful results. When we see our customers improving their processes and products, we know we're doing our job right.

Methods Evolution and Unified FEA are two recent ABAQUS initiatives that support our strategic focus. Reconsidering current analysis workflows and practices with an eye toward more realistic simulation and shortened analysis turnaround time is what Methods Evolution is all about. And nothing energizes us more than when we hear your ideas on how to improve ABAQUS to evolve and even revolutionize your current processes. We thrive on partnerships with customers. As many of you know, close collaboration between ABAQUS and our customers has yielded many of the innovations and breakthroughs our industry has witnessed over the past 10 years.

Our Unified FEA initiative responds to our customers' need to consolidate the number of analysis products they use. We are broadening the scope of simulation attributes that ABAQUS can address, and when it makes sense, building successful partnerships with other suppliers to enable integrated analysis. Unified FEA is in some ways a forward-looking vision, but it is no doubt a vision we should be pursuing with you. We have already seen several ABAQUS user conference papers that bear out its viability.

Developments in ABAQUS resulting from these initiatives are paying off. Customers have told us that advances in ABAQUS software for fracture and failure analysis are on target. Methods in this area will become more widely adopted over the next five years, just as nonlinear analysis is today. Capability for analyzing fluid-structure interaction is another initiative that is bearing fruit. Some customers report more progress in a few weeks using ABAQUS in concert with FLUENT and MpCCI software than they have made in months of effort with other software.

As I began my tenure as ABAQUS CEO just over four years ago, I saw ABAQUS as the real jewel in our industry. I still do. We have the best technology, the finest development and support teams, and a reputation for ethical business dealings second to none. At the time our plan was to keep growing the ABAQUS development and support teams while prudently building our marketing team, strengthening our product management and alliances functions, and also increasing our efforts to spread the good news about our customers' accomplishments. I hope you will agree with me that the changes at ABAQUS over the past four years have been smooth and productive. I would like to thank our customers and the ABAQUS staff for their tremendous efforts. You have made the evolution over the past four years an unqualified success.

Now, on to the next stage of our evolution. For the last five years Dassault Systèmes has been looking at our market and assessing various organizations that might fit with their vision for powerful and easy-to-use tools to support the product lifecycle management (PLM) market. As their research deepened, they discovered a simple fact: ABAQUS is successful as a company because our customers are satisfied with, and even excited about, using our software. Dassault Systèmes approached us earlier this year, and over the several months that followed, the ABAQUS Board of Directors and the senior management team came to the conclusion that accepting Dassault Systèmes's offer to acquire our business was in the long-term best interest of our customers, employees, business partners, and technology.

What does this next stage in our evolution mean to you? We expect you to see little change. You will continue to work with the same sales, support, and services teams in our network of local offices. All the ABAQUS products will continue to be enhanced aggressively, and we will continue to partner with those companies whose software is an integral part of your processes. As part of a bigger company, we will have access to more resources and be able to consider a broader scope of simulation than we have been able to in the past. We invite you to help us continue to improve the state-of-the-art in simulation and to provide solutions that allow you to engineer your products and manufacturing processes more effectively and efficiently. Keep the energetic dialog going with our product management and development teams and challenge us to provide the tools and services that you will need to drive your businesses forward.

On behalf of all of us at ABAQUS, I would like to reconfirm our commitment to delivering the best possible technology, software, and technical support to our customers. We truly value the close relationships we have with you, and this will not change. As you recognize ways to help us improve our products or the way we do business, I encourage you to speak with your local support team. I also invite you to attend one of our Regional Users' Meetings, which we hold each year in the fall. Key technical staff from headquarters, along with me or other members of the senior management team, will be at each of these events. We would be delighted and grateful to hear first hand your thoughts on how to make this next stage of evolution at ABAQUS a success.

Sincerely,

Mark Goldstein
Chief Executive Officer
ABAQUS, Inc.

Regional Users' Meetings

If you are looking for an opportunity to network with nearby ABAQUS users or want to learn more about ABAQUS real-world applications, then check out your local Regional Users' Meeting. These events will review the newest enhancements of ABAQUS Version 6.5 and give you a brief glimpse of what's ahead in ABAQUS Version 6.6. For more details, please visit **www.abaqus.com/RUM2005.**

North America

September 27–28, Lafayette, IN
September 29, Cleveland, OH
October 5–6, Toronto, Ontario, Canada
October 19, Houston, TX
October 24–26, San Diego, CA
October 26–27, Plymouth, MI
November 8, Baltimore, MD
November 10, Philadelphia, PA
November 15, Westboro, MA

Europe

September 19–20, Nuremberg, Germany
September 27–28, Graz, Austria
September 29–30, Istanbul, Turkey
October 10–12, Milano, Italy
October 25, Prague, Czech Republic
November 10–11, Pamplona, Spain
November 15–16, Northamptonshire, UK
November 16–17, Winselerhof, Landgraaf, The Netherlands
November 17, Paris, France
November 25, Poznan, Poland

South America

November 14–15, Buenos Aires, Argentina

Asia Pacific

November 1, Tokyo, Japan
November 3–4, Taipei, Taiwan
November 2, Osaka, Japan
November 7–9, Hangzhou, China
November 15, Kuala Lumpur, Malaysia
November 16, Bangkok, Thailand
November 18, Singapore
November 23, Bangalore, India
March 2006, Korea

2005 ABAQUS Users' Conference Recap

The city of Stockholm, built on 14 islands, connected by 57 bridges, embracing the unique character of the 13th century Old Town and modern high-tech architecture, was the home of the 2005 Annual Users' Conference. Surrounded by beautiful landscape and historical attractions, Stockholm provided an energetic and entertaining environment for customers, guests, alliance partners, and staff. We were delighted to see an increase in attendance again this year, with well over 200 attendees.

Attendees gathered for the Conference Banquet at the Hasselbacken Hotel & Restaurant.

Technical Content Surpassed Expectations

It was a privilege to have Nokia and Scania join us as guest lecturers. The speakers contributed a familiar local presence as well as a historical appreciation, as some of the early capabilities of the ABAQUS product line were developed in partnership with our Scandinavian office and the region's early customers. Marcus Theman from Nokia presented, "Evolution of a Mobile Phone Drop Test Simulation at Nokia" and Joakim Orbom from Scania presented, "FE-Analysis at Scania. Using ABAQUS to Predict the Dynamic Behaviour of Joints in Truck Structures."

The conference included technical presentations by users as well as General Lectures and Tutorials by ABAQUS staff. Attendees were satisfied with the balance between ABAQUS staff and user presentations as well as enthused by the outlook on future releases.

> ***"The networking opportunities were extremely valuable. I enjoyed learning how other companies use ABAQUS and how their organization interacts with the product development community within their company."***

Interaction with ABAQUS Staff

With nearly 50 of the ABAQUS technical staff in attendance, users and staff were able to interact and build relationships. The opportunity to hold both smaller conversational meetings, as well as formal Customer Review Team (CRT) meetings, proved effective and informative.

Complementary Technologies

The conference included a strong presence by some of our business partners. The Complementary Technology Exhibits area featured Safe Technology, Ltd.; Linux Networx; SGI; Fluent, Inc.; Dassault Systèmes SA; Altair Engineering; HP; GNS-mbH; IBM; BETA CAE Systems SA; Fraunhofer SCAI; and CD-adapco. Attendees made use of coffee breaks and free time to learn more about the benefits these companies offer.

In addition, Presentation Only sponsors included AVL List GmbH, AMD, FE-Design GmbH, and e-Xstream engineering SA. During the evening Partner Reception, attendees joined ABAQUS and our business partners to combine technical and social conversation, further building essential relationships.

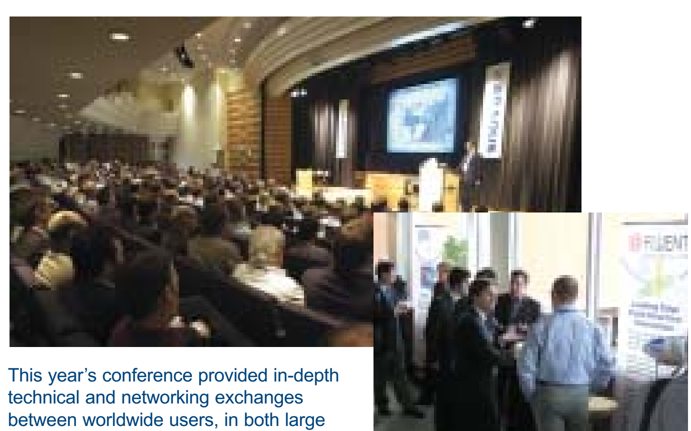
We appreciate the feedback received from the online event survey. Over 80% of attendees were highly satisfied with their overall experience, the venue and event logistics, as well as the ABAQUS content presented at the conference.

Additionally, more than 70% of attendees praised the user presentations and evening programs. Your feedback better prepares us in planning next year's conference. ABAQUS is encouraged by the level of the user papers presented as well as the continuous and increasing attendence this annual conference attracts.

> ***"The interactive presence of top level ABAQUS personnel at the conference demonstrated the company's commitment to customers and excellence."***

Mark Your Calendars

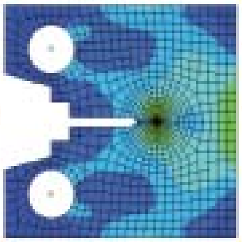
The 19th annual Users' Conference will be held May 23–25, 2006, at The Charles Hotel, Boston, Massachusetts. Prospective authors of contributed papers are invited to submit abstracts of 250 words or less for review by ABAQUS. For more information, please visit **www.abaqus.com/auc2006**.



This year's conference provided in-depth technical and networking exchanges between worldwide users, in both large and intimate environments.

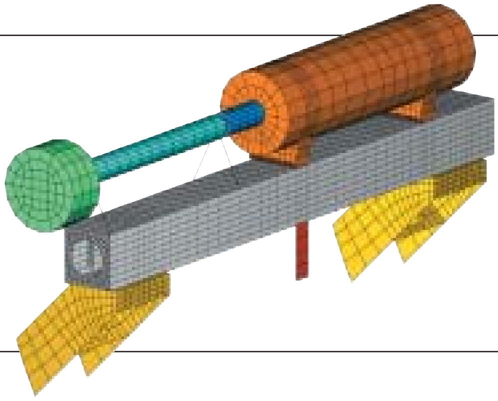
ABAQUS Technology Briefs

Several new Technology Briefs have been published since the Winter 2005 edition of ABAQUS Insights. Electronic copies of each document are available at **www.abaqus.com**.



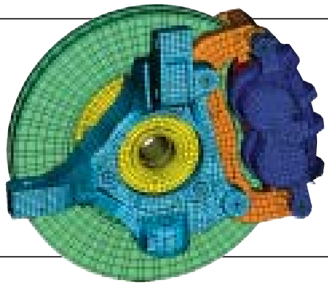
Fracture Mechanics Study of a Compact Tension Specimen Using ABAQUS/CAE
With the release of Version 6.5, ABAQUS/CAE includes modeling and postprocessing capabilities for fracture mechanics analyses. These new features provide interactive access to the contour integral fracture mechanics technology in ABAQUS/Standard. In Tech Brief TB-04-FMCAE-1, these capabilities are further detailed in a study of J-integral calculations for a compact tension specimen.

Dynamic Design Analysis Method (DDAM) Response Spectrum Analysis with ABAQUS
The Dynamic Design Analysis Method (DDAM) is a US Navy response spectrum analysis technique used for examining the behavior of structures experiencing underwater shock (UNDEX) loading. Because the DDAM analysis procedure is extensive, it is an ideal candidate for an ABAQUS/CAE custom GUI application. Tech Brief TB-05-DFA-1 describes the features of DDAM for ABAQUS, which is free for current users of ABAQUS.



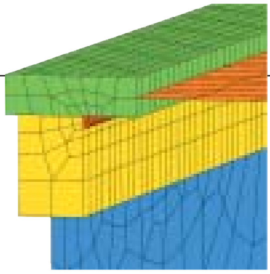
Fluid Structure Interaction Analysis with ABAQUS and FLUENT
Many engineering problems of interest involve the coupled response of a flowing fluid and a deforming structure. ABAQUS, Inc. and Fluent, Inc. have partnered to provide a fluid-structure interaction capability that is now available in current versions of ABAQUS and FLUENT. Several problems from different industries that have been analyzed with co-simulation capability are presented in Tech Brief TB-05-FSI-1.

Automotive Brake Squeal Analysis Using a Complex Modes Approach
Under certain operating conditions, automobile brakes can produce unwanted noise that is commonly referred to as “brake squeal.” The complex eigenvalue capability in ABAQUS/Standard can be used to study this phenomenon and eliminate squeal propensity in the design stage. The analysis procedure and a comparison with experimental data are given in Tech Brief TB-05-BRAKE-1.



Filament Wound Composite Pressure Vessel Analysis with ABAQUS
The analysis of a filament wound composite structure is complicated by the continually varying orientation of the filaments. Beginning with Version 6.5, ABAQUS provides support for plug-ins, which are small applications that can be used to extend the functionality of ABAQUS/CAE. In Tech Brief TB-05-FWC-1, an ABAQUS/CAE plug-in that simplifies the task of analyzing filament wound composite structures is presented.

Welding Analysis with ABAQUS
Metal welding processes are employed in various industries. Gas welding techniques use the heat from a flame to simultaneously melt a filler material and the parts to be joined. In Tech Brief TB-05-WELD-1, the use of ABAQUS for this class of problems is discussed and an example analysis is presented.



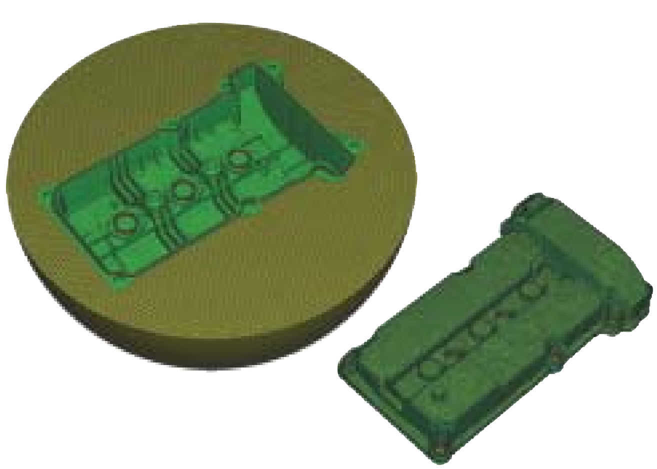
Engine Cover Acoustics with Infinite Elements

at Dana Corporation

In recent years, there has been an increase in the attention being paid to acoustic emission from automotive vehicles. Government regulations, warranty costs, and customer satisfaction have all had a role to play in this increased interest. Automotive suppliers and OEMs both realize the benefits that simulation can provide in this area. ABAQUS has been actively engaged with several customers on acoustics-related applications, both at the component and at the full-vehicle level, to better understand their requirements, and to incorporate functionality that will make their task easier and more efficient.

Acoustic infinite element model for a geometrically simple engine valve cover.

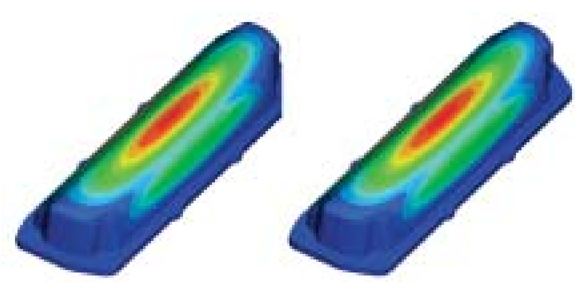
In this article, we briefly discuss work on engine cover acoustics performed by Charles Lu of the Dana Corp., and Karl D'Souza and Charlie Chin, engineering specialists at ABAQUS, Inc. The current project, presented as a paper at the 2005 SAE Noise and Vibration Symposium ("Sound Radiation of Engine Covers with Acoustic Infinite Element Method"), is a follow-up to an earlier 2003 SAE N&V paper titled, "Acoustic Analysis of Isolated Engine Valve Covers." Both papers discuss modeling issues related to a cover vibrating in an infinite medium. In the earlier project, the effect of the infinite medium was simulated using nonreflecting (impedance-type) boundary conditions. In this work, the impedance conditions are replaced with acoustic infinite elements.



Left: Bottom view of the acoustic finite element mesh that envelopes the engine cover. Right: Top view of the finite element model of a production engine cover.

The methodology is presented using two different models. In the first model, a cover with a simple geometric shape is analyzed. Due to the simplicity of the shape, the acoustic infinite elements can be coupled directly to the surface of the cover, without the need for an intermediate acoustic finite element domain. This technique results in rapid turnaround times, since the cover structural mesh may be reused for the infinite elements. The results from this model are compared to those from the earlier paper which used the same cover model, but required an acoustic domain to be created as well, to model the surrounding air. It was shown that the results from both approaches are in good agreement. However, the infinite element model proves to be significantly more efficient, both in terms of memory usage and computational time.

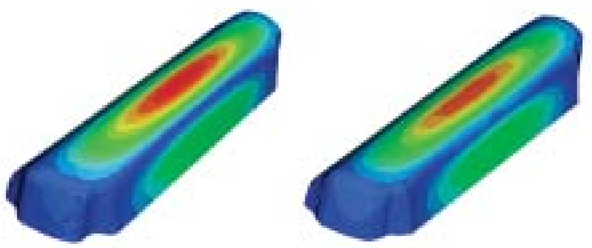
The second model is a fairly complicated cover assembly, displaying surface irregularities such as small raised portions and concavities. As the infinite elements provide the highest possible accuracy when placed on a convex surface, this model uses a combination of acoustic finite and infinite elements. A small volume of acoustic finite elements is first constructed such that the inner boundary of this volume is defined by the cover surface, while its outer boundary is convex. The infinite elements are then placed on this convex surface. By means of this construct, the model is able to accurately simulate both the complex near-field wave nature via the acoustic finite elements, as well the effect of the unbounded exterior via the acoustic finite elements.



Comparison of **surface displacements** obtained from acoustic infinite element model (left) and acoustic finite-element-only-model (right).

The results from this model have been compared with experimental measurements conducted at various engine operating conditions, and satisfactory correlations have been observed. Moreover, the use of a simple Python script (described in detail and available in Section 8.10.12 of the ABAQUS Scripting User's Manual) allows the visualization of acoustic pressures in the far field. This permits the analyst to mesh the acoustic domain only as much as is required to obtain accurate results, thereby further reducing the analysis cost.

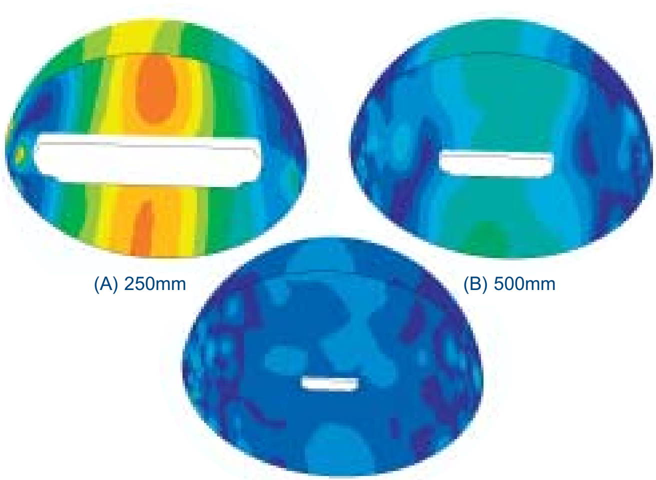
The analysis is carried out in a two-step manner. The first step involves the assembly of the cover, modeled as a static (*STATIC) process, and the second step is modeled as a harmonic vibration analysis (*STEADY STATE DYNAMICS, DIRECT) of the system at several different frequencies. In the assembly step, the gaskets and grommets are compressed until the metal limiter sleeves establish contact with the cylinder head. This statically equilibrated state forms the base state for the subsequent frequency response analysis. Once assembled, the cover model is excited through the ten bolts around the bottom flange. For comparison purposes, the cover surface acceleration and sound pressures in a plane 10cm above the cover are used.



Comparison of **acoustic pressures** obtained from acoustic infinite element model (left) and acoustic finite-element-only model (right).

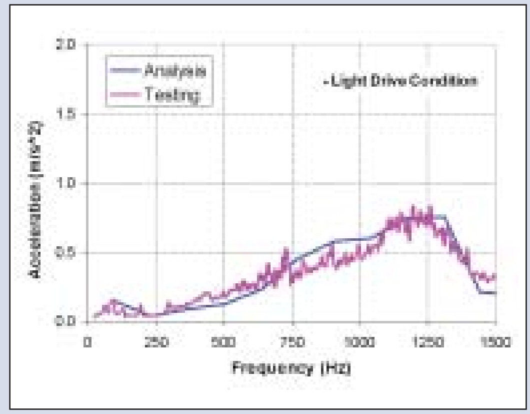
In this analysis, a fully coupled structural acoustic model was used, since the overall run time was not too great. In a more general situation, the analysis could be split in a sequential manner with the results of a structure-only force response analysis driving the subsequent acoustic-only model.

The conclusions were that the solution obtained from the infinite element approach was consistent with that using the impedance method, and also agreed well with experimental data. The infinite element technique extends the capability by allowing examination of far-field results as well, thus proving to be an accurate and efficient technique to model the sound radiation from engine cover structures.

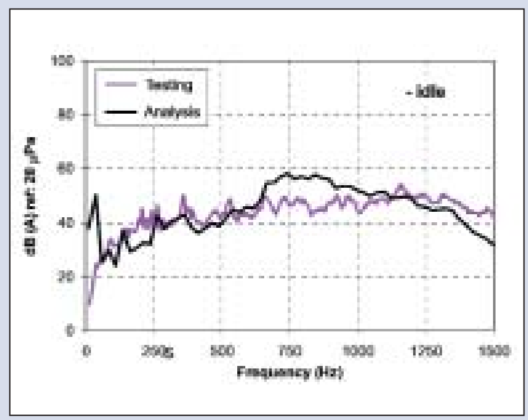


Far-field sound pressure from the infinite elements: sound field at a distance of (A) 250mm, (B) 500mm, (C) 1000mm.

To learn more, please email **insights@abaqus.com**.



Comparison of surface vibration of the production cover between testing and analysis at light-drive condition.



Comparison of radiated sound of the production cover between testing and analysis at idle condition.

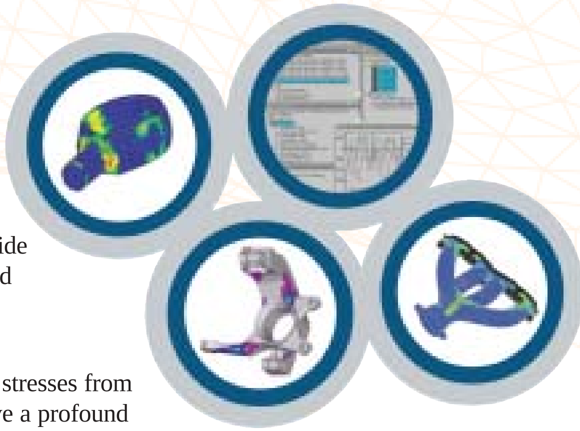
About the Authors

Charles Lu is a Senior Engineer at Engine Products Group, Dana Corporation, where he is responsible for structural and acoustic analysis of automobile engine components. He holds a Ph.D. in Mechanical and Materials Engineering from the University of Western Ontario.

Karl D'Souza is a Senior Engineering Specialist at ABAQUS, Inc. focusing on acoustic and underwater explosion simulations. He holds a M.S. in Mechanical Engineering from SUNY, Buffalo.

Charlie Chin is an Engineering Specialist at ABAQUS, Inc. focusing on vehicle NVH applications. He holds a Ph.D. in Engineering Mechanics from Virginia Tech.

Effective Fatigue Analysis with fe-safe



Introduction

For over six years, ABAQUS and fe-safe have shared a technical and business relationship, and today ABAQUS distributes Safe Technology's fe-safe software for fatigue analysis. This article discusses how various approaches to fatigue provide different levels of accuracy. Some of the basics of biaxial fatigue are explained and different methodologies are compared.

The Importance of Biaxial Fatigue

Fatigue cracks often initiate from the surface of a component. Examination of the stresses from an FEA analysis shows biaxial stresses at many locations. Biaxial stresses can have a profound and often adverse effect on fatigue life. Various approaches for estimating fatigue life will be discussed.

Using Principal Stresses

Early attempts to analyze biaxial fatigue were based on principal stresses, using a conventional S-N curve. However, it has become well known that, based on the results of fatigue tests for most commonly used steels, the torsional fatigue strength is much lower than the axial fatigue strength, with the allowable principal stresses in torsion being approximately 60% of the allowable axial stress. Calculating fatigue lives using principal stresses will be grossly optimistic for torsion loading, and allowable torsional fatigue stresses will be overestimated by a factor of about 1.7 (1/0.6). In fatigue analysis from an FEA model, this could mean the difference between identifying and missing a potential fatigue "hot spot."

Over the past 20 years it has been shown that principal stresses should be used only for fatigue analysis of "brittle" metals; for example, cast irons and very high strength steels. A fatigue analysis using principal stresses gives very unsafe fatigue life predictions for more ductile metals, including most commonly used steels.

Using Mises Stress

Mises stress is at first sight an attractive parameter for biaxial fatigue analysis because it can be obtained easily from FE analysis. However, Mises stress is always positive even when the principal stresses are negative which can lead to an error in the fatigue life calculation. The concept of "signed Mises stress" was introduced, but it does not correlate well with biaxial fatigue test data.

Using Strain-Based Approaches

Shear Strains

Fatigue cracks in ductile metals, including most commonly used steels, initiate on planes that experience the maximum shear stress amplitude or shear strain amplitude. One approach is to take the conventional strain-life equation (1) and rewrite it in terms of shear. The strain-life equation is derived for tests on uniaxially stressed specimens:

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma'_f}{E}(2N_f)^b + \varepsilon'_f(2N_f)^c \qquad (1)$$

where $\Delta\varepsilon/2$ is the strain amplitude for a fatigue cycle, N_f is the endurance in cycles, and the remaining terms are material properties.

Using shear strain as the fatigue parameter produces:

$$\frac{\Delta\gamma}{2} = 1.3\frac{\sigma'_f}{E}(2N_f)^b + 1.5\,\varepsilon'_f(2N_f)^c \qquad (2)$$

This is an equation for biaxial fatigue analysis, using shear strain as the fatigue parameter, with the constants chosen so the equation gives the same fatigue life as (1) for uniaxial stresses. The material properties are the same as (1); i.e., standard uniaxial properties, so no biaxial test data are required.

Experience with this equation shows a tendency to give conservative fatigue life estimates for a range of biaxial stress conditions for ductile metals. For pure torsion equation (2) is shown to predict that the fatigue strength (allowable stress) in torsion is 50% of the fatigue strength under axial loading, whereas a value of 55-60% is more typical for steels.

Combined Shear and Normal Strains

Findley proposed that, for a shear stress cycle, any applied normal stress must modify the fatigue life and fatigue lives should be calculated using the maximum shear stress amplitude with the normal stress as a modifier. McDiarmid developed Findley's proposal into a stress-based multiaxial fatigue criterion for high cycle fatigue. Brown and Miller took a similar approach using strains instead of stresses. Using the same method as for the shear strain criterion described above, Kandil, Brown, and Miller developed the equation:

$$\frac{\Delta\gamma}{2} + \frac{\Delta\varepsilon_n}{2} = 1.65\frac{\sigma'_f}{E}(2N_f)^b + 1.75\,\varepsilon'_f(2N_f)^c \qquad (3)$$

The widespread use and success of this Brown-Miller parameter for both proportional and nonproportional stresses is now well established.

The Wang-Brown Criterion
Wang and Brown proposed an alternative criterion for nonproportional stresses. This approach treats fatigue damage as a scalar quantity. As for the Mises stress criterion, additional rules are required to identify fatigue cycles. The Wang-Brown method introduces a pseudo-stress that can be cycle-counted. The method requires both axial and torsional test data, so additional materials test data are required for this algorithm. The success of the Brown-Miller method makes it difficult to justify the extra testing required to use the Wang-Brown algorithm.

Summary
For ductile metals, including most commonly used steels, a fatigue analysis using principal stresses or strains can give very unsafe life estimates. Further, Mises stresses do not always provide the correct fatigue cycles because Mises stresses are always positive.

The principal strain (or axial strain) parameter gives reliable life estimates for brittle metals. It does not require biaxial test data and can be used for uniaxial stresses. It is particularly suitable for cast irons and very high strength steels.

The Brown-Miller equation, using combined shear and normal strains, gives reliable life estimates for a wide range of materials and biaxial stress conditions. It does not require biaxial test data. It may be used for uniaxial stresses equally well. The Brown-Miller and principal strain criteria are attractive:

- because of their proven accuracy
- because they do not require the user to decide whether to use a uniaxial or biaxial analysis
- because they do not require biaxial materials properties

The Wang-Brown criterion requires additional multiaxial materials test data. Safe Technology does not recommend this method. Most companies do not have multiaxial fatigue data and would require additional (and expensive) materials testing to get it. Nor does this method produce as accurate a life estimate as a result of the additional materials data.

Implementation in fe-safe
For the reasons discussed, the major biaxial fatigue algorithms provided in fe-safe software are:

- Principal (or axial) strain, with mean stress corrections for brittle metals, including special modeling procedures for cast irons
- Brown-Miller combined shear and normal strain, with mean stress corrections, for ductile metals

The fe-safe software also includes the following approaches not discussed here but which are appropriate for special situations:

- Dang Van criterion for very high cycle fatigue design
- Principal (axial) stress for analysis of welded joints

To Learn More
To register for the web seminar replay held in cooperation with Safe Technology Ltd. visit **www.abaqus.com/webinar-replays**.
To learn more about fe-safe software visit **www.abaqus.com/products/fe-safe**.
For more in-depth information on fatigue, including more complete papers on this topic with references, visit Safe Technology's website to register and download additional case studies and technical papers at **www.safetechnology.com.**

Note: We would like to thank Professor John Draper and Stephanie Wood at Safe Technology Ltd. for their support on this article.

ABAQUS Services Spotlight - *Optimizing the Design of Rayovac Batteries*

Spectrum Brands (formerly Rayovac Corporation) is a global consumer products company and a leading supplier of batteries, lawn and garden care products, specialty pet supplies, shaving and grooming products, household insecticides, personal care products, and portable lighting. Spectrum Brands' products are sold by the world's top 25 retailers and are available in more than one million stores in 120 countries around the world. Headquartered in Atlanta, Georgia, Spectrum Brands generates approximately $2.8 billion in annualized revenues and has approximately 10,000 employees worldwide.

Spectrum Brands has been using ABAQUS to design its Rayovac batteries for many years, and contacted an ABAQUS office to develop a new analysis methodology for the battery manufacturing process and to transfer the technology to in-house engineers for design optimization.

Understanding the Problem

During the manufacture of dry cell batteries, a nylon seal is secured within the battery housing using a crimping process. This seal is designed to withstand internal pressure without leakage until the pressure exceeds a threshold level where a controlled release is provided by a sacrificial vent feature. In the situation in which the vent remains inactive, de-crimping of the seal and negative end plate occurs.

The final manufactured state (also the final analysis state).

The battery of interest for this project was a Rayovac LR20 dry cell battery (size D). This battery is shown above in the final manufactured state, which is also the final state of the analysis model.

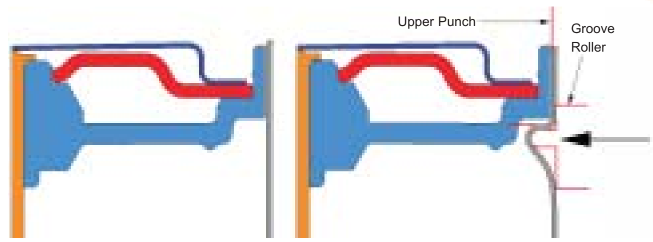
The initial state of the battery can with no groove.

In the figure to the right, we see the beginning state of the model, defined as a can with no groove or crimp.

The project developed by ABAQUS delivered a method that is straightforward and results indicate that the predicted de-crimp pressure is representative of actual pressures measured during physical testing. Battery component modeling included the can, gasket, bottom plate, washer, and collector nail.

Modeling the Manufacturing Process

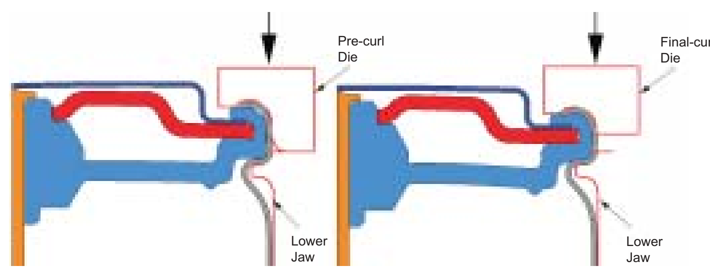
In addition to the battery components, the relevant tooling was also modeled. For groove forming, tooling included an upper punch and groove roller.



Groove Forming (Axisymmetric view).

Elastic-plastic material properties were used for all battery components modeled, which included both steel and nylon parts.

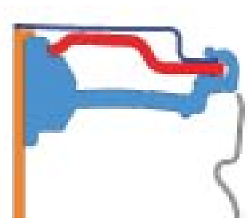
Crimping tools included a lower jaw, pre-curl die, and final curl die. All tooling was modeled using rigid surfaces since the tooling is stiff relative to the deformable battery.



Two Stage Crimping (Axisymmetric view).

Analysis Conditions

All analysis steps defined were displacement controlled except for the de-crimping step. That is, boundary conditions were used to define the groove rolling crimping tools movement. In the final de-crimping step, an overpressure was applied to the inside gasket surface. The overpressure load was gradually ramped from 0 to 1000psi, with de-crimping occurring before the full 1000psi magnitude was reached.



Internal pressure is applied until de-crimping occurs.

The Expert Touch - Analysis Challenges

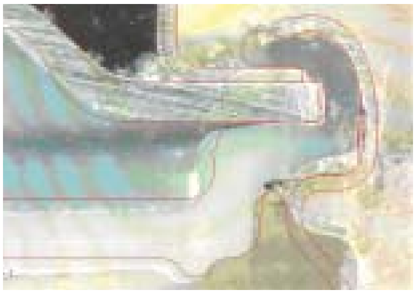
This seemingly simple analysis actually involves a number of very complex considerations. For example, this problem requires a 'real-life' representation of the contact conditions within the battery casing. This applies both during the forming process and the de-crimp analysis and requires the resolution of initial contact and an interference fit.

In addition, the forming process requires ABAQUS-specific modeling techniques to engage and disengage contact from the dies and forming tools.

In order to keep the problem stable and maximize efficiency, ABAQUS engineers employed appropriate solution controls, utilizing parameters such as time incrementation and contact tolerance controls. These advanced ABAQUS features allow the expert to tune the analysis for efficiency without any loss in accuracy.

Results

One result of interest from the analyses was the deformed shape of the battery. A qualitative assessment of results can be made by comparing the deformed shapes with actual shapes known from experience. The illustrations on the previous page highlight the deformed shape of the battery can at various stages of the analysis. The illustration below shows the comparison between an actual LR20 cross-section and the predicted LR20 cross-sections after forming.



Overlay plot of analysis versus actual formed can.

Cross-sections predicted by ABAQUS compare well with the actual resulting cross-section shape. The upper crimped portion is close to actual and gaps between components in the actual cross-section are also found in the predicted cross-section.

Conclusion

Working closely with engineers at Spectrum Brands, ABAQUS Central, Inc. successfully developed a new method for simulating the forming of a battery and for predicting de-crimp pressure. The analysis method developed is straightforward and the predicted de-crimp pressure fell within the typical de-crimp pressure measured, thereby providing a preliminary analysis procedure validation.

The models and methodologies developed will assist the engineers at Spectrum Brands in predicting the behavior of new designs, allowing for a faster, more efficient product development cycle.

Note: We would like to thank Spectrum Brands and ABAQUS Central, Inc. for their support with this article.

About ABAQUS Services

In addition to traditional consulting on design and analysis, ABAQUS offices and representatives offer several services tailored to suit specific needs across a variety of industries.

- **Methods Evolution & Development**
  Assist in migration to ABAQUS to optimize analysis processes and methodologies.
- **Process Automation**
  Customize ABAQUS/CAE to allow engineers greater access to FEA techniques and shorten the development cycle.
- **Performance Audits**
  Match typical analyses to the latest analysis techniques and code developments to optimize processes and reduce turnaround time.
- **Analysis Prequalification**
  Work directly with in-house QA departments to reduce or eliminate the time it takes to internally qualify a particular version of ABAQUS.
- **On-Site Support**
  Deliver in-house support by providing ABAQUS engineers to work on-site for an extended period of time. This provides an unparalleled level of support, methods optimization, innovation, and knowledge transfer. On-Site Support offers fast payback and immediately tangible results.

To learn more, please visit **www.abaqus.com/Services.**

Pedestrian Protection Impactors

Applus+ IDIADA Automotive Technology Validates Adult Headform FE Model Using ABAQUS/Explicit

In the European Union, over 7,000 pedestrians die each year in road accidents, and many more are injured. Pedestrian protection is an important aspect of new passive safety requirements for EU automakers. In order to reduce the severity of injuries to pedestrians in vehicle-pedestrian collisions, the European Commission has discussed making pedestrian protection mandatory for all new vehicles. EC Directive 2003/102/EC specifies the tests that automakers must perform to certify vehicles for pedestrian protection.

Self-commitment by the automotive industry is expected as of October, 2005, based on the requirements of test procedures recommended by EEVC (European Enhanced Vehicle-Safety Committee) Working Group 17. The requirements for pedestrian protection in the EU are to be met in full by all new vehicles as of 2010. As a final milestone, those requirements are to be met in full by all vehicles as of 2015.

Applus+ IDIADA has a comprehensive facility for automotive testing.

Similar standard procedures for assessing pedestrian safety have already been introduced as part of the Euro NCAP (New Car Assessment Programme) initiative. The mission of Euro NCAP is to serve as an independent evaluator of automotive safety performance and to encourage safety improvements in the design of new cars sold in Europe (**www.euroncap.com**). Euro NCAP announced several safety ratings in June, 2005. However, so far the maximum score obtained in Euro NCAP pedestrian protection tests is 3 out of 4 stars. Although car manufacturers are working hard in this field, most of them are still at the early stages of development, and the reality is that few vehicles in the market comply with the current specifications for pedestrian protection.

Experts in Passive Safety

Originating in 1971 as a research institute of the Polytechnical University of Catalonia, IDIADA became an independent company owned by the government of Catalonia in 1990. IDIADA Automotive Technology SA was created in 1999, with Applus+ Corporation as majority shareholder. Headquartered in L'Albornar (Tarragona), Spain, Applus+ IDIADA currently employs 500 people and operates subsidiaries and branch offices in 11 different countries. With expertise in global vehicle development, the company offers engineering, testing, and homologation services to the automotive industry worldwide.

Applus+ IDIADA serves as a test house for Euro NCAP. (Photo on right courtesy of SEAT)

Applus+ IDIADA is an official Euro NCAP test house and works directly with automotive OEMs across the world. Its multidisciplinary engineering team has many years of experience and know-how in the field of pedestrian protection. The company offers one of the most advanced laboratory facilities in Europe for testing vehicle pedestrian safety, including the Dynamic Impact Test System, which can carry out a wide range of protocols. Some of the best scores in Euro NCAP's pedestrian protection tests have been achieved as a result of collaborations with Applus+ IDIADA testing and CAE support.

Pedestrian Protection Impactors

In a road accident, a pedestrian's head and lower extremities are the most frequently injured body regions. EEVC procedures define five impactors for testing facilities to use when studying the physical forces involved in pedestrian impacts with a moving vehicle: adult headform, child/small adult headform, child headform, lower legform, and upper legform.

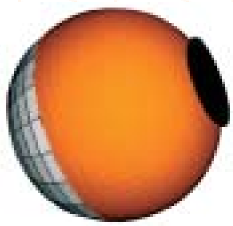
Pedestrian protection impactors.

Computer simulation has the potential to help car manufacturers develop vehicle bodies that comply with procedures for protecting pedestrians. However, to obtain good results in simulations, prior validation of the FE models for each of these pedestrian impactors is necessary. Several pedestrian protection projects are currently underway at Applus+ IDIADA. One of them is the full development of pedestrian impactors with the use of ABAQUS/Explicit code. In a technical paper presented at the 2004 ABAQUS Users' Conference, Applus+ IDIADA described the methodology used in the development and validation of the FE model of the adult headform impactor. This methodology is based on Applus+ IDIADA knowledge and experience gained from previous development of virtual pedestrian impactors using well-known explicit codes.

Simulation of adult headform over full vehicle. (Courtesy of SEAT)

The Adult Headform Impactor

The adult headform impactor consists of an aluminum core covered by a thick vinyl skin. The device has an outer diameter of 165mm and a total weight of 4.8 kg. A triaxial accelerometer is mounted inside, located at the center of gravity. The aluminum core components of the adult headform impactor are extremely stiff compared to the vinyl skin. Consequently, the core of the FE model can be represented as a rigid body. However, it is necessary to define the mass and inertia properties of the aluminum core about a local coordinate system. The thick vinyl skin is modeled using solid elements (C3D8R) connected to the aluminum core by means of a kinematic coupling between the base of the vinyl skin and the reference node of the rigid body. As the analysis progresses, the history of headform acceleration is monitored at a node located at the center of gravity of the complete adult headform.



FE model of the adult headform generated for certification tests.

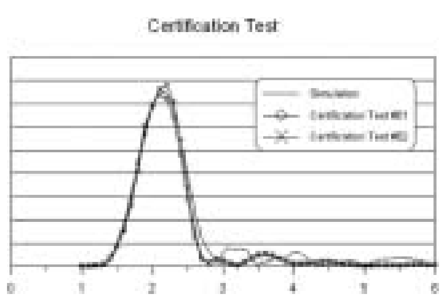
Requirements for an Accurate and Effective FE Model

For simulation to be a powerful predictive tool for vehicle development, the simulation results obtained should match real-life tests. Validation of the adult headform FE model was needed to ensure that the model would accurately predict results recorded by the real-life impactor during safety tests. The main objective was to validate physical tests performed at Applus+ IDIADA facilities. The validation process had two phases: a certification test phase and a realistic test phase. Both test phases yielded feedback for further development of the adult headform FE model.

Validation of Adult Headform Certification Tests

New European directives define standard test procedures that should be performed to certify impactors for pedestrian protection tests. These kinds of tests are called certification tests. In certification tests for the adult headform impactor, the device is suspended with the rear face at an angle between 25º and 90º with the horizontal. The certification impactor arm on the test apparatus is then propelled at a velocity of 10.0 m/s into the stationary adult headform.

Validating the certification tests for the adult headform required the following steps: (1) perform certification tests at Applus+ IDIADA facilities, (2) simulate certification tests (including generation of impactor geometry and FE model), and (3) validate certification tests and refine FE model. The final headform virtual model correlated with the certification test results.



Validation of certification test (acceleration versus time in ms).

Validation of Adult Headform Realistic Tests

After validating the certification test, it was necessary to study the impactor under quasi-real circumstances. Consequently, a realistic test was defined in order to validate the FE model already validated for certification tests. These realistic tests represent the test procedure of a pedestrian impact against a vehicle.

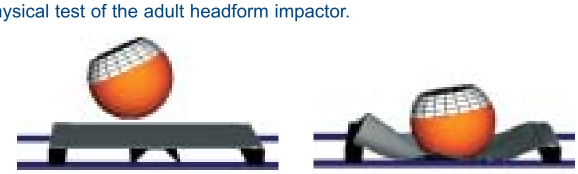
Using the adult headform impactor, realistic test impacts were performed against a flat metal sheet with a metal reinforcement on the underside. This configuration represents a hypothetical car bonnet and its underside reinforcement.

To measure the head injury that would result from a vehicle-pedestrian collision, the EC Directive specifies the following metric:

> 'Head Injury Criterion (HIC)' shall be calculated from the resultant of accelerometer time histories as the maximum (depending on t1 and t2) of the following equation:
>
> $$HIC = \left[\frac{1}{t2 - t1} \cdot \int_{t1}^{t2} a(t)\,dt \right]^{2.5} \cdot (t2 - t1)$$
>
> where 'a(t)' is the resultant acceleration as a multiple of 'g', and t1 and t2 are the two time instants (expressed in seconds) during the impact, defining the beginning and the end of the recording for which the value of HIC is a maximum. Values of HIC for which the time interval (t1 - t2) is greater than 15ms are ignored for the purposes of calculating the maximum value.

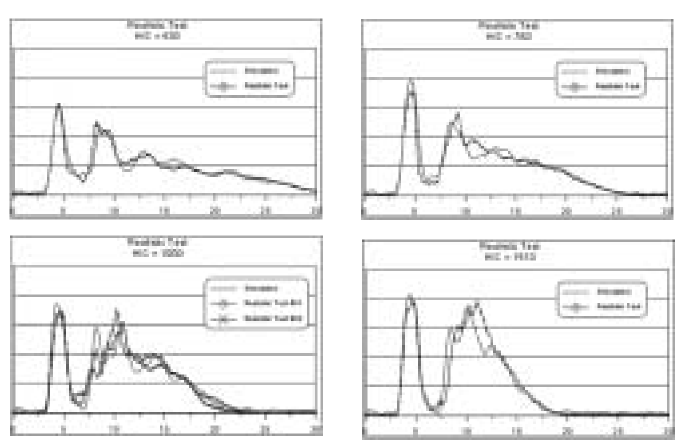
At present, the target range for pedestrian protection is an HIC value that is around 1000. Applus+ IDIADA performed realistic tests in four different configurations to obtain representative HIC values around four targets: 500.0, 750.0, 1000.0, and 1500.0.

Physical test of the adult headform impactor.



Realistic simulation of the adult headform impactor.

Validating the realistic tests required the following steps: (1) perform realistic test at Applus+ IDIADA facilities, (2) simulate realistic test, and (3) validate realistic test and refine FE model. The final headform virtual model correlated with realistic test results for all four HIC values.



Validation of realistic tests (acceleration versus time in ms).

continued on next page

By achieving validation objectives for the adult headform, Applus+ IDIADA obtained an accurate and effective virtual tool using ABAQUS/Explicit in developing vehicle passive safety.

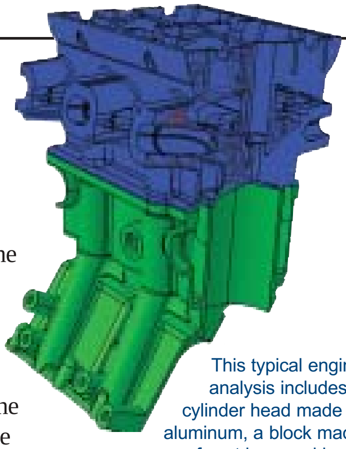
To Learn More

To download the complete contributed paper given by Applus+ IDIADA at the 2004 AUC, visit **www.abaqus.com/Insights0508.**

About the Authors

Pablo Cruz and Carles Mitjans are project managers in the CAE Department at Applus+ IDIADA Automotive Technology. Pablo holds a degree in mechanical engineering from the School of Industrial Engineering of Barcelona (ETSEIB) of the Technical University of Catalonia (UPC). Carles holds a degree in mechanical engineering from the School of Industrial Engineering of Vilanova i la Geltrú (EPSEVG) of the Technical University of Catalonia (UPC).

Engine Thermal-Mechanical Analysis Using ABAQUS for CATIA V5



This typical engine analysis includes a cylinder head made of aluminum, a block made of cast iron, and head bolts made of steel. Material properties include plasticity definitions, as well as temperature-dependence.

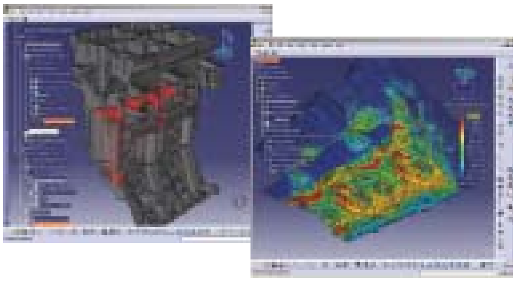
Working closely with our customers, ABAQUS pioneered many finite element capabilities that enable some of the most important analysis workflows for internal combustion engines. These sorts of simulations are conducted routinely by engineering specialists, albeit with a range of specialist tools and often requiring multiple modeling and visualization steps. One of these workflows, engine thermal-mechanical analysis, was identified by our ABAQUS for CATIA V5 Customer Review Team as a critical capability for this product. The discussion below briefly describes this sophisticated workflow, which can be conducted in a straightforward manner, all from within ABAQUS for CATIA V5.

Objectives and Goals

During engine design, the purpose of FEA is to evaluate engine designs and indicate possible improvements. One of the most important goals of the analyses is to determine areas of maximum stress in order to provide an understanding of engine durability.

Another important goal is to find regions that experience plastic deformation under operating conditions so that the design can be modified to eliminate or minimize such unwanted permanent material deformations. Obtaining an accurate understanding of the stress and deformation state of an engine requires nonlinear FEA analysis.

The first part of solving the problem requires performing a thermal analysis to determine the temperature distribution throughout the engine. The loads for the thermal analysis include the hot gases in the cylinders, as well as the flow of cooling liquid through channels that wrap around the cylinders. With the thermal analysis solved, the analyst can move on to the multi-step stress analysis, which uses the temperature distribution results from the completed thermal analysis as input.



Analysis Steps

The stress analysis consists of three sequential analysis steps, each using the end state of the previous step as its starting point. The first step in the stress analysis is a bolt-pretensioning step, which induces the assembly preloading of the engine, including large contact forces at the interface of the block and the head. An operating load step follows. During this step, the temperature distribution from the thermal analysis is applied along with additional mechanical loads, including explosive pressures in the combustion chambers. The final step involves cooling the engine to room temperature and removing operating loads. Cooling down at varying rates can induce additional plastic strains in some engine designs. If necessary, the service loading and cool down sequence can be repeated several times to provide additional information about the behavior of the engine after several loading cycles.

To Learn More

To download a 17 minute video (.avi file), showing how ABAQUS for CATIA V5 is used to conduct this analysis workflow, visit the web page for this issue of ABAQUS Insights at: **www.abaqus.com/Insights0508.**

To see how ABAQUS for CATIA V5 is being used in the aerospace industry, you may also visit the same web page to download presentations given by ABAQUS and Boeing at the 2005 ABAQUS Users' Conference this past May.

ABAQUS In the News

Recently, several ABAQUS customers and employees were featured in major articles in a wide range of engineering publications. Several of these noteworthy articles are summarized below. To read the full stories, please visit ***www.abaqus.com/news*** *and click on "In the News."*

Aerospace Engineering
July 2005, pp. 28-29
Crack Prediction Down Under
The Royal Australian Air Force needed to determine whether metal fatigue would reduce the safe life of a rotating turbine component used in its fleet of large military transports. "In aging aircraft," notes editor David Alexander, "low-cycle fatigue often reduces the useful life of components." This case study describes how the Defence Science and Technology Organisation, an independent branch of the Australian Defence Organisation, used ABAQUS/Standard with Zencrack to model crack propagation in the part. The analysis showed that fatigue or the appearance of a crack would not significantly alter part strength or life.


simulation on the spot

Making the Most of Boundaries—Fluid-Structural

Industry Week
March 2005, p. 23
The Analysis Challenge
For a column on Emerging Technologies, editor John Teresko interviewed Dale Berry, Manager of Engineering Applications at ABAQUS. The article leads off with the question, "How do you want your product to fail?" Dale explains why determining fracture and failure modes is important for manufacturers and describes the effort ABAQUS is making to create software for routine use. "We see a continuing need for advanced yet easy-to-use capabilities that will allow our customers to include fracture and failure in their everyday simulations," he says.

Machine Design
April 2005, pp. 90-95
The Glued-Together Car Body
Dow Automotive used ABAQUS/Explicit to model the behavior of BETAMATE™ crash-stable adhesives in automotive structural joints. In this case study based on an AUC paper, Dr. Colmar Wocke explains how toughened epoxy dissipates energy during a crash and compares predictions and test results for the material versus spot welds and brittle adhesive. "Simulations give significant engineering insight into the details of the toughening mechanism," Dr. Wocke writes, "and will help chemists improve adhesive formulations down the road."

Design News
May 2005, pp. 87-92
Bulletproof Analysis
In this article, Leopoldo Carbajal and Ted Diehl, Ph.D., of DuPont Engineering dig deep into the details of modeling the impact strength of woven fabric using ABAQUS/Explicit. Analyzing the design of a storm shelter gave them a headstart on the next project: simulating the behavior of bulletproof vest packs. "Our model was able to capture the behavior of a bullet turning nearly inside-out upon impact," they report. This story follows their adventurous use of ABAQUS to address a host of challenging technical issues for analysts.

Desktop Engineering (Elements of Analysis Special Supplement)
July 2005, pp. 8-10
Simulation on the Spot
BD Technologies practices close collaboration between analysts and engineers early in product development. In this how-to article, Anita Bestelmeyer, Analysis Manager at BD, steps through a shelf-life analysis for a sealing application. She also describes how her company uses desktop-sharing tools and ABAQUS/CAE to support real-time communication and problem solving throughout design. "We can easily swap alternative geometries in and out of analysis models, so the engineers can try creative solutions when making decisions," she says.

Desktop Engineering
July 2005, pp. 18-24
Making the Most of Boundaries: Fluid-Structural Interaction
Writer Louise Elliott explains the importance of the ABAQUS-Fluent partnership and describes how MpCCI technology from Fraunhofer SCAI supports a coupled analysis approach. Dale Berry, Manager of Engineering Applications at ABAQUS, comments on new developments in analysis software for fluid-structure interaction.

Briefly Noted
MCAD Magazine published an online review of ABAQUS/CAE 6.5 in March, 2005 (**www.cadserver.co.uk/common/viewer/archive/2005/Mar/14/news7.phtm**). The Spring 2005 issue of Body Engineering includes an article titled, "ABAQUS Has a Vision - Unified FEA," by Mark Bohm, General Manager of ABAQUS Great Lakes.

ABAQUS, Inc.
Rising Sun Mills
166 Valley Street
Providence, Rhode Island 02909-2499
+1 401 276 4400
e-mail info@abaqus.com

Europe
+31 43 356 6906
e-mail info.europe@abaqus.com

Japan
+81 3 5474 5817
e-mail info.japan@abaqus.com

WWW.ABAQUS.COM

Industry Events

Offshore Europe
September 6–9, 2005
Aberdeen, Scotland
www.offshore-europe.co.uk

2005 GM Global CAE Conference
September 12–14, 2005
Warren, MI

AIAS
September 14–17, 2005
Milan, Italy
aias05.mecc.polimi.it

24th Annual Conference on Tire Science and Technology
September 20–21, 2005
Akron, OH
www.tiresociety.org/mainpages/conference.htm

SAE Aero Tech Congress and Exhibition
October 3–6, 2005
Grapevine, TX
www.sae.org/events/atc

European CATIA Forum
October 4–6, 2005
Stuttgart, Germany
www.ecforum.com

CAT-Pro Messe 2005
October 4–7, 2005
Stuttgart, Germany
www.messe-stuttgart.de/CAT

COE Your Way: Automotive Industry Workshop
October 10–11, 2005
Dearborn, MI
www.coe.org/events/automotive/index.cfm

NAFEMS Wiesbaden
October 26–27, 2005
Wiesbaden, Germany
www.nafems.de

76th Annual Shock and Vibration Symposium
October 30–November 4, 2005
New Orleans, LA
www.saviac.org/76th_Symposium/76th_symposium.htm

Procter & Gamble 2005 Accelerate Symposium
November 2–3, 2005
Cincinnati, OH

Medical Design & Manufacturing Minneapolis
November 2–3, 2005
Cincinnati, OH
www.mdm-minneapolis.com

MAR_INS_0508