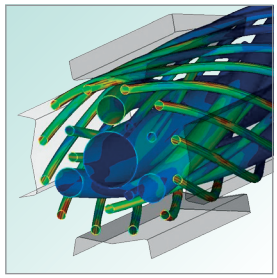
INSIGHTS

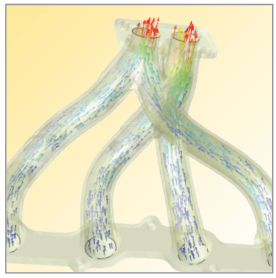
10
6 2010
Dassault Systèmes Realistic Simulation Magazine



HCL Analyzes Paper Jams with Abaqus FEA



Technip
Custom Umbilicals for Deep Offshore Wells



Abaqus 6.10
New Capabilities for Fluid-Structure Interaction and More

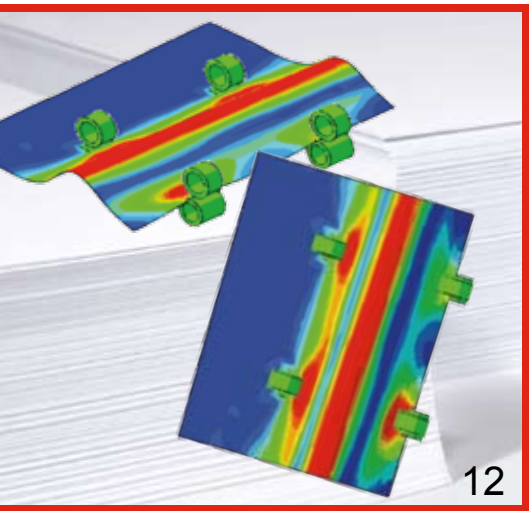


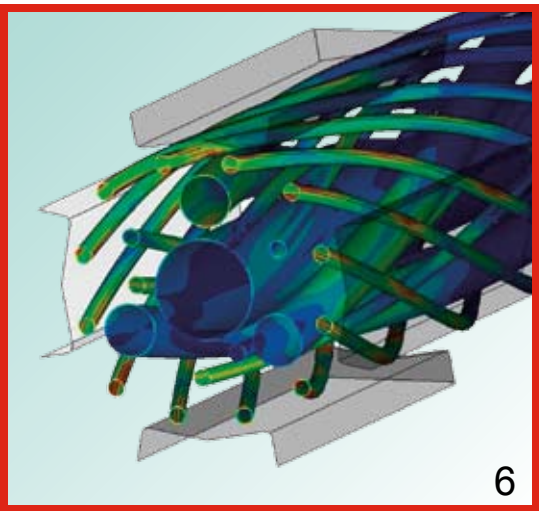
Amcor
Light-Weight Containers with FEA

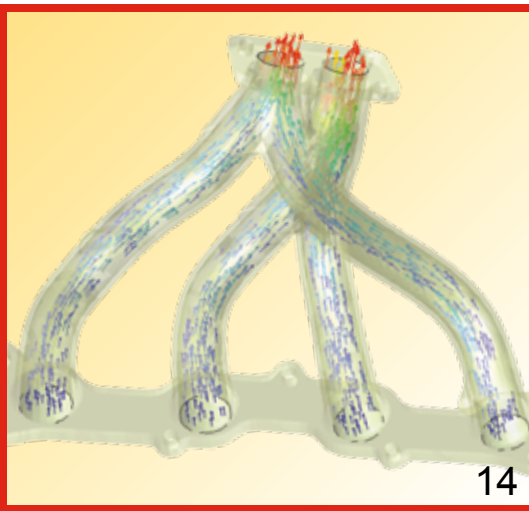
SIMULIA

INSIGHTS

May/June 2010







Inside This Issue

12 Cover Story
HCL Analyzes Paper Jams with Abaqus FEA

On the cover: (left to right) Mayilvaganan Thangavel and Venkata Mahesh

6 Customer Spotlight
Technip Designs Custom Umbilicals for Deep Offshore Wells

14 Product Update
Abaqus 6.10

In Each Issue

3 Executive Message
Scott Berkey, Chief Executive Officer, SIMULIA

4 Customer Viewpoint
Dr. Bijan K. Shahidi, Principal Consultant, Engineering Products, Inc.

8 Electronics Strategy Overview
David Cadge, Electronics Lead, SIMULIA

10 Solution Brief
SLM Cuts Qualification Process for Crash Dummy Models

15 Product Update
- Isight 4.5
- SIMULIA Execution Engine

16 Customer Case Study
Amcor Uses Realistic Simulation to Stay on Top in Plastic Container Market

19 Alliances
- Bodie Technology
- Cray Inc. at University of Alabama

20 Academics
- University of Idaho
- San Jose State University

22 In The News
- InnerPulse
- BMW Group

23 Events
2010 Regional Users' Meetings

INSIGHTS is published by
Dassault Systèmes Simulia Corp.
Rising Sun Mills
166 Valley Street
Providence, RI 02909-2499
Tel. +1 401 276 4400
Fax. +1 401 276 4408
simulia.info@3ds.com

www.simulia.com

Editor:
Tim Webb

Associate Editor:
Karen Curtis

Contributors:
Mayilvaganan Thangavel and Venkata Mahesh (HCL), Bijan Shahidi (Engineering Products, Inc.), Ian Probyn and Dave Fogg (Technip Group's DUCO Ltd.), Ted Diehl (Bodie Technology), Tim Masterlark (University of Alabama), Ahmed Abdelnaby (University of Idaho), Nandini Nagendrappa (San Jose State University), Parker Group, David Cadge, Scott Berkey, George Scarlat, Sridhar Sankar, Eric Weybrant, Asif Khan, Alex Van der Velden

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, 2010.

JUNE_INS_Y10_VOL 10



The New Decade Ahead – Focusing on Customer Success and Connecting Our Global Communities

In my last letter for *INSIGHTS* in October 2008, I stated that Dassault Systèmes SIMULIA was well positioned to increase the business value of realistic simulation technology and that our business momentum was based on the fundamental principles of technology innovation and customer satisfaction. I am pleased that this statement continues to hold true and, as we enter a new decade, more relevant than ever. Our strategy of providing robust simulation technology and excellent technical support has helped sustain and drive the business value of realistic simulation gained by our customers..

During the past few years, we have expanded our product portfolio which now includes Abaqus Unified FEA, Isight, and solutions for Simulation Lifecycle Management – our R&D team is also responsible for development of the DesignSight, SolidWorks Simulation, and CATIA Analysis products. We remain committed to enhancing the core mechanics technology of Abaqus, even as we expand our portfolio and add new multiphysics capabilities such as computational fluid dynamics in Abaqus 6.10. The latest release of Abaqus, demonstrated during the recent SIMULIA Customer Conference, provides improvements in fracture and failure, linear dynamics, performance, modeling and visualization (page 14).

The business value of realistic simulation is also being driven by an increasing number of users of our realistic simulation solutions. I believe the growth of our user community is a direct result of our close working relationships with customers and our focus on developing innovative technical capabilities that support industry-specific workflows. This combination is enabling more users to leverage realistic simulation technology on a wider range of industry applications such as BMW's use of Abaqus for automotive safety and crashworthiness (page 22), Amcor evaluating reliability of product packaging (page 16), InnerPluse analyzing the potential of new medical devices (page 22), and HCL analyzing the behavior of paper feeding for printers (page 12).

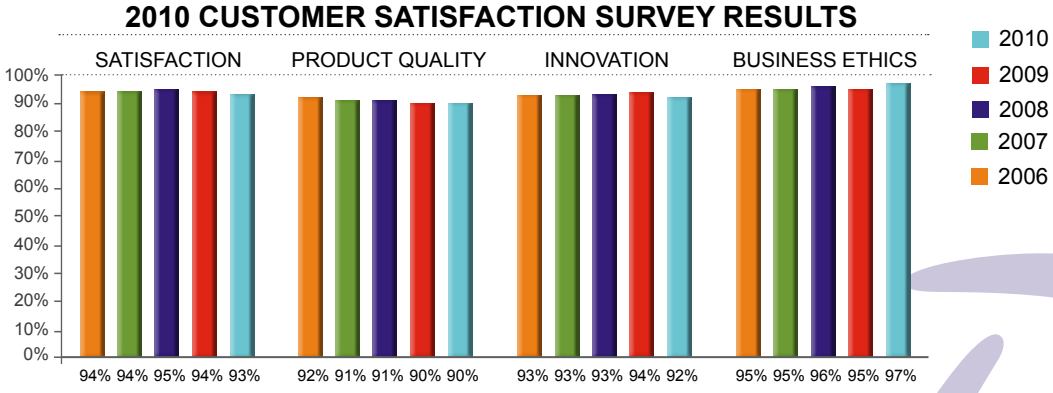
I would like to thank the more than 1,200 customers who recently took time to respond to our annual customer survey. Your feedback is ensures that we continue to meet your requirements and expectations. I am pleased to report that, while there is still room for improvement, the overall results of the survey indicate that our customers continue to be highly satisfied with the quality of our products and business practices.

Your participation at the international SIMULIA Customer Conference and Regional User Meetings (page 23) offers an opportunity to interact with SIMULIA and our vibrant community of users. Later this year, the RUMs will be held in more than 30 cities all over the world. The meetings will enable you to learn about the new capabilities in our realistic simulation solutions, discover how your peers are using realistic simulation to accelerate product development, and provide input on your requirements directly to SIMULIA management.

I look forward to meeting as many of you as possible in the next few months as we continue our strategy of developing innovative simulation technology, providing excellent technical support, and delivering quality products and services that ensure customer satisfaction and success.



Scott Berkey
Chief Executive Officer,
SIMULIA

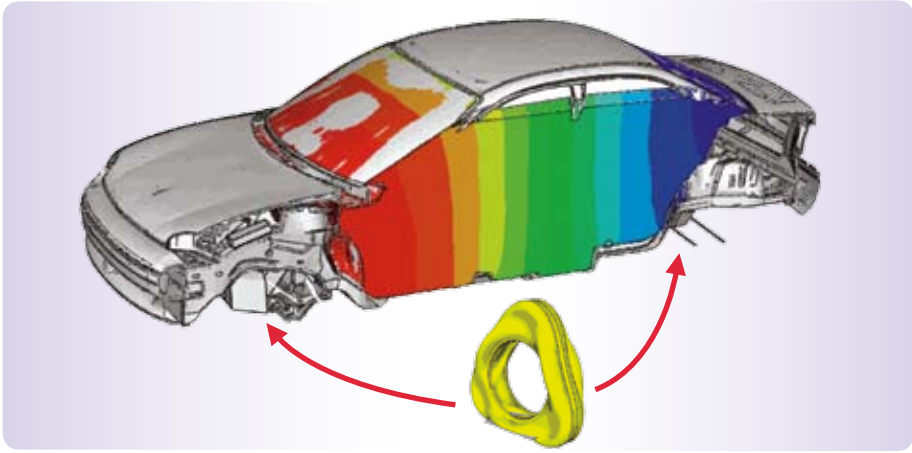


Evolution of Finite Element Analysis Helps Fine-Tune Product Development

Simulation expertise from automotive now benefits other industries

Dr. Bijan K. Shahidi, Principal Consultant, Engineering Products, Inc.

Take a good look at the automobile of today: Despite the overall downturn in consumer demand from that of 3 to 4 years ago, when overall U.S. new car and truck production reached volumes in excess of 16 million, to about 10 to 11 million adjusted annually—and still there are the serious economic challenges across the industry before full recovery—vehicles have clearly become quieter, safer, and more durable over the past decade. Many of these improvements came about partly due to increasing sophistication in the design and simulation tools available to engineers, particularly finite element analysis (FEA) software.



The usage of stationary or rolling tire substructures in the full vehicle N&V simulations.

In my years as a vehicle computer-aided engineering (CAE) manager at a major automotive OEM, I experienced the growth of FEA in both capability and applications. The software helped my design and development teams model, simulate, and analyze the behavior of automobiles under a wide range of loads and conditions. Our expertise evolved along with FEA, going from classic linear contact simulations to complex, nonlinear analyses such as full-body noise and vibration (N&V) complete with rolling tires, wind loads, and more.

These advances in FEA enabled our engineers to identify, earlier in the design phase than ever before, any necessary fixes needed to achieve consumer and regulatory targets. The CAE models' fidelity became so good that over time we found less and less need to build physical prototypes for validation and comparison. High-quality simulation results helped us present realistic cost forecasts to our senior management before they signed off on any production go-ahead.

Such simulation-driven advantages will certainly continue to prove their worth to the automotive industry as it focuses on vehicle redesign as the foundation of its revival strategy; however, the future potential of CAE is certainly not limited to automakers. Given the economic environment of today, manufacturing companies across the board are being pressed more than ever to improve engineering efficiency, lower development costs, and accelerate product innovation. Simulation will play an increasingly prominent role in helping every industry achieve those goals, and the evolution of FEA continues. With the release of each new version, simulation software enables engineers to get ever closer to truly realistic behavior through the refinement of nonlinear effects of materials including rubber, plastics, metal, and composites. Simulation results are also sharpening the engineer's vision by incorporating other nonlinear mechanical attributes embedded in a product due to the design, materials or manufacturing process—such as pre-stress, pre-strain, and/or pre-stiffness. The result is a thorough understanding of physics at play, less prototyping and testing, and much better correlation between FEA results and the real-world.

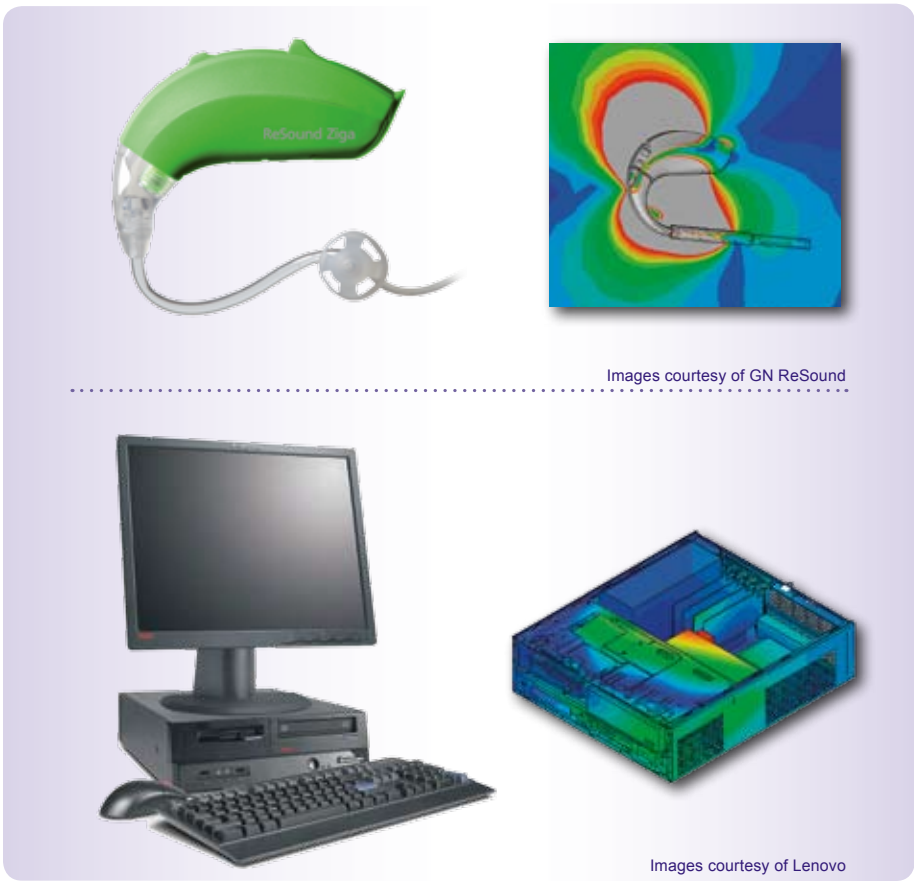
Although faster, cheaper, more powerful computer hardware is certainly supporting the evolution of FEA, the software itself is becoming so well integrated that the latest versions can run certain complex problems on fewer computer cores, rather than more. The sophistication of the software is also allowing engineers to create models with increasingly finer meshes; therefore, more accurate fidelity. As compute power increases, the models become finer and finer and the software becomes better aligned, resulting in better all around integration. Eventually, engineers won't see anything—except 3D "reality"—at all.

The automotive industry has certainly been one of the major staging grounds for large-scale use of FEA. The application of N&V analysis alone has grown tenfold in the last decade. Automotive engineers eager to share their results have spread the news of these capabilities at regional and global engineering conferences. The CAE lessons learned in automotive are feeding into other industries including construction, heavy vehicle, military, off-highway, aerospace, and shipbuilding engineering.

Newer mechanized industries, such as life sciences, are learning that the design challenges they face in N&V control of machinery and devices—such as oxygen delivery breathing apparatuses or hearing aids—can also benefit from such knowledge. The overall downturn adjustment in labor force in the automotive industry will likely cause an acceleration of N&V knowledge transferred to medical, pharmaceutical, and other industries that are in a better position to hire skilled engineers.

So what's in the future for simulation within automotive engineering, particularly in the U.S.?

First, the explicit FEA methods that have become part of crash and safety simulations



(Top) A 2-D FEA simulation is used to model and then verify sound pressure levels in and around a hearing aid design, helping engineers to improve hearing aid performance while shortening development time. (Bottom) Realistic simulation is used to compute the natural frequencies of a desktop computer enabling engineers to determine which parts to modify and where to add damping materials in order to make the computer quieter and more durable.

in recent decades (and have played a major role in helping engineers to design safer cars and trucks) will be increasingly applied to applications. This advancement will result in a greater demand for even faster computing hardware and better integration of implicit and explicit FEA techniques to help design lighter-weight, efficient, yet comfortable vehicles of tomorrow.

Second, and admittedly further off in the future, is the advent of isogeometric analysis (a technique using NURBS and T-Splines as a basis for constructing element shape functions) championed by Prof. Tom Hughes at the University of Texas. This approach heralds a whole new way of thinking about how finite elements are created which seeks greater accuracies to compute stresses, velocities, pressures, and buckling loads.

Those automotive companies willing and able to continue investing in realistic simulation technology and skilled staff will certainly benefit as they develop the next generation of Green vehicles. But the cat is out of the bag; engineers who excel at using N&V simulation technology will be in high demand, not only in the automotive industry, but also as innovation leaders needed to apply their knowledge of FEA across many other industries.



Dr. Shahidi is currently Principal Consultant for Engineering Products, Inc. and has over 20 years of experience as a professional engineer specializing in CAE. He holds a Ph.D. in Theoretical Mechanics and an M.B.A from Michigan State University and holds an M.S. and B.S. in Aerospace Engineering from the University of Michigan.

For More Information
bijan.shahidi@gmail.com

Technology Briefs

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

High Fidelity Anti-Lock Brake System Simulation Using Abaqus and Dymola

A co-simulation approach using Abaqus and Dymola is used to achieve a realistic system-level simulation of an anti-lock brake system (ABS). The tire, wheel, brake caliper mechanism, and road are simulated with a detailed Abaqus finite element model while the brake system control algorithm and hydraulics and are simulated with Dymola.

Modeling the Interaction of Subsea Pipelines with the Seabed

The understanding and prediction of the interaction of a subsea pipeline with the seabed is a complex phenomena crucial for subsea pipeline design. This Technology Brief describes how the Coupled Eulerian-Lagrangian (CEL) method in Abaqus/Explicit can be used to calibrate the parameters that define the pipeline-soil frictional behavior. These parameters are then used in the pipeline-soil friction user subroutine in Abaqus/Standard (available on Abaqus Answer 4094) as part of predicting the in-service buckling deformations of the pipeline.

Simulation of Adaptive Bone Remodeling

The long-term success of an orthopedic implant can be better predicted by including the adaptive bone remodeling process. This Technology Brief demonstrates the Abaqus/Standard implementation of one of the leading bone remodeling algorithms. User subroutine USDFLD is employed to capture solution dependent material properties, and the approach is used in the analysis of a total hip replacement design.

For More Information
www.simulia.com/techbriefs

Longer Life for Deep-Sea Lifelines

Abaqus FEA helps Technip engineers custom design umbilicals for deep offshore oil and gas wells

Umbilicals are the lifelines of deep-sea fields, connecting the well to the mother ship, offshore platform, or onshore terminal. They are critical—providing the power, control, communication, and fluid injection that keep deep-water wells healthy and pumping around the clock (see Figure 1).

Durability is essential whether the umbilical is hanging in the water column (dynamic), resting on the sea bed (static), or connecting important field infrastructure. That's because pressure and temperature extremes, wave and current action, and sour fluids all conspire to break, or at least damage, the umbilical and its contents.

Due to their important role in deep-sea hydrocarbon extraction, the cost of installation, the difficulty of on-site repair, and the expense of a field being down, umbilicals need to be designed and built to last. Typical umbilical design life is 25 years, but at Technip Group's DUCO Ltd.—considered the world leader in umbilical design and manufacture—they use the ISO standard and set design fatigue life at 10 times the design life. "For a 25-year design life, we design for 250 years in terms of fatigue," says Ian Probyn, senior engineer, R&D, at DUCO. "With offshore umbilicals, failure is not an option."

Deep-water installation

Building failure-proof umbilicals is difficult enough, but challenges of deep-water installation further complicate the task. Wound onto storage reels and then mounted on a specialized installation vessel, the umbilicals need to deploy in a highly controlled manner to reach a precise target on the ocean floor.

The umbilical is fed through a Vertical Lay System (VLS) that controls the unspooling by applying a holdback tension to the umbilical as it hangs from the ship. Four caterpillar tracks with V-shaped pads typically create the hold back tension, applying a radial crush force to the umbilical using friction to control the deployment.

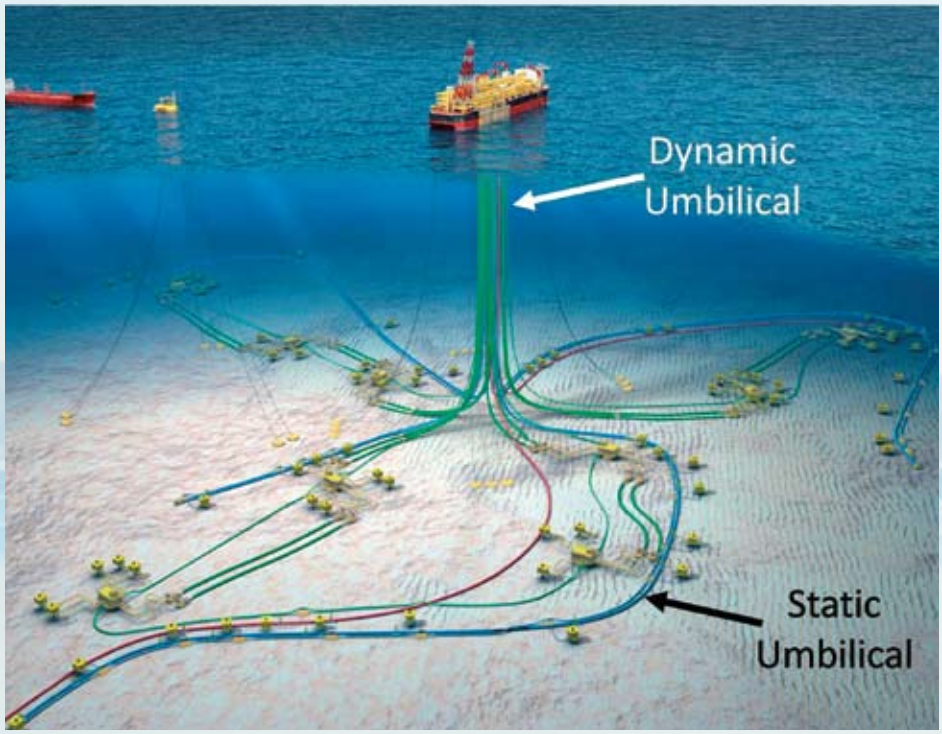


Figure 1. Umbilicals deployed from an installation vessel provide deep-water oil and gas fields with power, communication, and the necessary fluids required for hydrocarbon extraction.

As the depth of a deep-sea well increases, the tension and the crush load required to hold the weight of the lengthening umbilical also increase. Up to 30 tons per meter of radial load can be applied to a steel tube umbilical during deep-water installation.

Needless to say, this kind of pressure on the umbilical can cause deformation to the tubes, which have point contact (where tubes in adjacent counter-rotating layers cross) due to the umbilical's helical construction. DNV (Det Norske Veritas), Norwegian risk-management specialists, recommends that three percent residual ovality (permanent tube deformation following crushing) is acceptable; higher levels of deformation can negatively affect the umbilical tubes' resistance to hydrostatic pressure as depths increase. Residual ovality can also impair fatigue resistance to pressure cycling over time. As a result, understanding umbilical crush behavior in detail is critical to ensuring product integrity, establishing load limits, and designing out failure.

As projects get more expensive, there is more risk, and customers want more of the engineering work done up front. "Being able to prove that the design fits the purpose is critical," Probyn says. "With realistic simulation, we're able to see inside the umbilical. That's something you can't do with physical testing. FEA provides that level of detail."

Simulation customization for design flexibility

U.K-based DUCO first chose Abaqus FEA for their umbilical R&D in 2005. "We did an evaluation," says Dave Fogg, R&D team leader, "and Abaqus stood out because of its Explicit solver capability for analyzing highly nonlinear, dynamic behavior." This capability is important, he adds, given the helical structure of the umbilical, interaction between the components, and bending stiffness due to friction.

Abaqus also provides the ability to customize scripting tools. This customization is important because each client comes to DUCO with its own unique umbilical requirements. Since each product is essentially one-of-a-kind, the FEA tool and simulation process need to be flexible enough to accommodate this high degree of design variability.

DUCO, in collaboration with French-based IFP, a public-sector research and training center, developed a proprietary, validated engineering software tool—FEMUS or finite element model of an umbilical structure. This tool interrogates a database that includes all of the information required to build a model (think of the database as containing the DNA for any umbilical design, such as the component, material, and dimensional data). It then automates 3D-model building by gathering all of the data into a Python script (programmable language file used by Abaqus), which it then executes within Abaqus/CAE to create the FEA model.

Once the data is loaded in Abaqus, the script does the rest: It builds the umbilical-specific geometry; constructs the assembly; applies the section properties, element types and materials; and creates the load steps, contacts and request for history and field output data. Developed specifically for Abaqus and for use by non-FEA experts, the interface's goal was rapid model building using proven techniques. That goal was realized: In approximately 10 minutes the team can have a run-ready base model inside Abaqus.

Inside the umbilical

During the VLS installation, the umbilical is subject to tension, bending, and the crushing load from the caterpillar pads. For the crush load portion of the analysis, the DUCO team used the 3D model in Abaqus/Explicit to capture all of the interactions in the helically-oriented structure. The 3D analysis gave them the relationship between the crush load and the resulting ovality of the tube while under that load.

When the umbilical leaves the caterpillar, the crush load is relieved and the tubes elastically relax, resulting in a reduction of tube ovality. For the recovery of the tube, a simpler 2D analysis in Abaqus/Standard proved efficient. In the 2D environment, the team conducted a number of analyses for each tube and built up the relationship between the maximum ovality under load and the residual ovality of the tube following elastic recovery. They then combine the results from the 2D and 3D analyses to determine the overall residual ovality of a tube for a given caterpillar crush load.

For further efficiency, all analyses were run on models constructed from a single pitch of the umbilical—the length at which the helical pattern starts to repeat—which in this case was several meters. The team used shell elements for the tubes and solid elements for the polymer sheath, outer sheath, and fillers. For the crush pads, they used rigid elements and dimensions that matched the umbilical pitch length.

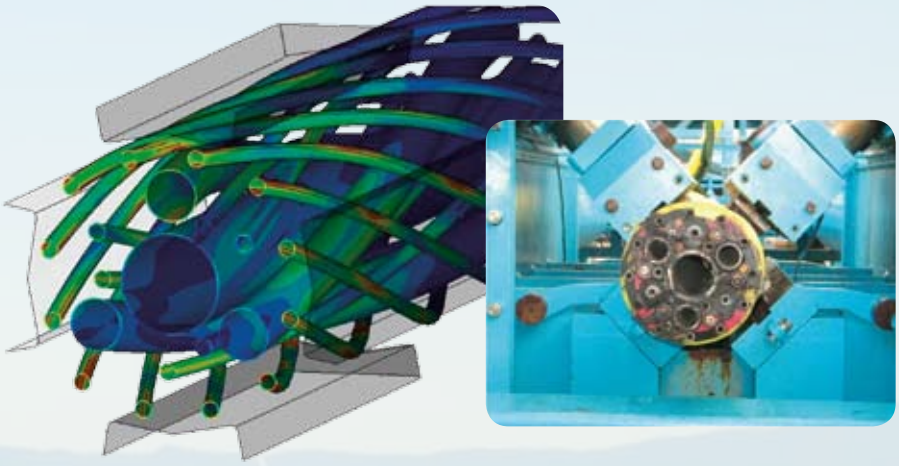


Figure 2. Comparison of the results of the umbilical FEA analysis (left) with the full-scale physical test using the four-track caterpillar crush rig (right) is used to validate the umbilical's residual ovality following the application of varying crush loads.

In the umbilical installation analysis, the DUCO R&D team considered the key variables: tube wall thickness, VLS crush load, internal tube pressure, and caterpillar pad geometry. To gain confidence in the simulation results, they ran four simulations that matched the conditions of four full-scale physical tests for a combination of internal tube pressure and caterpillar pad angle.

Validated simulation process provides confidence

Even streamlined, an umbilical installation simulation can be compute-intensive. A recent DUCO analysis had approximately half a million nodes and a similar number of elements. To handle this complexity the team used a cluster of CPUs with significant capacity. "The goal was to deliver an analysis in a reasonable time," adds Probyn, "and we've succeeded."

Comparing results, the team found good agreement between the FEA predictions and the physical tests (see Figure 2). For most loads, the differences between the FEA and test results were well within the measurement tolerances. The FEA predictions also showed the same trend as the test data in predicting reduced residual ovality as the geometry of the caterpillar pad V-angle was altered from a high to low angle. In addition, the model and tests were in agreement when an internal pressure was present in the steel tubes during application of the crush load. All results indicated that residual ovality was below the recommended 3 percent limit for all loads—well within the nominal crush loads. This gave the team confidence that, in specific cases, crush loads beyond the typical values could be applied.

Overall, the analyses demonstrated to the DUCO team that FEA could accurately simulate complex loading conditions involving multiple component contact and nonlinear material behavior. "Now that we've fully validated the FEA, simulation can be employed as a virtual prototype to perform additional analyses, such as optimization and reliability studies," says R&D team leader Fogg.

With a validated realistic simulation process, DUCO's customers can have confidence that their oil and gas lifelines will be healthy long into the future.

For More Information
www.technip.com
www.simulia.com/cust_ref

Accelerating Innovation in Electronic Product Development

David Cadge, Electronics Industry Lead, SIMULIA Technical Marketing



Smaller devices with more memory and features, environmental constraints, global sourcing, increased speed and decreased cost—these demands pose significant challenges for the electronics manufacturers who, arguably, have the shortest product lifecycle of any industry. Delivering the latest, greatest, smallest and next "must have" tech toy requires design and engineering solutions that will help the industry evaluate and improve product performance on the fly.

When I last wrote an Electronics Strategy Update for *INSIGHTS* magazine in October 2007, I discussed the industry need for unified Finite Element Analysis including: tools to capture and share simulation workflows, multiphysics—including multi-field simulation, advanced capabilities for material modeling, as well as technology for fracture and failure. Today, the industry challenges have only intensified. The good news is that SIMULIA's strategic R&D plans are on target and are helping our customers meet their product development demands. Consider some of the new and enhanced features added into the Abaqus products over the last two years such as: XFEM, low cycle fatigue, implicit dynamics, subcycling, and co-simulation. (See page 14 of this issue to learn more about the latest release of Abaqus 6.10)

In addition to expanding the capabilities in Abaqus, our R&D organization is now responsible for the development of an expanded portfolio of simulation solutions including Isight, Simulation Lifecycle Management, DesignSight, CATIA Analysis, and SolidWorks Simulation. Our electronics strategy now encompasses all of these solutions, and is bringing significant business value to the industry.

Our customers' motivation for using realistic simulation often focuses on reducing or replacing time-consuming and expensive physical tests with virtual tests. For example, an industry-standard moisture sensitivity test for a semiconductor might take several hours to complete—that is after waiting up to one month for a prototype part to be made and another week to pre-condition the specimens. A virtual test with Abaqus can replicate this physical test and can be completed within a matter of hours. This approach provides huge time and cost savings, while allowing the consideration of many more design alternatives.

Plus, realistic simulation can often reveal more than a physical test. Consider a cell phone drop test—simulation can provide views inside the device during the drop event that would be impossible to achieve from physical tests. Simulation also allows results from any location in the model and at any point in time during the analysis.

Unified FEA & multiphysics

Engineering work groups in the electronics industry need to perform a wide array of simulations. Abaqus FEA enables engineers to use a common simulation model and underlying technology to evaluate many different workflows.

In the case of cell phone manufacturing companies, engineers are doing more than just drop test simulation with Abaqus. They

are also using its range of capabilities for coupled structural-acoustics, thermal loading, bending/twisting, and flexible multi-body dynamics for mechanisms—all leveraging the same, underlying FE model.

Semiconductor companies are using Abaqus to perform virtual tests for thermal and power cycles (see page 20), vibration, moisture, and stress. They are looking at simulations covering the complete lifecycle of the component, from manufacture, to assembly, right through to consumer usage and final failure.

As components become smaller and more complex, designing to avoid fracture, delamination, and failure grows ever more important. SIMULIA is the technology and industry leader for modeling and analyzing fracture and failure. We extended our leadership by delivering the first commercial release of the Extended Finite Element Method (XFEM) in Abaqus 6.9. This method enables users to study crack initiation and propagation along an arbitrary solution-dependent path without needing to remesh. It can also perform evaluations for an arbitrary stationary crack. This capability has been further enhanced to support contour integral output, to run in parallel on multiple cores, and to support the implicit dynamic option for transient analyses like thermal shock.

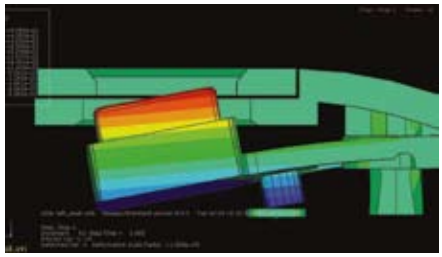
Abaqus 6.9-EF added the option to read multiple nodal output variables—temperature, normalized concentration, and electric potential—from previous Abaqus analyses. This technique enables customers to get the total stress state caused by coupled-fields with a single stress analysis; for example, the coupled response to temperature and moisture for a moisture sensitivity test or to temperature and cure shrinkage for a warpage simulation.

In Abaqus 6.10 we are releasing Abaqus/CFD which enables users to perform conjugate heat transfer simulations. The Abaqus/CFD solver can be easily coupled to an Abaqus/Standard model that has been created for thermal cycling and solder joint creep simulation, and used to perform cooling simulations.

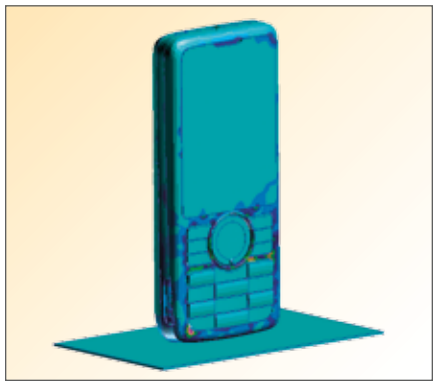
Simulation automation and optimization

Isight, which became part of our product portfolio in 2007, provides engineers with a suite of interactive tools for creating simulation process flows—consisting of a variety of applications, including commercial CAD/CAE software, internally developed programs, and Excel spreadsheets—in order to automate the exploration of design alternatives and identification of optimal performance parameters.

Isight enables users to automate simulation process flows and leverage advanced techniques such as Design of Experiments, Optimization, Approximations, and Design for Six Sigma to thoroughly explore the design space. Advanced, interactive postprocessing tools allow engineers to explore the design space from multiple points of view.



A process simulation for a Notebook power button is performed using Abaqus to analyze the stress caused when pushed. Realistic simulation enables design engineers to evaluate whether the Notebook's power button meets performance requirements. Image courtesy of ASUSTeK Computer, Inc.



Virtual drop tests of a cell phone are performed using Abaqus to analyze the stress and strain of main parts as the phone strikes a surface from various directions. Realistic simulation enables design engineers to evaluate whether the stiffness of the phone's components meets performance requirements. Image courtesy of Lenovo.

Managing simulation IP

Electronic product development companies continue to expand their use of coupled models for multi-field, multiphysics, and multi-scale applications resulting in data being transferred from one model to the next. They are also performing more simulations due to faster computing resources and the need to reduce physical testing. This activity is driving the need for solutions that allow engineers to capture and share simulation workflows while managing applications, computing resources, and simulation results. SIMULIA has responded to this industry demand by developing a product suite for Simulation Lifecycle Management (SLM). SLM accelerates product development by providing timely access to the right information through secure storage, search, and results visualization.

Customer engagements

SIMULIA is proactively engaged in the electronics industry. Our global team and customers present regularly at industry conferences (visit our website to download several of these papers). Our customers also participate in SIMULIA customer review meetings to provide input on their simulation requirements. We are responding to their requests by enhancing our product portfolio with robust technology for multiphysics, design optimization, and simulation lifecycle management. As a result, our customers are solving more complex engineering problems with fewer simplifying assumptions. Our goal is to help our customers create the next "must-have" electronic device faster and more affordably than ever before.



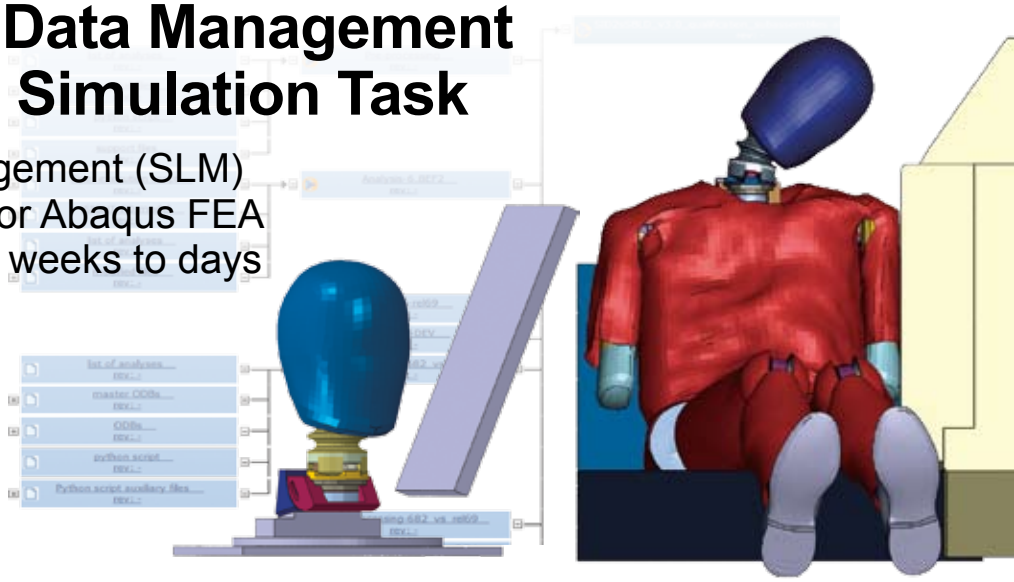
David Cadge
Electronics Lead, SIMULIA

David is responsible for developing and promoting our strategy for simulation within the Electronics industry. He has worked at SIMULIA since 1995 (initially in the UK office and then at the Providence, RI headquarters). David has worked in various capacities within the customer service and marketing teams. He has visited Electronics customers around the world to understand their simulation workflows and requirements. Information gathered during these visits helps SIMULIA provide enhancements for advanced technology, usability, and productivity so that simulation can become an integral part of Electronics design practices.

Download Electronics-related customer papers at: www.simulia.com/cust_ref

Crash Course in Data Management Speeds Up Huge Simulation Task

Simulation Lifecycle Management (SLM) cuts qualification process for Abaqus FEA crash dummy models from weeks to days



Car manufacturers are now legally obligated to certify the effects of crash events on the humans involved. As a result, crash dummies for front-impact ("Hybrid"), side-impact (SID), and rear-impact (RID) have been developed with engineers from around the world contributing over the years to their evolution. A major challenge in the ongoing development of physical crash dummies is the need to reasonably represent how the human body responds in an automotive accident. The ultimate goal of crash dummy research is to aid in creating design improvements for both vehicles and occupant restraint systems to reduce injuries and save lives. Today, physical crash dummies are a valuable part of every automotive OEM's product design, development, and testing arsenal.

Smart investment, big price tag

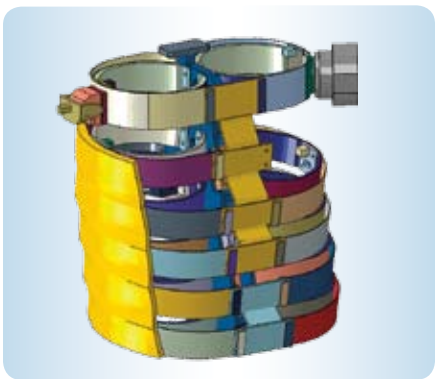
A very valuable part: A single physical crash dummy can cost more than $200,000. Made from a variety of different materials, including custom-molded urethane and vinyl, crash dummies are based on true-to-life human dimensions (a typical "dummy family" includes several different dummies, ranging in size from a toddler to a large adult male). They have ribs, spines, necks, heads, and limbs that respond to impact in realistic ways. They are loaded with sensors (44 data channels on the current front-impact standard, the Hybrid III) that record up to 35,000 items in a typical 100-150 millisecond crash.

As the market for each country's vehicles becomes increasingly global, automotive companies and government organizations continue to collaborate toward the acceptance of international safety standards (a "WorldSID" project is now underway) and harmonize methods of testing. Physical test dummies are only a part of the crash and safety certification process. As computer-aided engineering software and computing resources rapidly advance, there is increasing emphasis being placed on developing ever-more-accurate virtual crash dummies.

Simulating the crash simulator

Given the power of FEA to cost-effectively reduce real-world testing, in the case of expensive crash dummies, and even more expensive vehicle prototypes, it definitely pays to simulate the simulator. You can crash a virtual car and dummy many times, much faster, and at far less cost than a single physical test.

Standardization of FEA models is critical. Each virtual dummy must exhibit responses to crash impact loads and accelerations in a precise, repeatable manner that mirrors what happens to its corresponding physical crash dummy.



Abaqus FEA crash dummy model of a thorax.

What's more, the simulation must continue to run smoothly as each new and improved version of a physical crash dummy comes on the market and as each new version of crash simulation software is released. Simulation software companies go to great lengths to validate the consistency and accuracy of their software in a process called qualification. In the case of creating a new virtual crash dummy or updating an existing one, the software qualification process involves evaluating large quantities of FEA data, gathered from multiple simulations of various crash scenarios, run on different versions of simulation software, and in turn, correlated with new physical test data.

Data, data everywhere

At SIMULIA headquarters, a team of engineers qualify and support a range of virtual crash dummy models developed for their Abaqus FEA software by First Technology Safety Systems (FTSS), a leader in crash dummy innovation for over 40 years. The SIMULIA group also separately develops and qualifies its own virtual crash dummy models, which are versions of the BioRID (Biofidelic Rear Impact Dummy) and WorldSID (Worldwide Side Impact Dummy). "We need to make sure that every new version of each dummy model that's released will work accurately and give the same response no matter which version of Abaqus we, or our customers, are using," says Sridhar Sankar, Manager, Automotive Unified FEA, SIMULIA.

A typical FEA dummy model will have about 100,000 elements, 150,000 nodes and 500,000 degrees of freedom. "To ensure, within engineering tolerances, that you get the same results from the virtual dummies as from the physical tests of the real ones, we have to run component, sub-assembly, and full-model tests on each one," says Sankar. A component test is used to evaluate an individual FEA model of a

dummy neck being bent, a lumbar spine being shoved sideways, or a head being dropped on a hard surface. A sub-assembly test assesses the stresses on a full rib cage model hit from the side by a pendulum, with the ribs being individually deformed and possibly intruding into the body cavity. A full-body test incorporates an entire dummy model being hit from the side by a virtual solid barrier or subjected to a simulated sled test. Different testing standards (NHTSA, IIHS, etc.) require a variety of tests. "With 30 to 60 of these validation tests per dummy model, we end up with a very large number of outputs to generate and then compare," says Sankar.

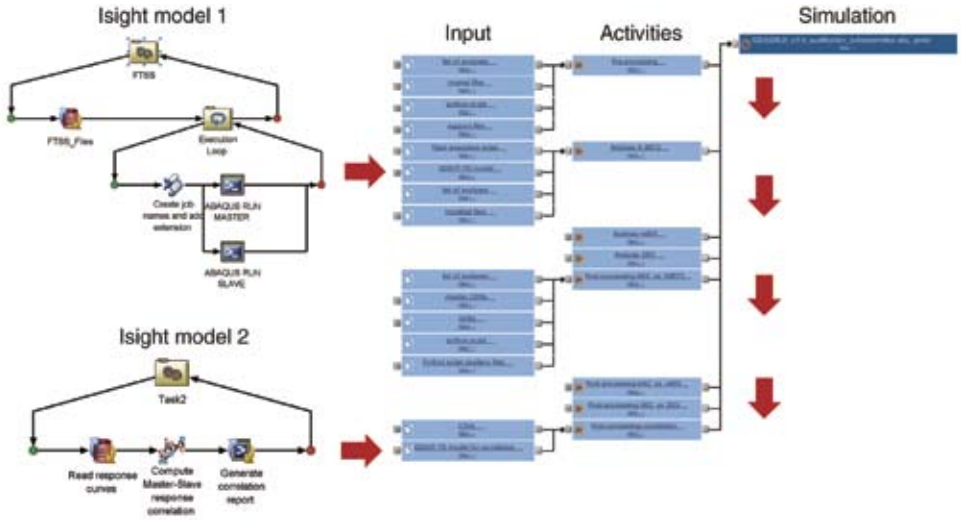
Manual qualification slows down the engineering team

Until recently, dummy qualification took the SIMULIA engineers about four weeks for each updated Abaqus virtual dummy model. "These kinds of challenges meant a lot of man-hours for our team," says SIMULIA crash engineering specialist George Scarlat.

Before even begining the analysis, Scarlat's group had to create its databases by manually modifying each of the previous validation test responses to add proper filtering (which has to meet industry standards, such as J211 or ISO 6487) to the variables so that the results between different versions of Abaqus could be compared.

Next, the engineers had to manually launch and run the simulations for the 30-60 tests in the current and previous versions of Abaqus (usually four or five total). Once they completed the various manual analyses, the team then had to run a post-processing step to generate the curve plots describing the analysis results. The amount of data continues to multiply at this point because the results of a single FEA analysis of dummy rib cage intrusions, for example, could produce up to 200 output variables (forces, displacements, etc.) per test.

Finally, a second post-processing step would take the analysis curves, two at a time, and generate statistical comparisons to quantify the agreement between the same variables in different versions of Abaqus. "So in terms of data you could have 60 tests multiplied by 200 variables multiplied by five different versions of Abaqus," says Scarlat. "This was a lot of manual work. To meet our deadlines, we really needed to improve the efficiency of the entire process."



Screenshot shows how SLM and Isight are used to qualify a crash dummy FEA model for two versions of Abaqus software.

SLM brings the power of PLM to Virtual Crash Dummy Qualification

As a result the group decided to apply a combination of SIMULIA's own Simulation Lifecycle Management (SLM) tool and Isight software for simulation automation and design optimization to automate and manage the tasks. The results were dramatic. "By using our own tools, which we also provide to our customers for automating and managing their simulation processes, we went from four weeks to four days for the qualification process," says Scarlat.

Using SLM as both a database and a process controller, the engineers could save and manage their simulation data, reuse simulations, retain performance metrics, protect intellectual property, and shorten design cycles. They used Isight software within SLM as an add-on tool for driving simulation process automation.

The crash dummy qualification team used SLM as the underlying driver for running each of the three main dummy qualification tasks (preprocessing, analysis and postprocessing) sequentially. SLM automatically exported all the necessary files from its database for each task (activity). It then automatically imported back into its database any specified result files after the activity was run.

Isight automates the qualification process further

SLM also leverages the capabilities of Isight, in this case for process automation. The crash group engineers first used Isight to create a workflow that enabled them to simultaneously launch all of the Abaqus analysis tasks on a compute cluster. A second Isight workflow was employed in the final postprocessing task to help determine the correlation between results from different versions of Abaqus software on identical dummy tests. A Python script was used to modify input files, compare results and generate comparison reports.

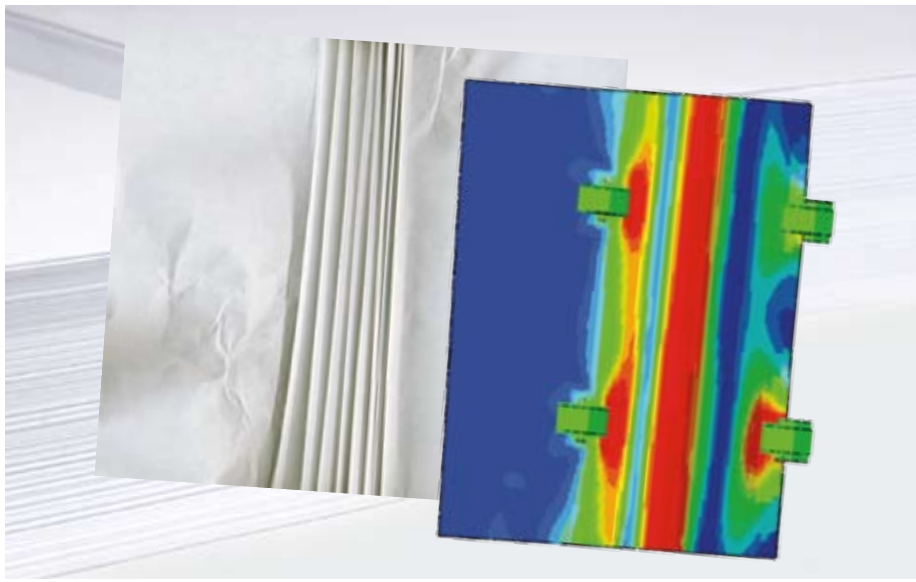
"Automating our tasks was a big help," says Scarlat. "No user intervention was needed during the complicated workflow execution, which resulted in a significant reduction of our process time for the whole project." Scarlat's team qualified five FTSS dummies in the first year of using the new workflow—taking about the same number of man-hours needed to finish only one dummy qualification project before.

The automobile safety engineering world is getting ever closer to the perfect crash dummy. Hybrid IV, also known as THOR, is a dummy currently under development with biomechanical and measurement enhancements that will generate more data than ever. "With such complicated, data-rich FEA in the pipeline, the use of SLM and Isight to automate and manage it all will be even more crucial to the efficiency of our engineering team," predicts Sankar.

For More Information
www.simulia.com/products/slm
www.ftss.com

Designing Your Way Out of a Paper Jam with Realistic Simulation

HCL Technologies uses Abaqus FEA to help keep high-speed printers on track



Remember how the invention of the personal computer was supposed to do away with the need for paper? We all know how that turned out. Despite the proliferation of digital files, email, online publications, and social media, there are more printers in the home and office than ever before. Early laser printers sold for as much as $17,000 in the 1970s—now a low-end black and white printer costs under $75.

The highest growth rate for printers these days lies in the developing world, where increasing prosperity is fosters strong demand for print-generating PCs. India leads the pack. With a national print market for printing equipment, paper, and supplies predicted to expand more than 70% from 2006 to 2011, according to the Print Industries Market Information and Research Organization (PRIMIR), those printers need to run as smoothly as possible: A paper jam is a no-no in any language.

HCL, India's largest manufacturer of PCs, anticipated the boom in printers and other IT-related equipment by spinning off a software services division in the late 1990s, HCL Technologies Ltd. As the inventors of the Indian computer in 1978—concurrent with Apple and three years before IBM—the parent company has long been aware of the importance of starting from good design to ensure product quality and reliability. HCL Technologies now offers engineering and R&D services from initial concept to validated prototype to a wide variety of IT-related equipment makers in India and abroad. A large portion of their work focuses on those ubiquitous printers.

It's all about the paper path

The functional bottom line in printer design is the paper path: the route that a sheet takes through a printer from entry to exit. Of course the inner workings of a printer can vary quite a bit: large or small capacity, faster or slower speed, inkjet or laser technology, monochrome or full-color toner, standard prints or copying, scanning and faxing as well. But if the finished product is imperfect, damaged, or never comes out at all (paper jam!), you end up with an unhappy user.

"Paper path design is a challenge no matter what product we are working on," says Thangavel Mayilvaganan (Mayil), Associate Project Manager, CAE Centre of Excellence, responsible for printer projects at HCL Technologies in Chennai, India. "Meeting the final requirements of each printer manufacturer depends on defining, and then designing-in the proper flow of paper through their particular machine."

Cutting costs with simulation

To make this customization process cost-effective, HCL has been using a variety of CAD and CAE tools for product design and development for more than ten years.

"Real-world verification is always the final proof of functionality, but it is expensive and difficult to develop individual physical paper flow path tests," says Mayil. "Simulation has become the backbone of our R&D process. By using virtual prototyping, starting at the earliest concept stage, we've been able to reduce our product development costs an average of 40 percent."

In a typical computer model used by HCL for a paper flow path analysis—in this case employing Abaqus FEA from SIMULIA, the Dassault Systèmes brand for realistic simulation—the challenge is to accurately represent and analyze the contact and forces that a single sheet of A4 paper encounters on its way from the feeder zone to the outlet tray. The hazards to be avoided are many: the paper can skew out of alignment, flow at the wrong speed, bend and stub (paper jam!), or slip at the roller interface.

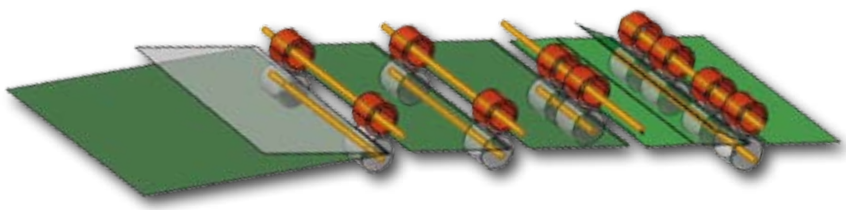


Figure 1. 3D CAD model of a typical printer paper flow path. The paper is pulled into the printer ('nipped') between a rotating upper rubber roller (the drive) and a stationary, spring-loaded lower plastic roller (the driven, or idler). During the printing process the paper is conveyed through a series of these rollers, with baffles and stationary guides (not shown) directing its path.

Simulating the effects of all these variables helps HCL's engineers predict and rectify potential paper flow roadblocks at the earliest stages of design development. Employing Abaqus FEA in a feedback loop with the design department, they can quickly fine-tune and perfect their virtual prototypes before building and validating the final physical prototypes for their customers.

Geometric simplicity, analytical dynamism

To chart and then analyze the path of a virtual piece of paper, the engineers begin with a CAD model (see Figure 1) of the components of a proposed flow path design, including the paper itself.

The CAD model is then meshed using Abaqus/CAE to prepare for an explicit dynamic analysis in Abaqus. This part of the process is fairly straightforward: Neither a piece of paper, nor the components of the paper path, are geometrically complex. The paper is modeled as beam elements with rectangular section properties when 2D analyses (generally side views of paper movement) are being run, and shell elements for 3D (more detailed problems like buckling and skewing). The paper material is considered to be linear elastic isotropic. The baffles, guides and plastic rollers are considered to be rigid. A hyperelastic Neo-Hooken material model is used for the rubber rollers to capture their deformation.

Straightforward, yes, but when the simulation model is set in motion, it becomes a finely choreographed dance in which the correct function of each component is dependent on the proper operation of the previous step. It's a fast-evolving, highly nonlinear FEA problem that has to account for a host of variables: grade of paper, complexity of flow path, roller pre-loads, roller rotation speed, transport velocity and acceleration, materials, friction, even the effect of gravity—at speeds approaching two meters per second (up to 100 pages a minute). The analysis time is estimated as per flow path length and roller RPM.

Fixing the virtual paper jam

Within this multifaceted engineering environment, HCL's engineers can focus in on the behavior of a particular piece of paper as it travels along a proposed design flow path configuration (see Figure 2) . They can simulate what happens when the paper gets out of alignment (skew), and is corrected again by guides. They can study the effects of paper weight and roller drive on bends (buckle) in the paper that lead to potential stubbing points (paper jam!), and they can measure the amount of slippage at the roller interface.

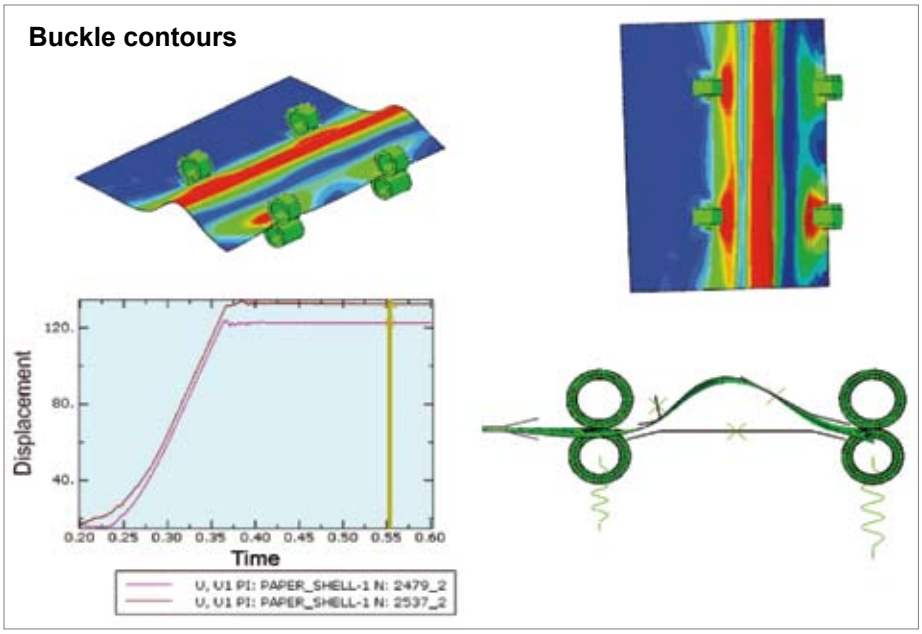


Figure 2. Results from 3D paper buckling contour analysis using Abaqus FEA. When paper flow and roller nip forces are imperfectly balanced, a buckle of paper can rise up, filling the paper path and resulting in paper slip and/or stubbing (paper jam). HCL's engineers use such data to examine the relationship between such forces in different printer designs so they can make modifications that resulting in optimum paper flow.

With their Abaqus simulation results in hand, the engineers can then modify design variables, and combinations of variables, within a flow path. They can vary the distance between rollers, roller positioning, and circumference, the angle of the guides, and so forth—and then run the virtual piece of paper through it all over again. They can also modify the characteristics (weight, thickness, composition) of the paper itself to test the full range of capabilities of each proposed path design.

Abaqus FEA passes the reality test in record time

"Abaqus' advanced contact algorithm capabilities and extensive material models support our simulations for a broad range of customer needs," says Mayil. "To ensure that our final designs are robust enough for a particular printer configuration, we build and test physical prototypes. When we compare our results against the FEA, we see very good correlations." And they accomplish all this with significant time savings: "We've been able to cut three months off overall project time for designing and validating a typical A4 printer using simulation," says Mayil.

Realistic simulation will continue to play a pivotal role for HCL's global CAE team. Future work will focus on the effects of inducing electrostatic charges on paper (a step inherent to the laser-printing process), and consideration of environmental conditions like humidity and temperature.

"With a constant drive for product innovation, cost and weight reduction, the highly competitive high-tech electronics industry is continuously challenged to update products in a very short design cycle," says Mayil. "Reliable functionality is one of the major goals for information technology and electro-mechanical products. Realistic simulation helps us achieve that for our customers."

For More Information
www.hcltech.com
www.simulia.com/cust_ref

Abaqus 6.10 Native CFD Capability for Fluid-Structure Interaction, Plus More than 100 Customer-Driven Enhancements

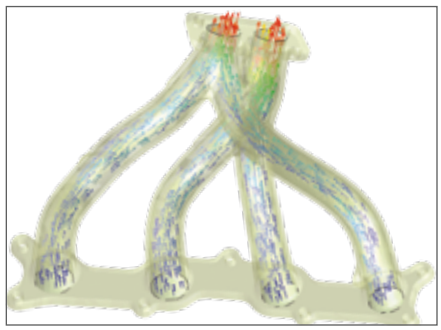
The Abaqus 6.10 release delivers on more than 100 customer-driven enhancements for modeling, performance, usability, visualization, multiphysics, and core mechanics.

Abaqus 6.10 introduces a new multiphysics capability for performing Computational Fluid Dynamics (CFD) simulation. This enhancement enables users to perform coupled physics simulations with Abaqus/Standard and Abaqus/Explicit, such as fluid-structure interaction between a medical device and fluid flow; thermal analysis of electronic systems undergoing convection cooling; or transient thermal analysis of engine exhaust systems.

"In order to simulate performance of engine components in a closer-to-reality environment, we are pleased that Abaqus 6.10 provides the Computational Fluid Dynamic capabilities for fluid-structure interaction which will enable us to perform accurate fluid and solid co-simulations," says Dr. Fred Yang, technical leader of bearing analysis from Federal-Mogul Powertrain Sealing and Bearings Group USA. "The new solution certainly gives us significant enhancements to explore multiphysics interaction in our designs and optimize our products to reduce engine power loss and lower overall material costs."

The release also reinforces SIMULIA's commitment to providing an open multiphysics platform through improvements to the direct co-simulation coupling interface. This capability allows SIMULIA partners and customers to couple their applications directly with Abaqus for best-in-class multiphysics simulation.

With this release, SIMULIA also extends its leadership in the simulation software industry by delivering innovative technology for realistic fracture and failure analysis. Abaqus 6.10 features enhancements to the extended finite element method (XFEM) that improve the process for modeling fracture of composite materials. It also provides dramatic performance improvements for parallel processing of simulations that use XFEM, allowing more simulations to be performed in less time.



Abaqus 6.10 provides native CFD analysis capabilities for fluid-structure interaction and conjugate heat transfer simulations. This image depicts the transient thermal analysis of an engine exhaust system.

"It is important to use the best technologies and methods available to assess the safety of nuclear power plant components," states Dipl.-Ing. Axel Schulz, TÜV Nord. "The XFEM capabilities in Abaqus 6.10 will make it easier to perform virtual safety evaluations of nuclear power plant piping systems and pressure vessels. With this new release, realistic simulation of crack propagation based on both the cohesive segments method and linear elastic fracture mechanics is now feasible, while using the implicit dynamic procedure for improved stability."

Key Features:

Multiphysics

- Interface for CFD modeling, execution, and visualization in Abaqus/CAE
- Coupling with Abaqus/Standard or Abaqus/Explicit for Fluid-Structure Interaction and Conjugate Heat Transfer; Incompressible (transient or steady) Flows; Turbulence modeling
- Co-simulation interface for third parties to integrate their software to Abaqus for coupled multiphysics simulation

Mechanics

- Improved performance and extended feature coverage for general contact in Abaqus/Standard
- Enhanced modeling of fracture of composite materials with XFEM
- Parallel processing improvements for simulations that use XFEM or the implicit dynamic procedure
- A new iterative equation solver offers significant performance enhancements for simulations involving large blocky structures such as oil reservoirs and engine blocks
- Enhanced coupled temperature pore-pressure displacement for modeling heat transfer in porous materials. This is useful for analyzing petroleum reservoirs, nuclear waste repositories, or freeze/thaw cycles in buried pipelines
- A new model for capturing high-rate impact of ceramics and other brittle materials, based on the well-accepted Johnson-Holmquist formulation
- New capability to analyze structures subject to air blast loading

Modeling and Meshing

- Expanded set of geometry edit tools for creating midsurface representations of thin solid parts for more efficient simulations
- General 3D sweep capability for creating complex, curved geometric features, including solid, shell, or cut geometric features such as exhaust manifolds of engines, or window frames in aerospace structures
- Several meshing improvements for quality and robustness of surface and tet meshing
- Improved interface for controlling local mesh gradation and density with enhanced usability and additional controls including double-biased seeding option

Usability and Visualization

- Part- or assembly-based view cuts capability for both meshes and geometry allows interior of models to be visualized which makes it easier to position assembly components and assign attributes
- Enhanced overlay and vector symbol plot capabilities and multiple view cuts for results visualization
- Numerous performance improvements in Abaqus/CAE including faster handling of a large number of connectors, sets and surfaces and faster loading of large databases

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

New Release of Isight and SIMULIA Execution Engine

Parallel Algorithms and Enhanced Optimization Methods, Plus New Abaqus Token Usage Policy

SIMULIA continues its commitment to enhancing Isight, the market-leading simulation process automation and design optimization solution, as well as SIMULIA Execution Engine (formerly Fiper) for distributing Isight simulation process flows across compute resources.

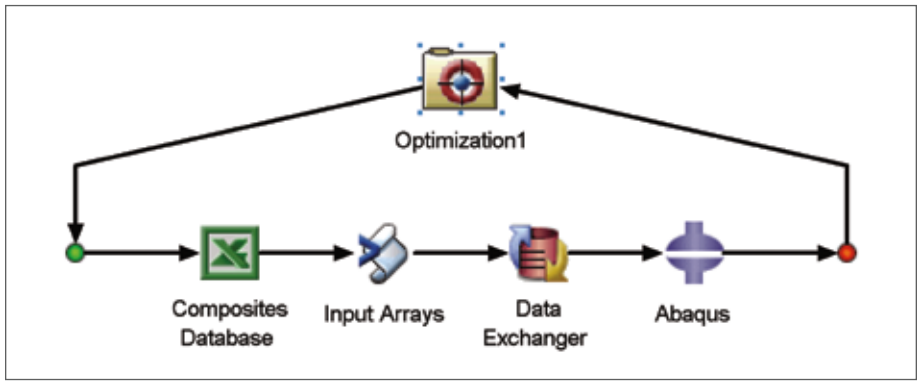
Isight provides designers, engineers, and researchers with an open system for integrating design and simulation models, created with various CAD, CAE and other software applications, to automate the execution of hundreds, or even thousands, of simulations. It allows designers to save time and improve designs by optimizing against performance or cost variables through statistical methods such as Design of Experiments or Design for Six Sigma.

Users of SIMULIA's Abaqus Unified FEA will benefit from a new licensing policy that dramatically reduces the cost for using Abaqus Unified FEA in automated design studies with Isight. Using the combination of these products allows customers to reduce their Abaqus token usage by as much as 60 percent.

In their presentation at the SIMULIA Customer Conference, engineers from Baker Hughes, a top-tier oilfield services company, shared their experience in using Isight with Abaqus to optimize downhole expandable tubulars. "Historically, at least two months of analysis had been required to ascertain an acceptable geometry," stated Jeff Williams, Project Engineer, Baker Hughes Inc. "With Isight, the development period was reduced to two days."

Isight 4.5 provides new scalable parallel algorithms for leveraging multicore computing resources; enhanced approximation and reliability methods to evaluate product performance across a range of real-world operating variables; improvements to multi-objective optimization and data mining which provides deeper insight into performance attribute tradeoffs.

"The new features and enhancements in Isight 4.5 will enable our customers to explore design options that were previously impossible, due to time and cost constraints," stated Steve Crowley, director of product management, SIMULIA, Dassault Systèmes. "By leveraging the new parallel algorithm and optimization features in Isight combined with the distributed computing capabilities of Simulation Execution Engine, our customers will be able to evaluate more design alternatives in less time, resulting in better products at lower cost."



Isight enables users to connect various applications into a simulation process flow to accelerate design optimization.

"Historically, at least two months of analysis had been required to ascertain an acceptable geometry," stated Jeff Williams. "With Isight, the development period was reduced to two days."

—Jeff Williams, Project Engineer, Baker Hughes Inc.

Key Features of Isight 4.5

- Parallel version of Pointer and Multi-Objective Particle swarm
- Kriging approximations
- Consumption of Abaqus tokens is reduced in automated Isight studies

Key Features of SIMULIA Execution Engine 4.5

- Library enhancements: Add, delete, move & copy folders and enable folder access control (ACL)
- Database size monitoring and control
- Planned SEE shutdown, restart and recovery

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

Several SIMULIA customers presented their successful use of Isight at the 2010 SIMULIA Customer Conference.

Advanced Body in White Architecture Optimization
—Pan Asia Technical Automotive Center

Benefits of Simulation Process Automation for Automotive Applications
—INERGY Automotive Systems Research

How Can We Make Best ... Better: Using Abaqus and Isight to Optimize Tools for Downhole Expandable Tubulars
—Baker Hughes Incorporated

Integrating Business and Technical Workflows to Achieve Asset-Level Production Optimization
—Halliburton

Isight-Abaqus Optimization of a Ring-Stiffened Cylinder
—General Dynamics Electric Boat

Simulation Driven Design Enabling Robust Design
—Rolls-Royce

Download papers at
www.simulia.com/cust_ref

Lighten up! Amcor Uses Realistic Simulation to Stay on Top in Plastic Container Market

Abaqus FEA helps industry giant slash design cycle time, reduce unit weight and enhance product performance



Image courtesy of Amcor

The dynamic, competitive landscape of the consumer packaged goods (CPG) industry demands nimble, adaptative strategies. PET (polyethylene terephthalate) plastic container manufacturers are juggling business consolidation, increasing government regulation, and the need to demonstrate corporate and social responsibility. At the same time, ever-changing consumer preferences as well as energy and raw material costs are driving an exponential expansion of product portfolios. The PET customer is demanding that manufacturers develop a wider variety of top-quality, innovative containers in ever-shorter time periods and at lower unit prices.

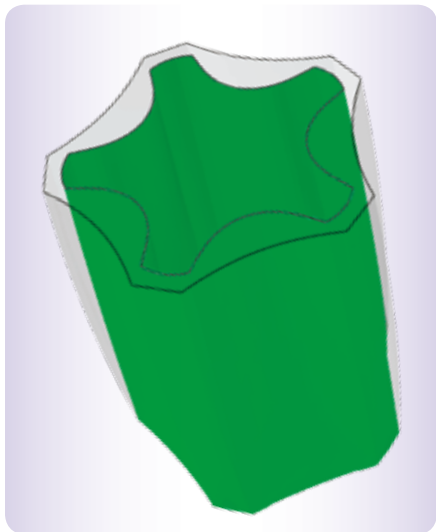
To meet these challenges, the world's largest supplier of PET containers, Amcor's Rigid Plastics Division (renamed from Amcor PET after its parent bought Alcan in late 2009), has found a way to significantly reduce costs—from product design to materials parameters to methods of production—while adhering to strict industry performance standards. They use Product Lifecycle Management (PLM) solutions from Dassault Systèmes to integrate 3D virtual design, finite element analysis (FEA), and collaborative product development software into their product design and development process.

The results: a 50 percent drop in design cycle times, enhanced communication between designers and engineers, less physical prototyping, and faster time-to-market. Plus quicker, more creative response to customer requests for new ideas—and lighter-weight, high-performance product solutions that lower everyone's costs all along the supply chain from raw materials to transportation.

A few grams shaved means millions saved

Amcor's Rigid Plastics Division has 63 facilities in 12 countries that provide packaging for many of the world's leading brands of carbonated soft drinks, juices, teas, water, condiments, salad dressings, sports drinks, soaps, shampoos, pharmaceutical and health care products. The Michigan-based division produces about 25 billion units of bottles, jars, cans and other product configurations per year. Multiply that number by even a few grams saved per unit and the sustainability impact is staggering.

"A container made with too much, or too little, material can be very expensive," says Amcor's Advanced Engineering Services group manager Suresh Krishnan. "Too little material can lead to containers failing, and too much can cost us a fortune. 'Lightweighting' our products is one of the key things that has sustained Amcor against our competition during these tough times, and computer-aided engineering (CAE), within a PLM environment, has been critical to achieving that."



"Origami" concept vacuum panels are included in a PET container for designed collapse that compensates for shrinkage during cooling to maintain structural strength and integrity. Original shape is clear, final shape is green.

The goal of "lightweighting" resonates with engineers in every industry, from aerospace to cell phones. But the weight savings of plastic over glass have dramatically transformed the liquid container business in recent decades. While glass has been used for centuries, and its physical properties are well known, the move to PET in the 1970s required a step-up in sophistication on the part of manufacturers.

Simple product, complicated design challenge

"A PET container is a simple product, but it's a complex design problem to make it right," says Krishnan. For example, the popular two-liter carbonated soft-drink bottle, seen on supermarket shelves everywhere, has to be custom-designed to individual brand specifications and must retain its blow-molded shape during cold-filling, carbonation, sealing, labeling, packing and shipping (hot-filled containers need to withstand additional temperature, vacuum and pressure fluctuations). No container should fail if accidentally dropped, nor excessively dent or lean when stacked.

To cost-effectively produce such a high-performance product, Amcor's Advanced

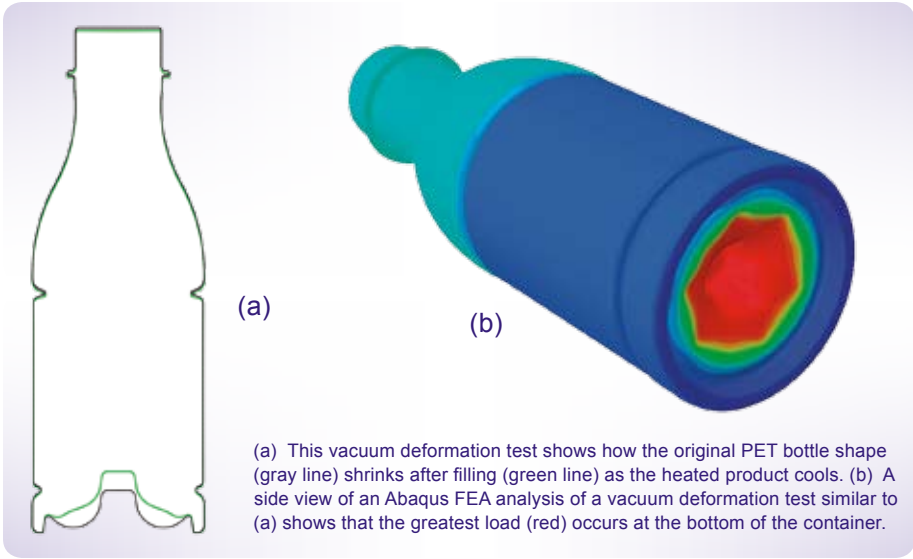
Engineering Services group uses computer modeling to simulate, or virtually test, the behavior of a bottle under these diverse loads and stresses while it's still in the design stage. At the core of their regimen is Abaqus Unified FEA software. Amcor employs Abaqus to generate simulation data that can guide design modifications, material thickness parameters, even manufacturing processes, in order to reach the lightest possible result that satisfies both customer and regulatory requirements.

Visualizing the challenges

Based on an initial concept that the industrial design department has worked out with the customer, the design engineers start by building a 3D virtual model in CATIA. They then use customized scripts and knowledge templates within CATIA to accurately determine the critically important surface area, volume and weight for the bottle's final design. "CATIA's capabilities save us a lot of time," says Krishnan. "Whenever the analysis shows that we need to make a design change, we can do so and the model automatically adjusts to reflect that. And instead of starting a new design from scratch, we can begin with an existing design and quickly modify it."

Next, the engineers mesh the geometry of the virtual bottle with either Hypermesh or Abaqus/CAE ("our designers are increasingly using Abaqus/CAE because it has a CATIA-like look so it's easier for them to work with," says Krishnan), then bring it into Abaqus Unified FEA for physics-based performance simulation. A typical Abaqus model for a top load analysis (such as bottle capping, or container stacking) has about 150,000 shell elements and about 350,000 degrees of freedom. A more complex, Coupled Eulerian-Langrangian drop analysis (which simultaneously shows the fluid-structure interactions between a container, its contents, and the floor) can have up to 800,000 d.o.f. The group runs its analyses on a Microsoft Windows HPC Server.

Amcor tried a different FEA software in the past, but realized they were not getting satisfactory results and switched to Abaqus, a change that empowered the group to begin exploring the full scope of its design challenges. "Abaqus was the better choice for us because it offered a breadth of simulation disciplines that cover more significant performance requirements for PET containers," says Krishnan.



(a) This vacuum deformation test shows how the original PET bottle shape (gray line) shrinks after filling (green line) as the heated product cools. (b) A side view of an Abaqus FEA analysis of a vacuum deformation test similar to (a) shows that the greatest load (red) occurs at the bottom of the container.

Kicking the container around with simulation

It offers quite a range of disciplines. The group began with top loading and vacuum pressure simulations. They moved on to drop-testing, blow molding, conveyance, denting, and leaning. They are currently working on pasteurization and retort (heating during sterilization) simulations. They're even starting in on ergonomics, to simulate the effects of a human hand putting pressure on a container. "Being able to simulate multiple load conditions at the same time is very important to us," says Krishnan. "You have to take into account a number of parameters simultaneously, such as fluid-structure interaction, temperature, pressure, and material strain rate."

With their FEA results in hand, the Advanced Engineering Services group has a clear vocabulary for discussing the viability of a design with the industrial designers. Using multiple iterations between CATIA and Abaqus, the parties can collaborate to arrive at the best solution that validates the appearance, performance and functionality of a particular container. Such improved communication pays off: "One of our performance metric targets was to reduce the number of design revisions we made by 20 percent in a year," says Krishnan. "Right now we are well ahead of that goal."

"The benefits from virtual testing can extend beyond the testing laboratory all the way to manufacturing," Krishnan says. "When we achieve an optimum top load value via simulation, we can use that data to provide actual section weights to the process engineers in the plant, so they can more easily produce the container that gives the desired performance."

PET plastic behavior is complex

The PET material itself brings unique challenges to this whole process. PET is highly nonlinear, with biaxial properties that vary with the amount of stretching it undergoes. A semi-crystalline thermoplastic, PET softens at a "glass transition temperature" of approximately 76 degrees C. Above that, it becomes elastic and can be formed, a property effectively utilized in the stretch blow molding process.

But when PET containers are filled with a hot liquid, they are susceptible to shrinkage back towards their "remembered" previous shape (the preform), a characteristic that has to be taken into account when designing the initial container configuration. The bottles also collapse slightly due to vacuum pressure resulting from cool-down after hot-filling. So the design for a hot-fill PET bottle includes 'vacuum' panels for designed collapse. "We can now easily model these kinds of physics-based characteristics with Abaqus FEA, using a customized script for hydrostatic fluid elements that enables us to accurately simulate the behavior," says Krishnan.

The contents of every type of PET container must also be taken into account in Amcor's simulations, from adjustments in the density and viscosity values of liquids (from pure water to sticky paint) to the internal pressure fluctuations inherent to carbonated soft drinks.

Continued on page 18

Amcor continues working on advanced material properties for their models. While PET is 100 percent recyclable, containers made from recycled PET (RPET) may have slightly different material properties than the originals. Initiatives also are underway in the industry to develop biodegradable PET using ethanol. "Although we are not simulating either of these materials at the time, this is certainly a consideration for the future," says Krishnan.

Managing all that data

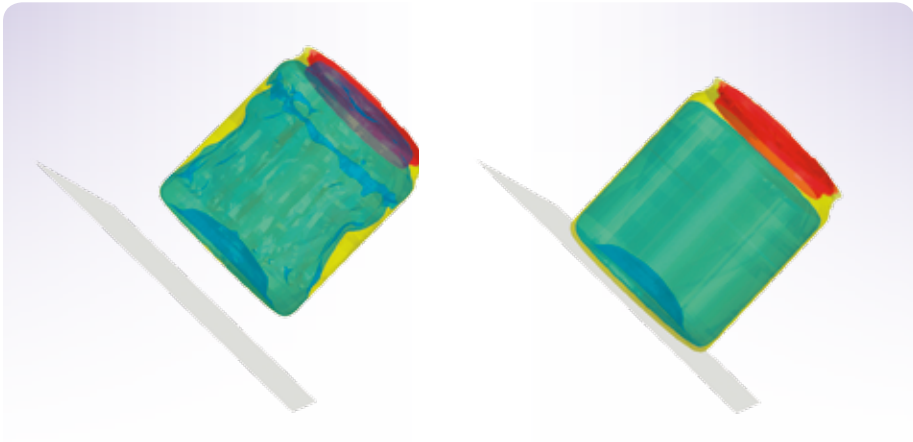
It all adds up to a vast amount of simulation data. Amcor keeps track of everything the Advanced Engineering Services group generates by using Dassault Systèmes' ENOVIA solution for collaborative product development, which facilitates the organization and easy retrieval of all CATIA and Abaqus data for each container design while managing all processes to keep them in synch.

"Whoever in our organization—from the Advanced Engineering Services group of 14 engineers all the way to our manufacturing plants—needs information about a specific project, they can pull up the report in ENOVIA and find the latest version, completely standardized, which is very helpful," says Krishnan. "ENOVIA automatically saves the history of every previous iteration as well, allowing for easy reference, tracking and communication among our project teams."

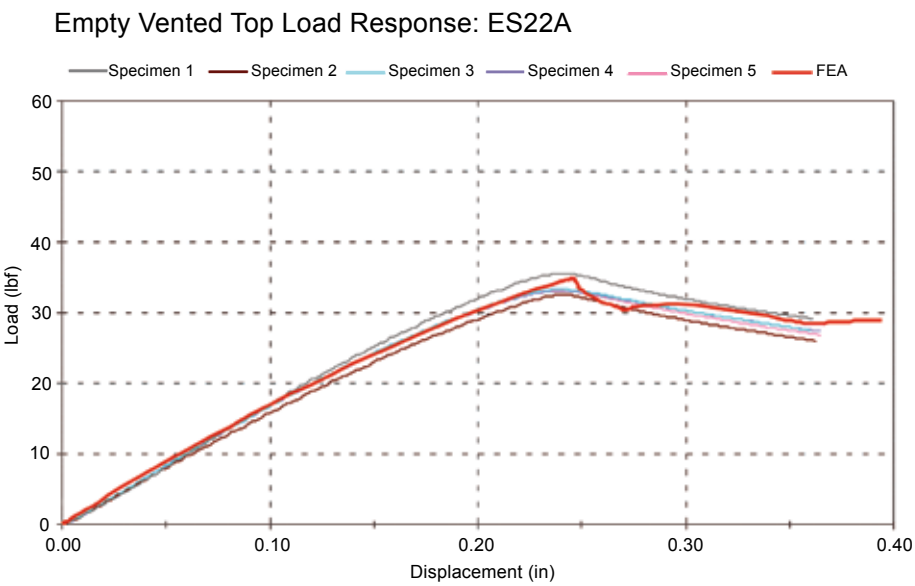
Results rise to the top with simulation-driven lightweighting

The growth of Amcor's physics-based simulation capabilities has been the driving force behind the company's lightweighting initiative. Krishnan cites one example where a 63-gram container design was reduced to 43. "We used realistic simulation to validate performance while trying out various Amcor-developed technologies and eventually met all performance requirements with the lighter design," he states. "Simulation helped us try many more options than we normally would and compare multiple designs with one another."

Although Amcor still validates their virtual tests with physical testing, the ever-increasing accuracy and refinement of their computer predictions has allowed them to decrease physical prototyping dramatically. "We see a close match between the curves that Abaqus provides and the test results so we've got a lot of confidence in simulation now," says Krishnan. "We've cut our design cycle down to nine months from 12 to 18, which has significantly reduced our product development costs. And we've gained a lot of management buy-in to our methodology."



Abaqus FEA container drop test uses a Coupled Eulerian-Lagrangian analysis to show the interaction between the container, the fluid it holds, and the surface it impacts. The top must stay on even when the container is dropped 3 ½ feet to a hard floor.



Graph of empty vented top load response test results shows how accurately Abaqus FEA (red line) predicted the behavior of the container.

CAE promotes creativity

The use of CAE has proved of value for Amcor when proposing new ideas to clients, Krishnan says. "We include animations of our Abaqus simulations in all our presentations. We can demonstrate how we create a design, perform FEA on it, and try out as many options as we want." Any industrial design proposal can be quickly simulated; if a customer puts in a request in the morning, animations can ready by that evening. "It really frees the designers to explore whatever ideas they have," says Krishnan.

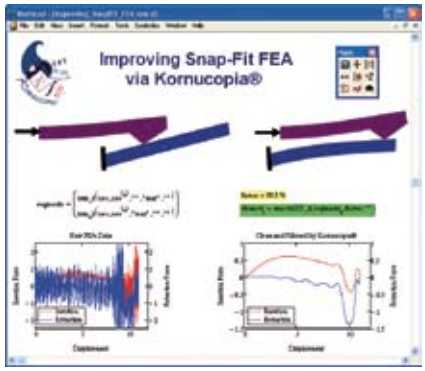
"It's a fast-changing business and the next new design is just around the corner," he adds. "Somebody else is always looking to capture that design so we have to be really fast—and with CAE in our arsenal, we are."

For More Information
www.amcor.com

Streamline Model Correlation, Snap-Fit Simulations, and Repetitive Shock Analyses with Kornucopia® and Abaqus

Abaqus analysts work with nonlinear simulation and experimental data that often has non-ideal aspects such as noise, discontinuities, and drifting. At Atrium Medical, David Heim's CAE team frequently uses Bodie Technology's Kornucopia® software to provide a consistent means of interpreting measured data from testing of implantable medical devices. Using a combination of trimming, shifting, smoothing and derivative functions from Kornucopia, they efficiently "clean" measured data providing a more accurate and reliable basis for correlation which ultimately improves the fidelity of their Abaqus models. Additionally, their reusable worksheets provide clear and traceable documentation of their data cleaning processes.

At BD, engineers developing medical devices often work with polymeric snap-fit features that endure highly nonlinear post-yield behavior. To obtain efficient and robust simulations, BD uses Abaqus/Explicit in conjunction with Mathcad and Kornucopia for post-processing and correlation to large volumes of physical test data. According to Dr. Arun Nair, CAE Project Engineer, BD,



Reusable, self-documenting Kornucopia® worksheet.

"We utilize several easy-to-use Kornucopia DSP and data processing functions to separate out assembly and disassembly responses, analyze frequency content, and smooth data. All the methods, data and plots, including subsequent evaluations of dimensional and material changes, are stored in one Kornucopia worksheet which provides clear traceability to each of the individual procedures. The advanced functionalities in Kornucopia have enabled us to confidently and quickly post-process FEA and physical test data to make accurate and important engineering decisions."

Designing construction equipment capable of withstanding severe repetitive shock is another challenging problem being tackled by combining Abaqus with Kornucopia. In this customer case, applying Kornucopia's decimation feature to the experimental acceleration data driving the model cut simulation time by 50 percent, yet maintained essential frequency content and random characteristics. Initial day-long Abaqus implicit transient dynamic simulations were further reduced to a couple hours by switching to modal transient analyses while still achieving 80 percent of the desired response metric for accuracy.

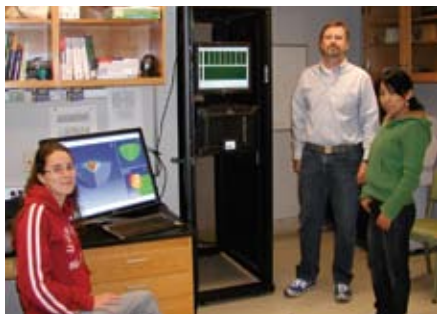
Other keys to success were additional Kornucopia-based modifications to the driving signal—using ramped windows, highpass filters, and integration/derivative functions to minimize the artificial discontinuity of an "abbreviated" experimental signal which would otherwise cause improper ringing and drifting.

For More Information
www.BodieTech.com

Exploring the Interior of an Active Volcano with Abaqus and Cray

The recent eruption of the Eyjafjallajökull volcano in Iceland caused the worst air travel bottleneck in history. For Dr. Tim Masterlark, Assistant Professor at The University of Alabama, and his fellow researchers—who study the physical behavior of earthquakes and volcanoes using Abaqus FEA, satellite imagery, and seismometer data—it was a signal that their work is critical and more urgently needed than ever before.

Due to the recent seismic activity and history of volcano eruptions in Iceland, Masterlark and his colleagues are in a race against time. To help accelerate their Abaqus simulations, they acquired a Cray CX1. "Estimating the parameters that best describe the magmatic behavior is numerically intensive," states Masterlark. "This process involves automated algorithms that incrementally adjust and improve the Abaqus model configurations until the simulations accurately predict observations. This can require thousands of simulations to obtain an optimal set of parameters. Our new Cray



Erika Ronchin, Dr. Tim Masterlark, and Dr. Wei Tao.

CX1 provides the computational firepower to achieve this goal in an acceptable amount of time."

The team's Cray CX1 configuration was setup specifically for running Abaqus simulations. It includes 40 cores and 120 GB RAM in a standalone configuration that is fully dedicated to simulating volcano deformation. The scalability of the Cray CX1 system accommodates future expansion of Dr. Masterlark's research group.

The University of Alabama team is extending the concepts learned in previous studies (an exercise in hindsight) to attempt to forecast the timing of a future eruption of Hekla, one of Iceland's most active volcanoes. Researchers believe that Hekla's next eruption is imminent and presents a narrow window of opportunity to forecast the specific timing of the upcoming eruption.

"We are planning an expedition to Hekla to deploy seismometers and collect seismic data to construct tomographic images, which will help us design and constrain Abaqus-based simulations of magmatic migration," states Masterlark. "These simulations will ultimately guide our forecasts for the upcoming eruption. If we are successful, Abaqus will play a key role in eruption hazard assessments for active volcanic systems."

For More Information
www.geo.ua.edu/faculty/Masterlark.php

Simulation of Back Grinding Process for Silicon Wafers

Semiconductor devices are key components for a wide range of electronic applications. Silicon wafers are commonly used as substrates to build the vast majority of semiconductor devices. Part of the reason for their success has been the ability to reduce costs year upon year while meeting stringent size and weight specifications for electronic packages.

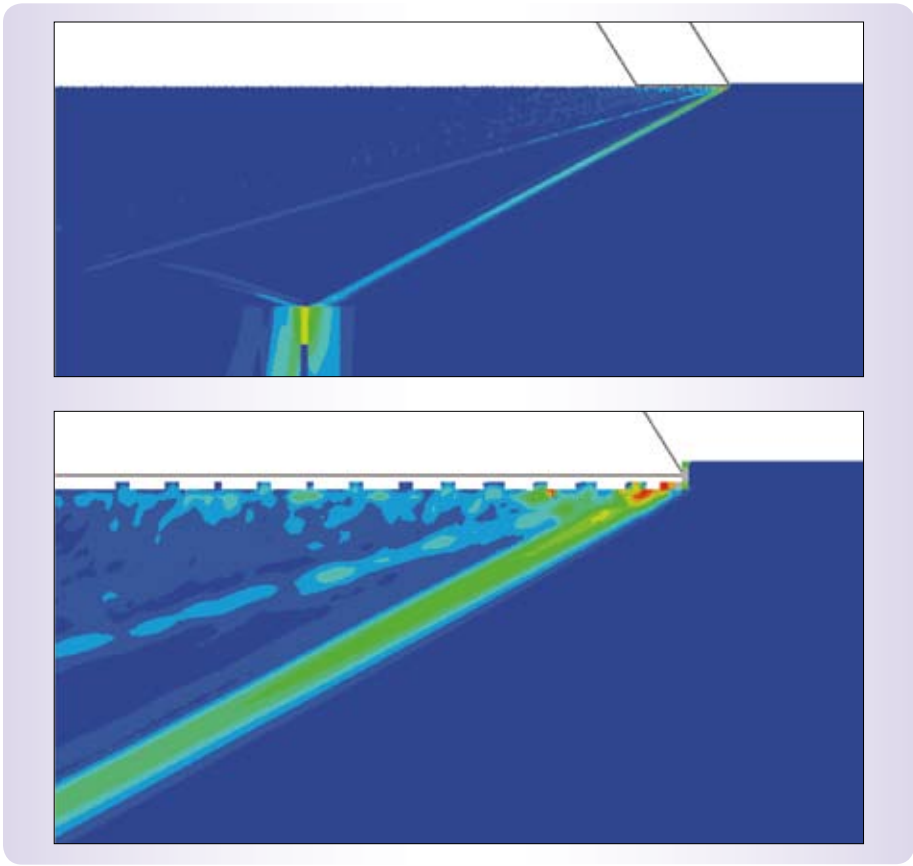
As consumers continue to demand smaller, lighter, and higher capacity devices at low price points, meeting shrinking weight and size requirements poses significant challenges in the development of modern electronic devices. Silicon wafer thickness greatly affects package size, thus thinner wafers result in smaller packaging dimensions. To manufacture the thinnest wafers possible requires a process called back grinding of the wafer, which also poses engineering challenges.

In the Mechanical and Electrical Engineering Departments at the University of Idaho, Professors Potirniche and Barlow with graduate student Abdelnaby worked in collaboration with the researchers from Micron Technologies to simulate the back grinding operation of silicon wafers in order to predict residual stresses and achieve a thorough understanding of the plastic deformations and damage processes during a grinding operation. The numerical simulations involve varying grinding parameters to determine optimum conditions that will minimize the residual stresses and surface damage.



Researchers Abdelnaby (left) and Potirniche in their computer lab.

Traditionally, researchers have used macro-scale or the micro-scale approach to simulating the grinding process. Macro-scale models consider the overall wheel–workpiece interaction which captures the aggregate effects of the abrasive wheel on the workpiece and makes no attempt to study the deformation and damage at the crystallographic grain level of the wafer. On the other hand, micro-scale models focus on the individual grain-workpiece interactions. These models attempt to elucidate mechanisms involved in the material removal at the micron length scale. They simulate the micro-scale grinding process, which includes the high fidelity modeling of a single diamond crystal (abrasive grain) cutting through successive silicon layers. Micro-scale models have the potential to estimate the grinding forces directly, without resorting to measurements or empirical measurements.



(Top) A 500 micron-long silicon wafer being cut by a diamond grain.The figure illustrates the von Mises stress distribution during grinding. (Bottom) Stress distribution near the tool tip and damage localization near the surface of a silicon wafer during back grinding.

The University of Idaho and Micron Technologies corporation researchers have built a two-dimensional model to simulate the cutting process of a silicon wafer by a small diamond particle. Using parallel processing capabilities of Abaqus/Explicit and a set of properly defined boundary conditions, accurate simulations of the grinding process at the micro-scale were achieved.

The residual stress field, as obtained from the numerical simulations, was compared with experimental data from Raman spectroscopy measurements and excellent agreement was obtained. This example shows the robustness and the availability of a wide range of material models provided by Abaqus/Explicit. These extensive capabilities allowed accurate simulation of grinding in order to better understand and improve this challenging manufacturing process.

A.H. Abdelnaby[1], G.P. Potirniche[1], F. Barlow[1], A. Elshabini[1], R. Parker[2], T. Jiang[2]

[1]University of Idaho, [2]Micron Technologies

For More Information
www.uidaho.edu/engr

San Jose State University Simulates the Thermal Characterization of Fan-in Package-on-Packages

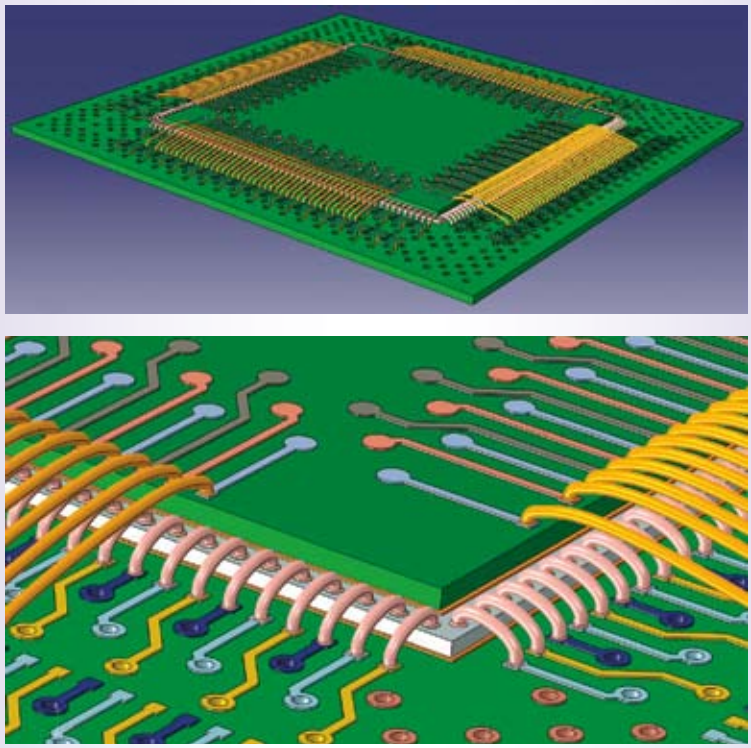
The need to integrate more device technology in a given board space for handheld applications such as mobile phones and medical devices has driven the adoption of innovative packages which stack such devices in the vertical or third dimension (3D). Further reduction of size, thickness, and cost of this Package-on-Package (PoP) solution was possible through the development of Fan-in Package-on-Package (FiPoP) technology which enabled more device integration while maintaining reliability requirements of typical handsets.

One common mode of failure of FiPoPs occurs due to thermal conditions in the stack. Providing a thermal path for heat dissipation is the only option to maintain the junction temperatures in these packages due to space and cost constraints. Though various factors affect package thermal performance, graduate student Nandini Nagendrappa with guidance from Professor Nicole Okamoto and Professor Fred Barez of San Jose State University, chose to focus on varying the internal design parameters only to observe the change in thermal performance.

Based on the research of earlier generation models, the parameters chosen for analysis in this work included (a) number of thermal vias (b) solder ball Input/Output (I/O) density and (c) die size. Geometrical and materials parameters for a typical FiPoP were acquired from STATS ChipPac Ltd. The stacked package within FiPoP chosen for analysis included two metal layers, 14 x 14 mm body size, 9 mm x 9 mm die size, 0.075 mm thickness for both top and bottom packages, and 0.5 mm solder ball pitch.

For this study a test package was modeled in Abaqus/CAE, as it provides the ability to create intricate parts at the micron level, which was required to model the package involving vias, traces, and wirebonds.

JEDEC-specified standards and environment were applied to carry out the steady-state finite element thermal analysis. Thermal boundary conditions applied were power dissipation, ambient temperature, and a combined heat transfer coefficient for natural convection and radiation (typical of still air conditions). Also, changes in thermal resistance were examined from one test run to another rather than absolute values.



(Top) FiPoP model with stacked top and bottom package. (Bottom) Bottom package with interposer close-up copper traces and wire bonds.

The simulation on the stack was carried out by either powering (loading) the top or bottom package one at a time or by powering both. This was done to study the effect of thermal loads separately and combined.

In development of mobile handsets and medical devices, designers must pay particular attention to the design of efficient heat paths to the die package. Heat flows from the silicon die to the ambient through two main mechanisms. One is through conduction from the silicon die of both top and bottom packages through the die attach, substrate, and solder balls to the PCB. The other is through conduction from the die through the mold compound to the top and sides of the package. From the package the heat is transferred to the surroundings through convection and radiation. A surface-to-surface contact approach was adopted in Abaqus to define the electrical/thermal I/O path. The contact detection toolset in Abaqus/CAE automatically generated all the required surfaces and interactions making it very easy to define the extensive thermal contacts in this model.

The importance of the analysis results lies in the change in resistance from one simulation to the next rather than the absolute value, since the total resistance includes the convection resistance which is a constant for all cases. For each case, absolute values of results were obtained and percentage changes between simulations were tabulated.

The analysis predicted that the thermal resistance of the bottom package of a FiPoP decreases with the increase in the number of thermal vias and solder balls placed under the package. As expected, the thermal resistance of the entire package increases as the die size drops.

The article is based on the paper presented at the 26th IEEE Semiconductor Thermal Measurement & Management Symposium - 2010, entitled "Thermal Characterization of Fan-in Package-on-Packages," by Nandini Nagendrappa, Nicole Okamoto, and Fred Barez from San Jose State University, San Jose, California, USA.

For More Information
www.engr.sjsu.edu

InnerPulse Accelerates Medical Device Innovation with SIMULIA Solutions

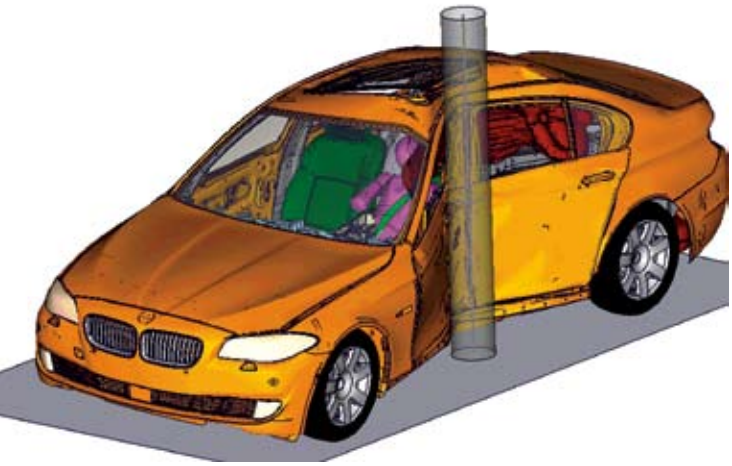


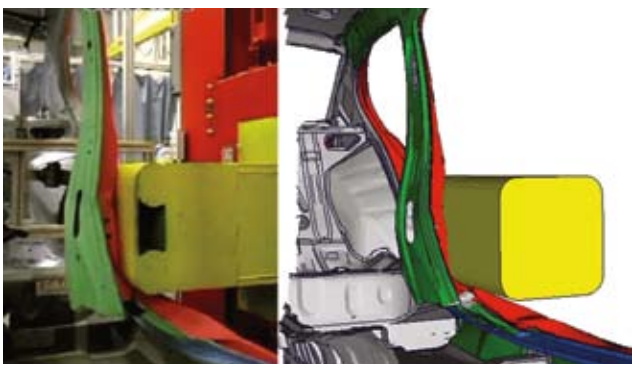
Founded in 2003, InnerPulse is a medical device company pioneering a novel technology for those patients with cardiac rhythm disorders. Using Abaqus FEA to assist in development of their technology designed in SolidWorks CAD software, InnerPulse has developed a new, truly minimally invasive treatment for patients with cardiac rhythm disorders.

According to the Sudden Cardiac Arrest Association, approximately one American life is lost every two minutes due to cardiac arrest, with an estimated more than 7,000,000 lives lost per year worldwide. An overwhelming majority of these deaths are caused by ventricular fibrillation, or rapid, uncoordinated contractions. InnerPulse's new device, a percutaneous implantable defibrillator (PICD), allows simple implantation of the defibrillator within a patient's vasculature using a catheter procedure. InnerPulse leveraged SolidWorks design and simulation capabilities in Abaqus FEA software, providing engineers accurate analysis for simultaneous device and tool design—ultimately lowering costs and saving development time.

With new technology allowing the development of industry-changing devices, InnerPulse and SIMULIA help lead the way for new procedures in the advancement of saving lives.

>> www.inner-pulse.com





BMW Group, "Predictive Crashworthiness Simulation in a Virtual Design Process without Hardware Testing", 2010 SIMULIA Customer Conference

Leading German Automaker Selects SIMULIA Solutions for Passive Safety

As an extension to the recent five-year partnership with Dassault Systèmes, BMW Group has renewed its commitment to use Abaqus Unified Finite Element Analysis (FEA) software for the engineering of passive safety in the automaker's virtual design process.

BMW first began employing Abaqus as its exclusive tool for crash simulation in 2004, when vehicle development projects were largely supported by hardware testing and the focus of simulation was on global vehicle behavior. More recently, BMW has begun a strategic shift toward a more complete virtual development process.

Following extensive evaluations conducted by BMW, ranging from component-level to full-vehicle simulations—and involving key applications in car body technology as well as occupant restraint systems—results showed Abaqus FEA consistently delivered higher levels of predictiveness and repeatability against physical tests than other simulation software. This robustness and reliability is critical as BMW moves toward a more efficient and cost-effective virtual vehicle development process that depends less and less on physical prototyping.

The strong correlation between physical test and simulation results obtained with Abaqus enables BMW to achieve its aggressive process improvement goals, resulting in substantial cost and time savings for each vehicle project, while meeting stringent safety requirements.

>> www.simulia.com/cust_ref



2010 RUM Schedule

Attend the upcoming Regional Users' Meeting in your area. Learn about the latest enhancements to our products and the ongoing strategy of SIMULIA. For additional information, visit www.simulia.com/events/rums.

Americas

Date	Location
September 28	Chicago, IL
October 19	Houston, TX
October 20–21	São Paulo, Brazil
October 25	Southern CA
October 26	Canada
October 27	Northern CA
October 28	Beachwood, OH
October 29	Pacific Northwest
November 10	Plymouth, MI

Europe/Middle East/South Africa

Date	Location
September 20–21	Germany
September 23	Athens, Greece
September 23–24	Oslo, Norway
October 12–13	Prague, Czech Republic
October 28–29	Torino, Italy
November 4–5	Istanbul, Turkey
November 9	Vienna, Austria
November 10–11	United Kingdom
November 11	Belgium
November 15	Madrid, Spain
November 18	Vélizy, France

Asia Pacific

Date	Location
September 7–8	Tsingdao, China
September 10	Korea
September 16	India
October 28–29	Kuala Lumpur, Malaysia
November 2–3	Taipei City, Taiwan
November 17	Tokyo, Japan



Germany



Korea



Scandinavia



Americas

SIMULIA Helps Keep My World Green



Simulation for the Real World

Electronics manufacturers are eliminating lead-based materials in chips and circuit boards while creating portable products that stand up to everyday use. Our customers use SIMULIA solutions to understand the behavior of leadfree solder connections to optimize designs and prevent fracture. We partner with our customers to deploy innovative simulation methods and technology which helps them drive innovation and keep our world a little greener.

SIMULIA is the Dassault Systèmes Brand for Realistic Simulation. We provide the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and an open PLM platform for managing simulation data, processes, and intellectual property.

Learn more at: **www.simulia.com**

SIMULIA

www.3ds.com

The 3DS logo, SIMULIA, CATIA, 3DVIA, DELMIA, ENOVIA, SolidWorks, Abaqus, Isight, Fiper, 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, 2010