I want to personally welcome you to Berlin and the 2015 SIMULIA Community Conference. Over the next three days, you will hear from some amazing customers who have used simulation technology to take their models into exciting new areas. We invite you to open your mind and be ready to learn, share and network so you leave motivated and inspired.

This year’s agenda delivers more technical content than ever, including an opening address from Bernard Charlès, President and CEO of Dassault Systèmes, four keynote and special guest speakers, more than 90 customer presentations, three Technical Forums and seven Technology & Product Updates. Our plenary sessions bring a perfect combination of retrospection and innovation. As head of Airbus Airframe and Technical Authority, Dominique Moreau will share an overview of the design of the Airbus A350, while Bruce A. Dale, Chief Subsurface Engineer at ExxonMobil, will discuss energy in the 21st century. Dr. Martin Hilchenbach of the Max Planck Institute for Solar System Research will tell us how he used SIMPACK technology to land a space probe on a comet more than 500 million miles away. And our own Frans Peeters will talk about how simulation has evolved over the past four decades.

During the General Lectures, you will have the chance to hear directly from members of our Research & Development department and other SIMULIA experts about simulation innovation in the age of experience, the benefits of the 3DEXPERIENCE® platform and updates to our established portfolio. At the core of our conference you will always find our customer papers. These presentations provide an opportunity to hear about the recent simulation accomplishments of your peers. We have put together a special series of oil and gas presentations to be featured during our Geomechanics Super Track. Also, building on the success of last year’s Optimization Forum, we have expanded the program to include two more forums that focus on Multi-Body Simulation and Fatigue & Durability.

As always, one of the keys to our success is our strategic alliances. This year, we want to give a special thank you to CD-adapco, a longtime partner and sponsor of our event. We also want to welcome new sponsors, like CST to the SIMULA alliance lineup. Learn more about these complementary solutions through the partner demo stations and integrated solution sessions and take some time to meet with them during breaks in the Exhibit Hall. I also encourage you to visit the SIMULIA booth, as well as our sister brands.

As with all of our events, this year’s SCC is designed to benefit those who make SIMULIA a success — you, our users. Our goal is to make you better at what you do while providing the most cutting-edge technology to help you do it. Learn from our experts. Hear from your peers. Reconnect with old friends and meet new ones. Make a decision to leave as an innovator!

SCOTT BERKEY
CEO SIMULIA and
NAM Managing Director
Thank You to Our 2015 Alliance Program Partners
Who Co-sponsored This Conference:

PREMIER SPONSORS

CD-adapco is the world’s largest independent CFD focused provider of engineering simulation software, support and services.
Booth P1

CST
Founded in 1992, CST offers the market’s widest range of 3D electromagnetic field simulation tools. CST develops high-performance software for the simulation of electromagnetic fields in all frequency bands. Today, the company enjoys a leading position in the 3D EM simulation market and employs 260 personnel around the world.
Booth P2

EXHIBITORS AND ADDITIONAL CONFERENCE SPONSORS
**Numerical Simulation on A350: A Success Story**

The A350 was designed with a major step forward regarding use of numerical simulation for structure analysis. Numerical simulations have been extensively used on the program to speed up trade-off studies, reduce cost and lead time, address technical issues and mitigate technical risks.

There is great potential for numerical simulation to foster cost and lead time reduction if further steps are taken. This presentation will present a technical maturity status regarding numerical simulation in Airbus, highlighting the topics where a continuous effort will be required to improve confidence, reliability and cost-effectiveness of numerical simulation.

The presentation will provide Airbus’ view regarding the future of simulation, especially on three different axes:

- Lead time and cost improvements
- End-to-end process with improved CAD/CAE interfaces
- New challenges generated by 3D printing technology

**Energy in the 21st Century – Building on the Fundamentals**

Energy is the engine for economic growth and a catalyst for human achievement. Moreover, the global energy challenge for producing and delivering energy is critical to impacting the lives of people all around the world.

In the last decade, natural gas and associated hydrocarbon liquids from unconventional resources such as shale emerged as attractive energy sources due to their efficiency as clean-burning fuels and great natural abundance. The advent of horizontal well drilling and hydraulic fracturing reshaped the energy landscape and significantly improved the energy self-sufficiency of North America, and offers similar potential elsewhere across the globe.

This presentation highlights energy in the 21st century and describes a collaborative partnership undertaken by ExxonMobil and SIMULIA to develop new, fundamental improvements in simulation to address key drilling, completion and production challenges in the upstream oil & gas industry.

**Rosetta Philae - Impact on a Comet as Envisaged by Multi-Body Simulations**

On November 12, 2014, after a 10 year journey, the ESA Rosetta spacecraft ejected the cometary lander Philae and, after 7 hours of free fall, Philae touched down for the first time on the surface of comet 67P/Churyumov–Gerasimenko. Much earlier, in 1996, we started to get a grip on the potential landing scenarios of Philae with the multi-body simulation program package SIMPACK. The objective of the simulations was the analysis and testing of Philae in a micro-gravity environment or the 3-D visualisation of Philae’s inelastic impact on and potential rebound off the comet surface. The most intriguing part from the dynamical analysis point of view are the 3 subsystems essential for landing and operation on the comet surface: The damping mechanism, the anchoring harpoons and the cold gas hold-down thruster.

**Advanced Finite Element Analysis - From Expert Analysis Tool to Realistic Simulation Platform**

Having worked for the past forty years as a researcher and as a software vendor providing nonlinear FEA solutions, the presenter has been at the forefront of the development of FEA technology and of its industrial deployment. The overview presentation will review the historical evolution during the past four decades, from limited expert analysis tool to widely used software solution for realistic simulations. It will address obstacles that formed a severe hindrance for a broad usage of nonlinear FEA tools. In addition major breakthroughs will be presented that have contributed to the ever increasing popularity, importance and use of the tools for Realistic Simulation. Based on the presenter’s experience and insight in the evolution and state-of-the-art of the simulation technology he will give a personal view and outlook on future trends in the adoption, usage and application of this technology.
GENERAL LECTURES, LUNCH GROUPS AND SUPPORT DESK

General Lectures

General Lecture 1: Powering Innovation in the Age of Experience
Over many decades, simulation has now proven itself as a practice that can improve product design, guide design along the right path, reduce overall cost, and shorten the design cycle. It has now also proven itself as a practice that can power innovation. Numerous examples at prior SIMULIA Community Conferences have shown that simulation reveals new design possibilities, takes design teams along new paths, and produces completely new product concepts that would not have been tried otherwise. In the Age of Experience, Simulation powers Innovation. And not just in some cases. The potential is now to transform the role of the Analyst from “solve” to “innovate.” Using concrete examples, this general lecture will illustrate this point, show how users can become innovators, and illustrate the three essential things in the users’ arsenal of tools that are needed to navigate this transformation and increase your value within your company. Tuesday, May 19, 17:30, Potsdam I/III

General Lecture 2: Simulation on the 3DEXPERIENCE Platform: Added Benefits for Existing Abaqus Users
The Dassault Systemes 3DEXPERIENCE platform delivers integrated data management, collaboration, business dash boarding, cloud computing and mobility to your organization. The 3DEXPERIENCE platform provides additional functionality to simulation users to supplement their existing tools. Through concrete technical examples, this lecture will highlight several of these added values such as direct modeling capabilities, best in class batch-meshing and collaborative modeling for large assemblies, and engineering templates. Also shown will be process integration and automation tools for all users enabling easy capture of manual simulation processes and big data analysis via results analytics to assimilate simulation data. The examples shown will illustrate agility and the ability to change simulation processes as needed, empowering you to bring your products to market faster. Wednesday, May 20, 17:00, Potsdam I/III

General Lecture 3: Technology, Functional, and Usage Updates in the Power of the Portfolio for 2015
Simulation technology is at the heart of our past, present, and future at SIMULIA. Our established product portfolio (Abaqus, Isight, fe-safe, and Tosca) continues to deliver the power that drives simulation applications and innovation. These technologies remain at the core of our business and vision. On the foundation provided by the simplified Extended Packaging strategy for all of these products, SIMULIA is delivering more technology than ever before to our core existing user base. This lecture will review release plans for 2015 for these products, coming enhancements, functional usage, and highlighted examples of existing and new applications that benefit from these enhancements. Thursday, May 21, 9:15, Potsdam I/III

Early-Bird Hands-on Sessions
Schöneberg II
Wednesday and Thursday
Begins 60 minutes prior to the conference
Occurring for one hour before the conference opens on Wednesday and Thursday, the Early-Bird Hands-on sessions provide an opportunity for interested attendees to come and explore the processes, functionality, user interfaces, and scope of the 3DEXPERIENCE platform, the platform that provides the basis for a radically expanded visibility, value, and role of the analyst within his or her company.

Based on Abaqus technology, the simulation processes within the 3DEXPERIENCE platform provides you with an unparalleled opportunity to amplify the importance of your simulation work to your enterprise. But don’t take our word for it, come and see for yourself! Take advantage of this unique opportunity to test drive the software while guided by experienced hands from our field support team.

Lunch Groups
At this year’s SCC, attendees have been divided into two lunch groups: Lunch Group 1 and Lunch Group 2. Lunch Group 1 will eat lunch in Potsdam II at 12:15 Tuesday, Wednesday and Thursday, while Lunch Group 2 will eat lunch at 13:15 Tuesday, Wednesday and Thursday. Your SCC attendee badge will say which lunch group you are in.
Alternative with the lunches, we will also present our seven Technology & Product Updates (see page 8 for topics and room locations). These updates will repeat during lunchtime and again Thursday afternoon so you will be able to attend up to four updates during this year’s SCC!

Support Desk–Schöneberg II
Open during breaks, lunch, and technical paper sessions. Customer Support and application engineers will be available for on-one help and advice with your current simulation models at the Support Desk. No appointment necessary!
TUESDAY, MAY 19

**cosin scientific software—Booth 12**

**FTire, the Virtual Tire in MBS and FE Environments: Physics Beats Magic**
Mr. Gerald Hofmann

Cosin scientific software is the developer of FTire, the leading higher-frequency tire model in the market.

**Wolf Star Technologies, LLC—Booth 19**

**Load Measurement and Linear Dynamics with Wolf Star Technologies**
Dr. Tim Hunter

Wolf Star Technologies provides first to market solutions for Abaqus/CAE: True-Load™ and True-LDE™ generate strain correlated loading and intuitive post-processing for dynamic FEA solutions.

**GNS mbH—Booth 11**

**Managing Large Abaqus Simulation Results for Efficient File Transfer and Storage**
Mr. Christoph Kaulich

GNS mbH is a German engineering company. Customers, mainly from the automotive industry, are provided with engineering services and high quality software.

**BETA CAE Systems SA—Booth 18**

**Complete Laminated Composite Modeling and Simulation Process Using the ANSA and μETA Software Tools for the SIMULIA Abaqus Solver**
Dr. Ioannis Nerantzis

BETA CAE Systems is a CAE software company committed to the development of high performance systems for demanding simulation processes. JSOL—Booth 21

JMag—Booth 21

**JMAG: Simulation Technology for Electromechanical Design**
Thiebauf Pfister

JMAG is comprehensive electromechanical design software and a standard tool in companies leading HEV/EV technology. JMAG is enhancing its functionality through a partnership with SIMULIA.

Zentech—Booth 7

**Application of Abaqus to Analysis of 3D Cracks and Fatigue Crack Growth Prediction**
Mr. Chris Timbrell

Zentech develops software for 3D crack analysis and crack growth prediction. The software, ZenCrack, uses Abaqus as a solution engine to drive fracture mechanics analyses.

SGI—Booth 1

**Innovative Solutions for Complex FEA Problems**
Mr. Scott Shaw

SGI is a global leader in high performance solutions for compute, data analytics and data management that enable customers to accelerate time to discovery, innovation and profitability.

Cradle North America—Booth 20

**More Innovation Using 2-way Direct Coupling Using SC/Tetra and Abaqus for FSI Simulations**
Mr. Yuya Ando

Software Cradle is a leading provider of Computational Fluid Dynamics (CFD) software including SC/Tetra (unstructured mesh), scSTREAM (Cartesian mesh), and HeatDesigner (Cartesian mesh for electronics).

WEDNESDAY, MAY 20

**CD-adapco—Booth P1**

**A Study of the Flow Induced Vibration of a JUMPER Due to Multi-Phase Internal Flow with an Objective of Understanding Life-Cycle Fatigue**
Dr. Alan Mueller

CD-adapco is the world’s largest independent CFD focused provider of engineering simulation software, support and services.

**Granta—Booth 22**

**Materials Gateway 3.0 for Abaqus/CAE - Enterprise-wide Materials Knowledge for Abaqus Users**
Sebastian Schwaegele

Granta MI: Materials Gateway from Granta Design provides access to enterprise-wide materials knowledge for Abaqus users. Get materials properties and related information within your Abaqus/CAE environment.

**Endurica—Booth 13**

**Fatigue Analysis of an Elastomeric CV Joint Boot Cover with fe-safe/Rubber**
Dr. William Mars

Endurica LLC, the world’s most versatile and best-validated elastomeric fatigue life simulation software system, empowers users to analyze fatigue performance of elastomers under real-world conditions.

**CST—Booth P2**

**Adding Electromagnetics to your Multiphysics Simulations**
Dr. Irina Munteanu

Founded in 1992, CST offers the market’s widest range of 3D electromagnetic field simulation tools. CST develops high-performance software for the simulation of electromagnetic fields in all frequency bands. Today, the company enjoys a leading position in the 3D EM simulation market and employs 260 personnel around the world.
**THURSDAY, MAY 21**

**MAGNA Engineering Center Steyr GmbH & Co KG—Booth 14**

**Using Abaqus for Fatigue Analysis of Components with Large Displacements (e.g. rubber)**

Mr. Axel Werkhausen

Engineering Center Steyr (ECS), a company of Magna International, is an internationally acknowledged engineering supplier. As a full-service provider ECS develops innovative products and solutions for the vehicles of the future.

**ThermoAnalytics—Booth 23**

**Advanced Automotive Thermal Simulation Process**

Antti Jussila

ThermoAnalytics provides Total Thermal Solutions for complex heat problems and engineering analysis. Our RadTherm software is the leading thermal analysis program for rapid design and optimization of transient heat transfer problems.

**KEONYS—Booth 5**

**How SIMULIA Products Accelerate Composite Design Process**

Mr. David Da Silva

At the heart of the digital company, Keonys has been a preferred PLM (Product Lifecycle Management) and 3D solutions partner to customers of all sizes and in all industries, to develop and manufacture better products and innovative services.

**Simpleware—Booth 4**

**Simpleware and Abaqus: Bridging Image Data and Simulation**

Mr. Theo Verbruggen

Simpleware develops world-class software solutions for converting 3D image data into models for design and simulation in CAD, CAE and 3D Printing applications.

**CEI GmbH—Booth 9**

**Huge and complex results data, and now? EnSight, Postprocessor for Scientific and Engineering Simulation Data**

Mr. Arnold Holler

Creator of EnSight, the solution for visualizing, analyzing, and communicating data from complex mono- and multidisciplinary numerical simulations, supporting shared- and distributed-memory parallel-processing and parallel-rendering.

**CATIA—3DS Playground**

**Release of SFE CONCEPT 2015 - Progress and Continuity in Simulation-driven Design**

Nikolai Baumeister

Wednesday, May 20 continued
OUR GOAL IS TO PROVIDE MORE TECHNICAL DETAIL AT THE 2015 SIMULIA COMMUNITY CONFERENCE THAN EVER BEFORE. THE TECHNOLOGY & PRODUCT UPDATE SESSIONS ARE AN IMPORTANT PART OF THE TECHNICAL PROGRAM OFFERING. FIRST OFFERED AT THE 2014 CONFERENCE AS TECHNICAL DEEP DIVES, OUR RENAMED TECHNOLOGY & PRODUCT UPDATES OFFER SEVEN CONCURRENT SESSIONS ON TECHNICAL TOPICS OF GREAT INTEREST TO ATTENDEES. THESE SESSIONS PROVIDE DETAILED INFORMATION STRAIGHT FROM THE SIMULIA R&D, CENTERS OF SIMULATION EXCELLENCE AND USER EXPERIENCE TEAMS THAT YOU CAN USE IMMEDIATELY. INFORMATION COVERED MAY INCLUDE (BUT IS NOT LIMITED TO) NEW ENHANCEMENTS, UNDER-UTILIZED FUNCTIONALITY, NEW APPLICATIONS, BEST PRACTICES AND TIPS AND TRICKS PERTAINING TO THE TOPIC.

UNLIKE THE PLENARY SESSIONS WHERE OPPORTUNITY TO INTERACT IS LIMITED, THESE SESSIONS ARE SPECIFICALLY DESIGNED TO ENCOURAGE INTERACTION WITH PRESENTERS. EACH SESSION WILL REPEAT SIX TIMES DURING THE EVENT. WITHIN EACH SESSION, INFORMATION WILL BE PRESENTED TOPIC BY TOPIC WITH AN OPPORTUNITY FOR Q&A BETWEEN EACH TOPIC. OF THE SEVEN SESSIONS, ATTENDEES WILL BE ABLE TO ATTEND ONE EACH DAY DURING THE CONCURRENT LUNCH, THEN ATTEND A FOURTH UPDATE ON THURSDAY AFTERNOON. THE PRESENTATIONS FROM ALL SEVEN SESSIONS WILL BE MADE AVAILABLE ON THE SIMULIA LEARNING COMMUNITY TO ALL SCC ATTENDEES AFTER THE CONFERENCE. WE HOPE YOU ENJOY THE TECHNOLOGY & PRODUCT UPDATE SESSIONS.

TECHNOLOGY & PRODUCT UPDATES

<table>
<thead>
<tr>
<th>Number</th>
<th>Session Title</th>
<th>Room</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Material Constitutive Modeling</td>
<td>Charlottenburg III</td>
</tr>
<tr>
<td>2</td>
<td>Abaqus Update: Contact</td>
<td>Charlottenburg II</td>
</tr>
<tr>
<td>3</td>
<td>Optimization (Isight and TOSCA) Update</td>
<td>Charlottenburg I</td>
</tr>
<tr>
<td>4</td>
<td>Introduction to 3D EXPERIENCE Roles</td>
<td>Tiergarten III</td>
</tr>
<tr>
<td>5</td>
<td>Fatigue &amp; Durability Update</td>
<td>Tiergarten II</td>
</tr>
<tr>
<td>6</td>
<td>Abaqus Update: Linear Dynamics</td>
<td>Tiergarten I</td>
</tr>
<tr>
<td>7</td>
<td>Best Practices in Abaqus/CAE</td>
<td>Tegel</td>
</tr>
</tbody>
</table>

Usability Testing

Help us develop the most usable software and participate in Usability Studies at the SCC! Sit with trained usability researchers and complete typical tasks in various parts of our software. Help us understand how findable and easy it is to learn new or enhanced aspects of our software. Give us your feedback on everything you see and do!

What is it?
Usability studies put a participant - you - in front of in-development software to complete a number of tasks, thus allowing our developers to get the feedback they need as improvements and enhancements are added to our software.

Why would you do this?
Participating in a Usability Study lets you, the customer, give your voice to us and give feedback prior to the release of software. It makes you a partner in our development effort.

Interested in helping? Sessions start on Monday afternoon and run throughout the conference. Contact the Usability Research team to schedule a session in room Schöneberg I.

SIMULIA Alliance Partner Free Hands-on Demonstrations: Learn how to Leverage your SIMULIA Investment

Six SIMULIA Alliance Partners (BETA CAE Systems, CD-adapco, Endurica, GNS, Granta Design, and Wolf Star Technologies) will be providing a 30-minute hands-on demonstration session at the SCC. The demonstration will show how their software integrates with SIMULIA software to create a complete, seamless solution. Attending the sessions is free.

Sign up at the SIMULIA Booth in the Pavillon.

Demonstrations will take place in Schöneberg III.
In close conjunction with the Technology & Product Updates is another opportunity to interact with SIMULIA technology: the Meet the Experts sessions. These sessions are constructed to facilitate and encourage direct one-to-one interaction between attendees and the SIMULIA team on technical topics of great interest. At the Meet the Experts sessions, attendees can gather and meet directly with technology leaders within SIMULIA to ask questions, make suggestions, ask for help, exchange business cards, and engage with the teams that are directly responsible for putting the power in the SIMULIA portfolio. We expect a lively, engaging rapid fire session where attendees will learn not only from asking their own questions, but from listening to the questions and answers from others. Emphasis will be placed on answering as many questions as possible, so have your questions ready and come prepared! Attendees are free to float from one topic to another within these sessions.

The list of Meet the Experts topics (subject to change) is as follows:

<table>
<thead>
<tr>
<th>Wednesday and Thursday, May 20-21, at 14:45</th>
<th>Room</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. <strong>Linear Dynamics, Vibration Response, NVH</strong></td>
<td>Tegel</td>
</tr>
<tr>
<td>2. <strong>Geomechanics</strong></td>
<td>Tegel</td>
</tr>
<tr>
<td>3. <strong>Contact Modeling</strong></td>
<td>Charlottenburg I</td>
</tr>
<tr>
<td>4. <strong>Analysis Techniques: Co-simulation, Import, Submodeling, Restart</strong></td>
<td>Charlottenburg I</td>
</tr>
<tr>
<td>5. <strong>Materials Modeling</strong></td>
<td>Charlottenburg II</td>
</tr>
<tr>
<td>6. <strong>Extreme Deformation: CEL, SPH and DEM</strong></td>
<td>Charlottenburg II</td>
</tr>
<tr>
<td>7. <strong>Fracture Mechanics</strong></td>
<td>Charlottenburg III</td>
</tr>
<tr>
<td>8. <strong>Modeling Features: Constraints, Connectors, Loading, Best Practice</strong></td>
<td>Charlottenburg III</td>
</tr>
<tr>
<td>9. <strong>Performance &amp; Equation Solvers</strong></td>
<td>Köpenick III</td>
</tr>
<tr>
<td>10. <strong>Fatigue</strong></td>
<td>Tiergarten I</td>
</tr>
<tr>
<td>11. <strong>Tosca</strong></td>
<td>Tiergarten II</td>
</tr>
<tr>
<td>12. <strong>Isight</strong></td>
<td>Tiergarten III</td>
</tr>
<tr>
<td>13. <strong>Abaqus/CAE Techniques</strong></td>
<td>Potsdam II</td>
</tr>
<tr>
<td>14. <strong>Abaqus/Explicit: Impact, Crash, Drop</strong></td>
<td>Potsdam II</td>
</tr>
</tbody>
</table>

**Make an Appointment**

Starting at 14:30 Tuesday, if you require additional assistance with topics not covered by the Technology & Product Updates and Meet the Expert sessions, please stop by Room Schöneberg II and make a one-on-one appointment! We will match you up with a SIMULIA technical expert dependent on your inquiry and type of help that you need. Appointments will take place all day Wednesday and Thursday from Room Schöneberg II.
**TRAINING DAY–MONDAY, MAY 18**

<table>
<thead>
<tr>
<th>Schedule</th>
<th>Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Training Day Begins</td>
<td>9:30</td>
</tr>
<tr>
<td>Break</td>
<td>10:30</td>
</tr>
<tr>
<td>Lunch</td>
<td>12:30</td>
</tr>
<tr>
<td>Break</td>
<td>15:30</td>
</tr>
<tr>
<td>Conclusion</td>
<td>17:30</td>
</tr>
<tr>
<td>Board buses for Opening Reception</td>
<td>17:45</td>
</tr>
</tbody>
</table>

### Technology Seminars

**Advanced Modeling, Simulation, and Analysis Techniques in Geomechanics including New Developments—Room: Charlottenburg III**

Deepak Datye, Chris Wohlever, Wouter Van Der Zee (Baker Hughes), Jean Daniel Lecuyer

This seminar provides an in-depth introduction to the variety of geomechanics problems solvable within the SIMULIA ecosystem today. Solution technology, functionality, usage and best practices are discussed in-depth to provide attendees with a solid foundation on how to address simulation problems in this domain. Advanced users will find coverage of under-utilized functionality and new developments to be valuable and will increase their expertise in this challenging field. Coverage of solution techniques available with partner products is included. Industrial applications will be presented. This seminar is the initial component of a complete program of material available at the 2015 SCC focusing on geomechanics simulation and is recommended for all engineers working in the geomechanics field.

**Fracture/failure Simulation: Technology and Techniques—Room: Tiergarten II**

Zhen-Zhong Du and Rassin Grantab

This seminar covers methods, technology, applications, techniques and recent developments in modeling fracture/failure with Abaqus. Topics such as J-integrals, CAE support and meshing, X-FEM, VCCT, cohesive elements, and material failure technology are covered. Attendees will learn how to define, model, apply, execute, and post-process models including fracture/failure functionality. Industrial applications of the techniques will be highlighted, including customer success stories to broaden the dialog. Both new and experienced users of these techniques will benefit as the topic and technology have advanced rapidly in recent years.

**Strategies and Tools for Material Model Calibration—Room: Charlottenburg I**

Tod Dalrymple

The key to success with FEA simulation is correlation to the real-world behavior of physical behavior and products. Lack of physical correlation is often directly related to the lack of accuracy in the material model. This class will give you extra tools to assess the material models that you are currently using and showcase new material models in Abaqus that might better capture the physics important to your component and its loading environment. We will cover advanced material modeling for metals, elastomers, polymers and engineering plastics. Attendees will come away with a thorough understanding of best practices that they can immediately apply in their projects to improve accuracy and performance.

### Hands-on Training Courses

**Hands-on with Coupled-Eulerian Lagrangian (CEL) technology: Straightforward Modeling of Extreme Deformation—Room: Köpenick III**

Paul Specker

The Eulerian analysis capability in Abaqus/Explicit allows for effective modeling of fluid flows and extremely large deformations in solids. Coupling the power of this capability with the traditional Lagrangian approach makes it possible to simulate complicated multiphysics phenomena such as fluid-structure interactions where highly deformable materials interact with relatively stiff bodies. This hands-on course with workshops aims to provide users with an understanding of the Coupled Eulerian-Lagrangian (CEL) method and illustrate approaches to setting-up and analyzing a variety of real-world problems using this advanced analysis method.

**Hand-on with Substructures: Simulating Complex Models More Efficiently—Room: Tiergarten III**

Jens Lueke

The ever-increasing size and complexity of modern designs pose challenges to today's analyst as conventional modeling approaches are ill-suited to simulate and validate them effectively. Substructures offer an attractive alternative because they allow the analyst to break a large problem into several smaller components and subsequently re-use the components to create arbitrarily complex models capturing advanced physical behaviors. With careful use, substructures can speed up solution time in both linear and nonlinear problems by order of magnitude. This hands-on course with workshops focuses on using the Abaqus/CAE interface to substructures and aims to provide users with an understanding of the substructure method and illustrate its usage through examples.
Application Seminars

Practical Usage of fe-safe for Durability Simulation for Abaqus Users—Room: Charlottenburg II
Anders Winkler and Pawel Sobczak
Through Extended Packaging, all Abaqus users now have access to fe-safe, Tosca and Isight products. This course will focus on the practical benefits and usage of fe-safe along with Abaqus to extend the attendee’s simulation skillset to include durability simulation. By considering a product’s durability, an Abaqus user can not only simulate the performance, stress, deflection and load behavior of his models from one-time load application, but can also simulate the lifecycle, durability, likely failure modes and fatigue behavior because of many repeated loading cycles. This capability allows analysts to improve the quality, functionality, durability and usable life of their company’s products. The course will teach fundamentals of fatigue technology and application in practical settings and will provide a firm foundation in using fe-safe in a user’s routine simulation activities.

Practical Usage of Tosca Structure for Abaqus Users—Room: Tiergarten I
Steven Ribeiro-Ayeh
Through Extended Packaging, all Abaqus users now have access to fe-safe, Tosca, and Isight products. This course will focus on the practical benefits and usage of Tosca Structure along with Abaqus to extend the attendee’s skillset to include topology and shape optimization. By optimizing a product’s topology and shape, an Abaqus user can not only simulate the performance, stress, deflection and load behavior of his models, but can also offer design guidance to specifically improve that performance as well as increase reliability, reduce weight, and minimize cost of the design. Using worked examples illustrating the Power of the Portfolio, the course will teach fundamentals of shape and topology optimization in practical settings and will provide a firm foundation in using Tosca in a user’s routine simulation activities.

Receptions and Wednesday Gala

Monday, May 18, 17:45
Join us at the front entrance to the Intercontinental at 17:45 as we board buses for a tour of Berlin. Learn about Berlin’s place in history and see some of its most important architecture—from the Brandenburg Gate to Checkpoint Charlie and parts of the Berlin Wall that are still standing. After the tour, we’ll stop at Postdamer Platz, one of the most prominent business districts in Berlin and site of the famous Grand Hotel Esplanade for our reception in the Palmenhof salon.

Tuesday, May 19, 18:30
Explore our exhibit hall and shake hands with our conference sponsors. Our annual SCC Partner Reception is the perfect time to make new contacts and reconnect with old ones. Don’t miss the DS Playground with the SIMULIA booth front and center. This is the time the SIMULIA Community really shines! The reception will be held in the Pavilion at the Intercontinental Hotel at 18:30. Don’t be late, the drinks are on us!

Wednesday, May 20, 18:00
Gather at the entrance to the Intercontinental at 18:00 to board buses for our group photo and gala evening. The group photo will be taken on the steps of Germany’s Chancellor’s office. Then we’ll walk across to the world-famous TIPI AM KANZLERMAT, a marquee theater regarded as one of Berlin’s most exceptional event locations for our reception and banquet. Sit back and be amazed as world-famous magician Stefan Leyshon performs for us. You won’t believe your eyes.
## AGENDA – TUESDAY, MAY 19

See page 66 for the conference floor plan

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:00</td>
<td>REGISTRATION</td>
</tr>
<tr>
<td>9:00</td>
<td>CONFERENCE KICKOFF, Potsdam I/III</td>
</tr>
<tr>
<td>9:15</td>
<td>EXECUTIVE REMARKS: Bernard Charlès, President &amp; CEO, Dassault Systèmes, introduced by Scott Berkey, CEO, SIMULIA, and NAM Managing Director</td>
</tr>
<tr>
<td>9:45</td>
<td>KEYNOTE: Overview of R350 Numerical Simulation Successes&lt;br&gt;Dominique Moreau, Head of Airframe Technical Authority, Airbus</td>
</tr>
<tr>
<td>10:45</td>
<td>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</td>
</tr>
<tr>
<td>11:15</td>
<td>MULTI-BODY SIMULATION FORUM IN BELLEVUE</td>
</tr>
<tr>
<td>11:15</td>
<td>ENERGY, PROCESS &amp; UTILITIES</td>
</tr>
<tr>
<td>11:15</td>
<td>CHARLOTTENBURG III</td>
</tr>
<tr>
<td>11:15</td>
<td>CHARLOTTENBURG II</td>
</tr>
<tr>
<td>11:15</td>
<td>AEROSPACE &amp; DEFENSE</td>
</tr>
<tr>
<td>11:15</td>
<td>COMPOSITES</td>
</tr>
<tr>
<td>11:15</td>
<td>Process Integration Between CAD and CAE</td>
</tr>
<tr>
<td>11:15</td>
<td>Applications for Isight Based DOE Analysis</td>
</tr>
<tr>
<td>11:15</td>
<td>Davide Pinna</td>
</tr>
<tr>
<td>11:15</td>
<td>Amsaldo Energia</td>
</tr>
<tr>
<td>11:45</td>
<td>Hydraulic Fracturing Simulation for Fracture Networks</td>
</tr>
<tr>
<td>11:45</td>
<td>Stephan Rindt</td>
</tr>
<tr>
<td>11:45</td>
<td>Baker Hughes</td>
</tr>
<tr>
<td>11:45</td>
<td>Pressure Vessel New Design Subjected to Blast Loads</td>
</tr>
<tr>
<td>11:45</td>
<td>Laurence Cousin</td>
</tr>
<tr>
<td>11:45</td>
<td>CER/DRM/DIF</td>
</tr>
<tr>
<td>11:45</td>
<td>Methodology for the Analysis of Tolerances in the Assembly Process of a Wing Torsion Box Through FE Simulations Ruben Breto Pontaque ITRAI NOVA</td>
</tr>
<tr>
<td>13:15</td>
<td>LUNCH GROUP 1</td>
</tr>
<tr>
<td>13:15</td>
<td>LUNCH GROUP 2</td>
</tr>
<tr>
<td>14:15</td>
<td>SPECIAL GUEST: Dr. Martin Hilchenbach, Max Planck Institute for Solar System Research Rosetta Philae - Impact on a Comet as Envisaged by Multibody Simulations</td>
</tr>
<tr>
<td>15:00</td>
<td>MULTI-BODY SIMULATION FORUM IN BELLEVUE</td>
</tr>
<tr>
<td>15:00</td>
<td>INTEGRATED SOLUTIONS</td>
</tr>
<tr>
<td>15:00</td>
<td>INTEGRATED SOLUTIONS</td>
</tr>
<tr>
<td>15:00</td>
<td>INTEGRATED SOLUTIONS</td>
</tr>
<tr>
<td>15:00</td>
<td>TEGEL</td>
</tr>
<tr>
<td>15:00</td>
<td>CHARLOTTENBURG III</td>
</tr>
<tr>
<td>15:00</td>
<td>CHARLOTTENBURG II</td>
</tr>
<tr>
<td>15:00</td>
<td>ENERGY, PROCESS &amp; UTILITIES</td>
</tr>
<tr>
<td>15:00</td>
<td>AEROSPACE &amp; DEFENSE</td>
</tr>
<tr>
<td>15:00</td>
<td>COMPOSITES</td>
</tr>
<tr>
<td>15:00</td>
<td>Process Integration Between CAD and CAE</td>
</tr>
<tr>
<td>15:00</td>
<td>Applications for Isight Based DOE Analysis</td>
</tr>
<tr>
<td>15:00</td>
<td>Davide Pinna</td>
</tr>
<tr>
<td>15:00</td>
<td>Amsaldo Energia</td>
</tr>
<tr>
<td>15:00</td>
<td>Hydraulic Fracturing Simulation for Fracture Networks</td>
</tr>
<tr>
<td>15:00</td>
<td>Stephan Rindt</td>
</tr>
<tr>
<td>15:00</td>
<td>Baker Hughes</td>
</tr>
<tr>
<td>15:00</td>
<td>Pressure Vessel New Design Subjected to Blast Loads</td>
</tr>
<tr>
<td>15:00</td>
<td>Laurence Cousin</td>
</tr>
<tr>
<td>15:00</td>
<td>CER/DRM/DIF</td>
</tr>
<tr>
<td>15:00</td>
<td>Methodology for the Analysis of Tolerances in the Assembly Process of a Wing Torsion Box Through FE Simulations Ruben Breto Pontaque ITRAI NOVA</td>
</tr>
<tr>
<td>15:30</td>
<td>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</td>
</tr>
</tbody>
</table>
| 16:00  | Optimizing Wind Turbine Bearing Performance Using Flexible Bearing Ring Analysis
| 16:00  | Jose Torres                                                          |
| 16:00  | Romax Technology                                                     |
| 16:00  | Bird Strike Analysis for Impact-resistant Design of Aircraft Wing Krueger Flap
| 16:00  | Sebastian Heimbs                                                     |
| 16:00  | Airbus Group                                                         |
| 16:00  | An Approach Towards an Optimal Design of Composite Structures Barbara Goller INTRILES GmbH |
| 16:30  | Use of FEA and Radial Basis Functions for Reliability-based Design and Assessment of Connection Sealsibility
| 16:30  | Jueren Xie                                                          |
| 16:30  | C-FER Technologies                                                   |
| 16:30  | Residual Strength of the Carbon Fiber Panel with Delamination
| 16:30  | Mikhail Pavlov                                                       |
| 16:30  | TsAGI - Central Aerohydrodynamic Institute                           |
| 16:30  | Shrinkage Modeling of Thermoplastic Wing Rib
| 16:30  | Nikita Rozin                                                         |
| 16:30  | TsAGI - Central Aerohydrodynamic Institute                           |
| 17:00  | Finite Element Simulation of Rotary Steerable Drilling Systems Advance Understanding of Damaging Downhole Shock and Vibration
| 17:00  | Nader Abedrabbo                                                      |
| 17:00  | Weatherford                                                          |
| 17:00  | Enabling Multi-Fidelity Design Optimization Approaches in Isight
| 17:00  | Timoleon Kipouros                                                   |
| 17:00  | University of Cambridge                                            |
| 17:00  | Nested Micromechanical and Structural Models for the Analysis of Discontinuous Long-fiber Thermoplastic Composite Materials and Structures Hakan Kilic Greene, Tweed & Co. |
| 17:30  | GENERAL LECTURE 1: Powering Innovation in the Age of Experience, SIMULIA, Potsdam I/III |
| 18:30  | PARTNER RECEPTION IN THE EXHIBIT HALL                                |

For information about the lunch schedule and which group you are in, please see page 5.
### Charlottenburg I

**Transportation & Mobility I**
- Study on Body Rigidity, Strength and Fatigue Performance for Carbon Fiber Reinforced Plastics Vehicles
  - Oh Se-Hee
  - Hyundai Motor Company

**Integrated Solutions**
- Complete Laminated Composite Modeling and Simulation Process Using the ANSYS and pETRA Software Tools for the SIMULIA Abaqus Solver
  - BETRA CRE Systems S.A.
  - Ioannis Nerantzis

### Köpenick III

**Transportation & Mobility II**
- Simulation of Pipe Bending Process with Abaqus
  - Satoshi Ishikawa
  - IDAI Co., LTD.

**Integrated Solutions**
- JMRG: Simulation Software for Electromechanical Design
  - Thiebaud Pfister

### Tiergarten III

**Life Sciences**
- Drop Test of a Hearing Aid
  - Morten Birkmose Søendergaard
  - GN Resound R/S

**Integrated Solutions**
- Developing a FE Head Model for Impact Simulation in Abaqus
  - Philippe Young
  - Simpleware

### Tuesday’s agenda continues on the next page.
## AGENDA—TUESDAY, MAY 19

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:00</td>
<td>REGISTRATION</td>
</tr>
<tr>
<td>9:00</td>
<td>CONFERENCE KICKOFF, Potsdam I/III</td>
</tr>
<tr>
<td>9:15</td>
<td>EXECUTIVE REMARKS: Bernard Charlès, President &amp; CEO, Dassault Systèmes, introduced by Scott Berkey, CEO, SIMULIA, and NAM Managing Director</td>
</tr>
<tr>
<td>9:45</td>
<td>KEYNOTE: Overview of A350 Numerical Simulation Successes</td>
</tr>
<tr>
<td></td>
<td>Dominique Moreau, Head of Airframe Technical Authority, Airbus</td>
</tr>
<tr>
<td>10:45</td>
<td>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</td>
</tr>
<tr>
<td>11:15</td>
<td>TIERGARTEN II</td>
</tr>
<tr>
<td></td>
<td>MULTI-BODY SIMULATION FORUM IN BELLEVUE</td>
</tr>
<tr>
<td></td>
<td>INDUSTRIAL EQUIPMENT</td>
</tr>
<tr>
<td></td>
<td>CONSUMER PACKAGED GOODS</td>
</tr>
<tr>
<td>11:15</td>
<td>Robust and Optimal Strain Gauge Positions for Blade Vibration Testing</td>
</tr>
<tr>
<td></td>
<td>Piyawan Teufel</td>
</tr>
<tr>
<td></td>
<td>ABB Turbo Systems AG</td>
</tr>
<tr>
<td>11:45</td>
<td>Design Optimization of a Curved Fir-Tree Root of Steam Turbine Using Computer Aided Optimization and Finite Element Analysis</td>
</tr>
<tr>
<td></td>
<td>Shang Jiong Tan</td>
</tr>
<tr>
<td></td>
<td>Shanghai Turbine Works Company Ltd</td>
</tr>
<tr>
<td>12:15</td>
<td>LUNCH GROUP 1</td>
</tr>
<tr>
<td>13:15</td>
<td>LUNCH GROUP 2</td>
</tr>
<tr>
<td>14:15</td>
<td>SPECIAL GUEST: Dr. Martin Hilchenbach, Max Planck Institute for Solar System Research</td>
</tr>
<tr>
<td></td>
<td>Rosetta Philae - Impact on a Comet as Envisaged by Multibody Simulations</td>
</tr>
<tr>
<td>15:00</td>
<td>INTEGRATED SOLUTIONS</td>
</tr>
<tr>
<td></td>
<td>INTEGRATED SOLUTIONS</td>
</tr>
<tr>
<td>16:00</td>
<td>Coupled Analysis of Fracture Mechanics and Piezoelectricity in Active layers in the Abaqus Code Operated Through the Isight Tool</td>
</tr>
<tr>
<td></td>
<td>Mauro Parodi</td>
</tr>
<tr>
<td></td>
<td>Politecnico Di Torino</td>
</tr>
<tr>
<td>16:30</td>
<td>XFEM Simulation of Radial-Median Crack Evolution in Knoop Indentation of Brittle Materials</td>
</tr>
<tr>
<td></td>
<td>Felix Rickhey</td>
</tr>
<tr>
<td></td>
<td>Sogang University</td>
</tr>
<tr>
<td>17:00</td>
<td>Evaluation of the Fatigue Strength of Threaded Connections using Local Concepts</td>
</tr>
<tr>
<td></td>
<td>Dominic Olveda</td>
</tr>
<tr>
<td></td>
<td>Technische Universität Darmstadt, Construction Materials Center, Darmstadt</td>
</tr>
<tr>
<td>17:30</td>
<td>GENERAL LECTURE 1: Powering Innovation in the Age of Experience, SIMULIA, Potsdam I/III</td>
</tr>
<tr>
<td>18:30</td>
<td>PARTNER RECEPTION IN THE EXHIBIT HALL</td>
</tr>
</tbody>
</table>

For information about the lunch schedule and which group you are in, please see page 5.
The Multi-Body Simulation Forum is a special parallel track, supplementing Tuesday’s technical program of the SCC 2015. During the Forum, SIMULIA will showcase the worldwide leading Multi-Body Simulation (MBS) software SIMPACK, which is used for the dynamic analysis of any mechanical or mechatronic system. SIMPACK enables engineers to generate and solve virtual 3D models in order to predict and visualize motion, coupling forces and stresses. Primarily used in the automotive, engine, HiL/SiL, power transmission, railway, and wind energy industrial sectors, SIMPACK can be applied to any branch of mechanical engineering.

Since July 2014, SIMPACK is the strategic MBS component of the Dassault portfolio where a seamless integration between tools in the 3DEXPERIENCE is quickly becoming reality. Already there have been parameter and structural optimization of SIMPACK models using the tools Isight and Tosca and new coupling possibilities with Abaqus. The Multi-Body Simulation Forum provides you with the great opportunity to find out more about SIMPACK itself, to learn about new modeling and analysis strategies and to see how SIMPACK is utilized by customers.

Attendees are welcome to go from the technical tracks to the forum and back again throughout the day.

**Who should attend:**
- Those who are new to SIMPACK and would like to find out more about the software.
- Those who are already familiar with MBS and are looking to find out about the latest SIMPACK and SIMULIA developments.

<table>
<thead>
<tr>
<th>BELLEVUE</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>11:15 SIMPACK Technology</td>
<td>Wolfgang Trautenberg, SIMPACK GmbH</td>
</tr>
<tr>
<td>11:30 SIMPACK Applications</td>
<td>Steve Mulski, SIMPACK GmbH</td>
</tr>
<tr>
<td>12:15 Lunch Group 1 / Technology &amp; Product Updates</td>
<td></td>
</tr>
<tr>
<td>13:15 Lunch Group 1 / Technology &amp; Product Updates</td>
<td></td>
</tr>
<tr>
<td>14:15 SPECIAL GUEST: Dr. Martin Hilchenbach (Potsdam I/III)</td>
<td></td>
</tr>
<tr>
<td>15:00 Flexible Bodies in SIMPACK</td>
<td>Stefan Dietz, SIMPACK GmbH</td>
</tr>
<tr>
<td>15:30 BREAK &amp; PARTNER EXHIBITION</td>
<td></td>
</tr>
<tr>
<td>16:00 Making Virtual Road Load Data a Reality at Jaguar Land Rover Using SIMPACK</td>
<td>Prashant Khapane, Jaguar Land Rover</td>
</tr>
<tr>
<td>16:30 Analyzing the World Largest Bucket Wheel Excavator Using SIMPACK</td>
<td>Berthold Schlecht, TU Dresden, IMM</td>
</tr>
<tr>
<td>17:00 SIMPACK Coming Attractions</td>
<td>Wolfgang Trautenberg, SIMPACK GmbH</td>
</tr>
<tr>
<td>17:20 SIMPACK Q&amp;A</td>
<td>Steve Mulski, Stefan Dietz, Wolfgang Trautenberg, SIMPACK GmbH</td>
</tr>
<tr>
<td>17:30 GENERAL LECTURE 1 (Potsdam I/III)</td>
<td></td>
</tr>
</tbody>
</table>
## AGENDA – WEDNESDAY, MAY 20

### 8:00
REGISTRATION & EARLY BIRD HANDS-ON SESSION, SCHÖNBERG II

### 9:00
WELCOME & EXECUTIVE REMARKS: SIMULIA Business Review, Scott Berkey, CEO SIMULIA & NAM Managing Director, POTSDAM I/III

### 9:25
KEYNOTE: Energy in the 21st Century—Building on the Fundamentals
Bruce A. Dale, Chief Subsurface Engineer, ExxonMobil Production Company

### 10:15
BREAK & PARTNER EXHIBITION IN THE EXHIBIT HALL

### 10:45
**FATIGUE & DURABILITY FORUM IN BELLEVUE**

<table>
<thead>
<tr>
<th>TEGEL</th>
<th>CHARLOTTENBURG III</th>
<th>CHARLOTTENBURG II</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:45</td>
<td>Validation of Simulation Capabilities for Hydraulic Fractures Propagating in a Porous Medium Jing Ning ExxonMobil Upstream Research Company</td>
<td>Time Accurate Simulation of Aeroelastic Flap Deployment Nicolas Reveles RTR Engineering, Inc.</td>
</tr>
<tr>
<td>11:15</td>
<td>Fracture Stability Assessment during SRGD operations using 4D Geomechanical Models Wouter van der Zee Baker Hughes</td>
<td>Simulation of High Velocity Perforation using Various Plasticity and Fracture Models Vladislav V. Vershin Moscow State University of Civil Engineering</td>
</tr>
</tbody>
</table>

### 12:15
LUNCH GROUP 1 | TECHNOLOGY & PRODUCT UPDATES (see page 8 for topics and room assignments)

### 13:15
LUNCH GROUP 2 | TECHNOLOGY & PRODUCT UPDATES (see page 8 for topics and room assignments)

### 14:15
**FATIGUE & DURABILITY FORUM IN BELLEVUE**

<table>
<thead>
<tr>
<th>INTEGRATED SOLUTIONS</th>
<th>INTEGRATED SOLUTIONS</th>
<th>INTEGRATED SOLUTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>14:15</td>
<td>S-Life - Integrated Static and Fatigue Strength Assessment with Abaqus According to German FKM Guideline PRRT Engineering GmbH Wolfgang Korte</td>
<td>A Study of the Flow Induced Vibration of a JUMPER Due to Multi-Phase Internal Flow with an Objective of Understanding Life-Cycle Fatigue CD-adapco Alan Mueller</td>
</tr>
</tbody>
</table>

### 14:45
MEET THE EXPERTS: FOR TOPICS - SEE PAGE 9

### 15:15
BREAK & PARTNER EXHIBITION IN THE EXHIBIT HALL | AUTHORS-ONLY RECEPTION - GARTEN LOUNGE

### 15:45
**FATIGUE & DURABILITY FORUM IN BELLEVUE**

<table>
<thead>
<tr>
<th>GEOMECHANICS SUPER TRACK</th>
<th>AEROSPACE &amp; DEFENSE</th>
<th>COMPOSITES</th>
</tr>
</thead>
</table>

### 16:45
GENERAL LECTURE 2: Simulation on the 3DEXPERIENCE Platform: Added Benefits for Existing Abaqus Users, SIMULIA, Potsdam I/III

### 17:30
ADJOURN FOR THE DAY

### 18:00
MEET OUTSIDE HOTEL LOBBY TO BOARD BUSES FOR GROUP PHOTO AND SCC GALA / 11 PM RETURN TO HOTEL

For information about the lunch schedule and which group you are in, please see page 5.
### AGENDA – WEDNESDAY, MAY 20

#### CHARLOTTENBURG I

<table>
<thead>
<tr>
<th>TRANSPORTATION &amp; MOBILITY I</th>
</tr>
</thead>
<tbody>
<tr>
<td>Efficient Evaluation of Material Cards for Non-reinforced and Reinforced Thermoplastics</td>
</tr>
<tr>
<td>Peter Retahofer 4a</td>
</tr>
</tbody>
</table>

#### KÖPENICK III

<table>
<thead>
<tr>
<th>TRANSPORTATION &amp; MOBILITY II</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Effect of Rubber in Strength Analysis for Insulator Assembly</td>
</tr>
<tr>
<td>Hoo-Gwang Lee</td>
</tr>
<tr>
<td>Hyundai Motor Company</td>
</tr>
</tbody>
</table>

#### TIERGARTEN III

<table>
<thead>
<tr>
<th>ENERGY PROCESS &amp; UTILITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>A Numerical Investigation on the Bedding Resistance of Laterally Displaced Pipelines</td>
</tr>
<tr>
<td>Tim Gerlach</td>
</tr>
<tr>
<td>Institute for Geotechnical Engineering, Leibniz University Hanover</td>
</tr>
</tbody>
</table>

---

#### INTEGRATED SOLUTIONS

<table>
<thead>
<tr>
<th>INTEGRATED SOLUTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fatigue Analysis of an Elastomeric CV Joint Boot Cover with fe-safe/Rubber</td>
</tr>
<tr>
<td>Endurica</td>
</tr>
<tr>
<td>William Mars</td>
</tr>
</tbody>
</table>

#### INTEGRATED SOLUTIONS

<table>
<thead>
<tr>
<th>INTEGRATED SOLUTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Adding Electromagnetics to your Multiphysics Simulations</td>
</tr>
<tr>
<td>Irina Munteanu</td>
</tr>
<tr>
<td>CST</td>
</tr>
</tbody>
</table>

#### INTEGRATED SOLUTIONS

<table>
<thead>
<tr>
<th>INTEGRATED SOLUTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Release of SFE CONCEPT 2015 - Progress and Continuity in Simulation-driven Design</td>
</tr>
<tr>
<td>CARI</td>
</tr>
<tr>
<td>Nikolai Baumeister</td>
</tr>
</tbody>
</table>

---

#### TRANSPORTATION & MOBILITY I

<table>
<thead>
<tr>
<th>TRANSPORTATION &amp; MOBILITY II</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automated Analysis in CARI V5</td>
</tr>
<tr>
<td>Stephen Wallis</td>
</tr>
<tr>
<td>Jaguar Land Rover</td>
</tr>
</tbody>
</table>

#### TRANSPORTATION & MOBILITY II

<table>
<thead>
<tr>
<th>TRANSPORTATION &amp; MOBILITY II</th>
</tr>
</thead>
<tbody>
<tr>
<td>Innovative Anisotropic Material Modelling Approach for Fiber Reinforced Thermoplastics</td>
</tr>
<tr>
<td>Recep Yaldiz</td>
</tr>
<tr>
<td>SABIC</td>
</tr>
</tbody>
</table>

#### ENERGY PROCESS & UTILITIES

<table>
<thead>
<tr>
<th>ENERGY PROCESS &amp; UTILITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Increasing the Efficiency of Offshore Rigid Pipeline Lateral Buckling Assessments Using a Dedicated GUI and Isight</td>
</tr>
<tr>
<td>Carlos Charnaux</td>
</tr>
<tr>
<td>Subsea 7</td>
</tr>
</tbody>
</table>

#### ENERGY PROCESS & UTILITIES

<table>
<thead>
<tr>
<th>ENERGY PROCESS &amp; UTILITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation of Combined Forming and Injection Molding Processes</td>
</tr>
<tr>
<td>Timo Mueller</td>
</tr>
<tr>
<td>Karlsruhe Institute of Technology (KIT)</td>
</tr>
</tbody>
</table>

#### ENERGY PROCESS & UTILITIES

<table>
<thead>
<tr>
<th>ENERGY PROCESS &amp; UTILITIES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation of Hydraulic Fracturing of Unconsolidated Sands using Fully Coupled Poro-Elastoplastic Models</td>
</tr>
<tr>
<td>Ganesh Dasari, ExxonMobil and Juan Hurtado, Dassault Systèmes</td>
</tr>
</tbody>
</table>

---

*Wednesday’s agenda continues on the next page.*
## AGENDA–WEDNESDAY, MAY 20

See page 66 for the conference floor plan

### 8:00
**REGISTRATION & EARLY BIRD HANDS-ON SESSION, SCHÖENBERG II**

### 9:00
**WELCOME & EXECUTIVE REMARKS: SIMULIA Business Review**, Scott Berkey, CEO SIMULIA & NAM Managing Director, POTSDAM I/III

### 9:25
**KEYNOTE: Energy in the 21st Century—Building on the Fundamentals**
Bruce A. Dale, Chief Subsurface Engineer, ExxonMobil Production Company

### 10:15
**BREAK & PARTNER EXHIBITION IN THE EXHIBIT HALL**

### 10:45
**FATIGUE & DURABILITY FORUM IN BELLEVUE**

<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:45</td>
<td><strong>TIERGARTEN II</strong></td>
<td><strong>TIERGARTEN I</strong></td>
</tr>
<tr>
<td>10:45</td>
<td><strong>INDUSTRIAL EQUIPMENT</strong></td>
<td><strong>ARCHITECTURE, ENGINEERING &amp; CONSTRUCTION</strong></td>
</tr>
<tr>
<td></td>
<td><strong>Strength and Fatigue Analysis of Leaf Springs used in Heavy Duty Trucks</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mehmet Bakir</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mercedes-Benz Turk R.S.</td>
<td></td>
</tr>
<tr>
<td>11:15</td>
<td><strong>Modelling of Hydrogen Embrittlement Mechanism in Metals.</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Olga Barrera</td>
<td></td>
</tr>
<tr>
<td></td>
<td>University of Oxford</td>
<td></td>
</tr>
<tr>
<td>11:45</td>
<td><strong>Strip Shape Prediction using a Simplified Static Model of a 4-High Cold Rolling Process.</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mario Alberto Bello-Gomez</td>
<td></td>
</tr>
<tr>
<td></td>
<td>UANL</td>
<td></td>
</tr>
<tr>
<td>12:15</td>
<td>**LUNCH GROUP 1</td>
<td>TECHNOLOGY &amp; PRODUCT UPDATES** (see page 8 for topics and room assignments)</td>
</tr>
<tr>
<td>13:15</td>
<td>**LUNCH GROUP 2</td>
<td>TECHNOLOGY &amp; PRODUCT UPDATES** (see page 8 for topics and room assignments)</td>
</tr>
<tr>
<td>14:15</td>
<td><strong>FATIGUE &amp; DURABILITY FORUM IN BELLEVUE</strong></td>
<td><strong>INTEGRATED SOLUTIONS</strong></td>
</tr>
<tr>
<td>14:15</td>
<td><strong>Faster Innovation - Accelerating SIMULIA Abaqus Simulations with NVIDIA GPUs</strong></td>
<td><strong>Huge and complex results data, and now? EnSight, Postprocessor for Scientific and Engineering Simulation Data</strong></td>
</tr>
<tr>
<td></td>
<td>NVIDIA</td>
<td><strong>CEI GmbH</strong></td>
</tr>
<tr>
<td></td>
<td>Baskar Rajagopalan</td>
<td><strong>Arnold Holler</strong></td>
</tr>
<tr>
<td>14:45</td>
<td><strong>MEET THE EXPERTS: FOR TOPICS - SEE PAGE 9</strong></td>
<td><strong>INTEGRATED SOLUTIONS</strong></td>
</tr>
<tr>
<td>15:15</td>
<td>**BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</td>
<td>AUTHORS-ONLY RECEPTION - GARTEN LOUNGE**</td>
</tr>
<tr>
<td>15:45</td>
<td><strong>FATIGUE &amp; DURABILITY FORUM IN BELLEVUE</strong></td>
<td><strong>MULTI-INDUSTRY</strong></td>
</tr>
<tr>
<td>15:45</td>
<td><strong>Validation of Flow Simulation on Abaqus/CEL</strong></td>
<td><strong>ARCHITECTURE, ENGINEERING &amp; CONSTRUCTION</strong></td>
</tr>
<tr>
<td></td>
<td>Edgar Ramirez</td>
<td><strong>Improvement and Enhancement of Concrete Model</strong></td>
</tr>
<tr>
<td></td>
<td>Universidad Nacional Autónoma de México (UNAM)</td>
<td><strong>Yuji Nikaido</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Kajima Corporation</strong></td>
</tr>
<tr>
<td>16:15</td>
<td><strong>Achieving Higher Productivity on Abaqus Simulations with HPC and Clustering Technologies</strong></td>
<td><strong>Tangential Plasticity Effect on Buckling Behavior of a Thin Wall</strong></td>
</tr>
<tr>
<td></td>
<td>Pak Lui (HPC Advisory Council)</td>
<td><strong>Pier under Cyclic Loading Conditions</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>Hideto Momii</strong></td>
</tr>
<tr>
<td></td>
<td></td>
<td><strong>ESTECH Corporation</strong></td>
</tr>
<tr>
<td>16:45</td>
<td><strong>GENERAL LECTURE 2: Simulation on the 3DEXPERIENCE Platform: Added Benefits for Existing Abaqus Users,</strong></td>
<td><strong>SIMULIA, Potsdam I/III</strong></td>
</tr>
<tr>
<td>17:30</td>
<td><strong>ADJOURN FOR THE DAY</strong></td>
<td></td>
</tr>
<tr>
<td>18:00</td>
<td><strong>MEET OUTSIDE HOTEL LOBBY TO BOARD BUSES FOR GROUP PHOTO AND SCC GALA / 11 PM RETURN TO HOTEL</strong></td>
<td></td>
</tr>
</tbody>
</table>

For information about the lunch schedule and which group you are in, please see page 5.
The Fatigue & Durability Forum is a special parallel track supplementing Wednesday’s technical program. During the forum, SIMULIA will showcase fe-safe, the advanced fatigue analysis technology for Finite Element models in the 3DS Portfolio. fe-safe is used by leading companies in the transportation and mobility, industrial equipment, aerospace, defense, power generation, marine and offshore industries worldwide to determine fatigue life and to optimize designs for durability.

This event will bring together leading fatigue experts, fe-safe users and specialist fatigue engineers and analysts to share knowledge, expertise and methodology. You will see presentations on fatigue and durability technology, how fe-safe is applied in industry to achieve business advantage across a broad range of sectors and the value of using fe-safe as part of the wider SIMULIA product portfolio to ensure that designs are optimized for durability. The forum will have a high level of technical content.

Attendees are welcome to go from the technical tracks to the forum and back again throughout the day.

Who should attend:
- Those who are new to fe-safe and would like to find out more about the software and current fatigue technology and practices
- Those who are already familiar with fatigue and are looking for in-depth analysis and detailed applications
- Current fe-safe users

### BELLEVUE

<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Speaker(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:45</td>
<td>Analysis Led Design</td>
<td>Jeffrey Jones, Cummins Inc.</td>
</tr>
<tr>
<td>11:15</td>
<td>The Importance of Random Vibration Fatigue in Structural Dynamic Applications</td>
<td>Giovanni De Morais, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>11:45</td>
<td>Factors Determining a Material’s Fatigue Properties</td>
<td>Mary Starkey, Consultant, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>12:15</td>
<td>Lunch Group 1 / Technology &amp; Product Updates</td>
<td></td>
</tr>
<tr>
<td>13:15</td>
<td>Lunch Group 2 / Technology &amp; Product Updates</td>
<td></td>
</tr>
<tr>
<td>14:15</td>
<td>Consideration of Manufacturing Pre-History Influence of Fatigue Life of Sheet Metal Formed Automotive Parts</td>
<td>Dmitry Buzlaev, TESIS</td>
</tr>
<tr>
<td>14:45</td>
<td>Simplified Weld Modeling with the Verity Structural Stress Method</td>
<td>Dr. Pawel Sobczak, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>15:15</td>
<td>BREAK &amp; PARTNER EXHIBITION</td>
<td></td>
</tr>
<tr>
<td>15:45</td>
<td>Development of Relative Analysis Methods to Determine Fatigue Durability in Aluminum Cylinders</td>
<td>Anthony Megel, Southwest Research Institute</td>
</tr>
<tr>
<td>16:15</td>
<td>Fatigue of Plastics. Past, Present, Future</td>
<td>Dr. Anders Winkler, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>16:45</td>
<td>GENERAL LECTURE 2, Potsdam I/III</td>
<td></td>
</tr>
</tbody>
</table>
### AGENDA – THURSDAY, MAY 21

See page 66 for the conference floor plan

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:15</td>
<td>REGISTRATION &amp; EARLY BIRD HANDS-ON, SCHÖENBERG II</td>
</tr>
<tr>
<td>9:15</td>
<td>GENERAL LECTURE 3: Technology, Functional and Usage Updates in the Power of the Portfolio for 2015, SIMULIA, Potsdam I/III</td>
</tr>
<tr>
<td>9:55</td>
<td>SPECIAL GUEST SPEAKER: Advanced Finite Element Analysis – From Expert Analysis Tool to Realistic Simulation Platform Frans Peeters, Vice President, SIMULIA EMEA</td>
</tr>
<tr>
<td>10:15</td>
<td>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</td>
</tr>
</tbody>
</table>

### 10:45

**OPPORTUNITIES FORUM IN BELLEVUE**

<table>
<thead>
<tr>
<th>TEGEL</th>
<th>CHARLOTTENBURG III</th>
<th>CHARLOTTENBURG II</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:45</td>
<td>Development of the Automated Weight Optimization Process for Automotive Wheel Bearing Jungho An ILjin Global Corporation</td>
<td></td>
</tr>
<tr>
<td>10:45</td>
<td>Simulation Lifecycle Management in the Real World Andres Belies Meseguer Prime Reostructures</td>
<td></td>
</tr>
<tr>
<td>11:15</td>
<td>Structural Analysis of an Automotive Forming Tool for Large Presses within Dynamic FEM Simulation using Abaqus/Standard Krzysztof Swiderski OTH Regensburg</td>
<td></td>
</tr>
<tr>
<td>11:15</td>
<td>Finite Element Simulation of Thermal Barrier Coatings in Rocket Engines Martin Bäker Technische Universität Braunschweig, Institut für Werkstoffe</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Failure Analysis of CFRP Tubes with Integrated Rubber Layers Subjected to Transverse Low-Velocity Impact Loading Enrico Steldinger BTU Cottbus–Senftenberg</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Analysis of the Carbon Fiber Wing Torsion Box After Low Velocity Impact Damage Ivan Zharenov TsAGI - Central Aerohydrodynamic Institute</td>
<td></td>
</tr>
</tbody>
</table>

### INTEGRATED SOLUTIONS

<table>
<thead>
<tr>
<th>11:45</th>
<th>Simpleware and Abaqus: Bridging Image Data and Simulation Theo Verbruggen</th>
</tr>
</thead>
<tbody>
<tr>
<td>11:45</td>
<td>Using Abaqus for Fatigue Analysis of Components with Large Displacements (e.g. rubber) Engineering Center Steyr GmbH &amp; Co KG Axel Werkhausen</td>
</tr>
<tr>
<td></td>
<td>Advanced Automotive Thermal Simulation Process ThermoAnalytics Antti Jussila</td>
</tr>
</tbody>
</table>

### 12:15

**LUNCH GROUP 1 | TECHNOLOGY & PRODUCT UPDATES**

### 13:15

**LUNCH GROUP 2 | TECHNOLOGY & PRODUCT UPDATES**

### 14:15

**OPPORTUNITIES FORUM IN BELLEVUE**

**SIMULIA STAFF TECHNOLOGY APPLICATION PRESENTATIONS (Parallel sessions to be announced at the event)**

### 14:45

**MEET THE EXPERTS: TOPICS - SEE PAGE 9**

### 15:15

**BREAK & PARTNER EXHIBITION IN THE EXHIBIT HALL**

### 15:45

**OPPORTUNITIES FORUM IN BELLEVUE**

### 15:45

**TECHNOLOGY & PRODUCT UPDATES**

### 16:45

**KEY MESSAGES AND ADJOURN**

For information about the lunch schedule and which group you are in, please see page 5.
Thursday’s agenda continues on the next page.
# AGENDA – THURSDAY, MAY 21

See page 66 for the conference floor plan

<table>
<thead>
<tr>
<th>Time</th>
<th>Event Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:15</td>
<td><strong>REGISTRATION &amp; EARLY BIRD HANDS-ON</strong></td>
</tr>
<tr>
<td>9:15</td>
<td><strong>GENERAL LECTURE 3:</strong> Technology, Functional and Usage Updates in the Power of the Portfolio for 2015, SIMULIA, Potsdam I/III</td>
</tr>
<tr>
<td>9:55</td>
<td><strong>SPECIAL GUEST SPEAKER:</strong> Advanced Finite Element Analysis – From Expert Analysis tool to Realistic Simulation Platform Frans Peeters, Vice President, SIMULIA EMEA</td>
</tr>
<tr>
<td>10:15</td>
<td><strong>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</strong></td>
</tr>
<tr>
<td>10:45</td>
<td><strong>OPTIMIZATION FORUM IN BELLEVUE</strong></td>
</tr>
<tr>
<td>10:45</td>
<td><strong>TIERGARTEN II</strong></td>
</tr>
<tr>
<td></td>
<td><strong>INDUSTRIAL EQUIPMENT</strong></td>
</tr>
<tr>
<td>10:45</td>
<td>On the Numerical Implementation of a 3D Fractional Viscoelastic Constitutive Model Gioacchino Riotta, University of Palermo - University of Oxford</td>
</tr>
<tr>
<td></td>
<td>An Easy Procedure for Anisotropic Nonlinear Behavior of Short-fiber-reinforced Plastics Sascha Pazour, PART Engineering GmbH</td>
</tr>
<tr>
<td>11:15</td>
<td><strong>TIERGARTEN I</strong></td>
</tr>
<tr>
<td></td>
<td><strong>ARCHITECTURE, ENGINEERING &amp; CONSTRUCTION</strong></td>
</tr>
<tr>
<td></td>
<td>Modeling of Confined Inflatable Structures Eduardo Sosa, West Virginia University</td>
</tr>
<tr>
<td>11:45</td>
<td><strong>INTEGRATED SOLUTIONS</strong></td>
</tr>
<tr>
<td></td>
<td><strong>ARCHITECTURE, ENGINEERING &amp; CONSTRUCTION</strong></td>
</tr>
<tr>
<td>11:45</td>
<td>Simulating Strongly Coupled Fluid-Structure Interactions Using Abaqus and FlowVision Andrey Aksenov, CAPVIDIA</td>
</tr>
<tr>
<td></td>
<td>MpCCI – Multiphysics Interfaces Klaus Wolf, Fraunhofer Institute SCAI</td>
</tr>
<tr>
<td>12:15</td>
<td><strong>INTEGRATED SOLUTIONS</strong></td>
</tr>
<tr>
<td>12:15</td>
<td>LUNCH GROUP 1</td>
</tr>
<tr>
<td>13:15</td>
<td>LUNCH GROUP 2</td>
</tr>
<tr>
<td>14:15</td>
<td><strong>OPTIMIZATION FORUM IN BELLEVUE</strong></td>
</tr>
<tr>
<td>14:15</td>
<td><strong>SIMULIA STAFF TECHNOLOGY APPLICATION PRESENTATIONS</strong> (Parallel sessions to be announced at the event)</td>
</tr>
<tr>
<td>14:45</td>
<td><strong>MEET THE EXPERTS: TOPICS</strong> – SEE PAGE 9</td>
</tr>
<tr>
<td>15:15</td>
<td><strong>BREAK &amp; PARTNER EXHIBITION IN THE EXHIBIT HALL</strong></td>
</tr>
<tr>
<td>15:45</td>
<td><strong>OPTIMIZATION FORUM IN BELLEVUE</strong></td>
</tr>
<tr>
<td>15:45</td>
<td><strong>TECHNOLOGY &amp; PRODUCT UPDATES</strong></td>
</tr>
<tr>
<td>16:45</td>
<td><strong>KEY MESSAGES AND ADJOURN</strong></td>
</tr>
</tbody>
</table>

For information about the lunch schedule and which group you are in, please see page 5.
The Optimization Forum is a special parallel track supplementing Thursday’s technical program. During the Forum, SIMULIA will showcase Isight, Tosca and optimization enabled 3DEXPERIENCE Apps in the 3DS Portfolio. This event will bring together leading optimization experts, Isight & Tosca users as well as industry engineers to share knowledge, expertise and best practices. There will be presentations from industrial companies such as Audi and BMW, as well as academia. We will show how our new software reduces the time to author robust simulation models that can be optimized to create real-world break-through products. Attendees are welcome to go from the technical tracks to the forum and back again throughout the day.

**Who should attend:**
- Designers and Analysts who have an interest in applying optimization
- Experts in optimization who are looking for in-depth analysis and detailed applications

### BELLEVUE

<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:45</td>
<td><strong>Benefits of Optimization Driven Product Development</strong></td>
</tr>
<tr>
<td></td>
<td>Steven Ribeiro-Ayeh, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>11:15</td>
<td><strong>Mixed-Integer Nonlinear Optimization under Industrial Side Conditions</strong></td>
</tr>
<tr>
<td></td>
<td>Prof. Schittkowski, University Bayreuth, Satoshi Miyata, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>11:45</td>
<td><strong>INTEGRATED SOLUTIONS &amp; BRAND PRESENTATIONS</strong></td>
</tr>
<tr>
<td>12:15</td>
<td>Lunch Group 1 / Technology &amp; Product Updates</td>
</tr>
<tr>
<td>13:15</td>
<td>Lunch Group 2 / Technology &amp; Product Updates</td>
</tr>
<tr>
<td>14:15</td>
<td><strong>Thermomechanical Optimization of High-load Turbochargers with Tosca Structure</strong></td>
</tr>
<tr>
<td></td>
<td>Ekkehard Rieder, AUDI, Michael Werner, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>14:45</td>
<td><strong>Multi-disciplinary Optimization (MDO), RPC and Advanced CAE</strong></td>
</tr>
<tr>
<td></td>
<td>Dr Tayeb Zeguer, Jaguar Land Rover, Limited, Nikolai Baumeister, DS CATIA</td>
</tr>
<tr>
<td>15:15</td>
<td>Break &amp; Partner Exhibition</td>
</tr>
<tr>
<td>15:45</td>
<td><strong>The next step: end-to-end optimization enabled industrial workflows</strong></td>
</tr>
<tr>
<td></td>
<td>Alex Van der Velden and Claus Pedersen, R&amp;D, Dassault Systèmes SIMULIA</td>
</tr>
<tr>
<td>16:45</td>
<td><strong>Key Messages and Adjourn</strong></td>
</tr>
</tbody>
</table>
The 2015 Academic Poster Showcase is truly international with 26 posters from around the globe representing a wide variety of research and teaching endeavors. You are welcome to stop by our display in the Potsdam Foyer any time, but please make a note to join us during the afternoon break, Tuesday, May 19th, when we will hold our poster author “Meet and Greet”. While you’re there, please VOTE for your “favorite” poster and pick up a useful souvenir. The poster chosen by attendees will receive a special mention from the main stage and the submitter will receive an iPad Air!

Don’t forget to stop by the SIMULIA booth to meet representatives from our academic team and learn more about our new research and teaching portfolio products.

**Aerospace & Defense**
1. **Glyndwr University, UK**
   Alison McMillan, David Walker, Paul Rees, Marian Mielke, Robert Jones
2. **Texas A&M, USA**
   Darren Hartl, Björn Kiefer, Robin Schulte, Edwin Peraza Hernandez, Andreas Menzel
3. **Texas A&M University, USA**
   Austin Cox, Theocharis Baxevanis, Dimitris Lagoudas

**Architecture, Engineering & Construction**
4. **Ecole Nationale Superieure des Mines de Douai, France**
   M. Cassama, J. Kalfa, D. Luez, T. Strauss, S. Yixun
   Advisers: M. Lagardere, P. Hulot, D. Vasiukov
5. **Leipzig University, Germany**
   Holger Pankrath, Alexander Knut, Marco Barthel, Ralf Thiele
6. **Universidad Nacional Autónoma de México, México**
   Alan R. Saucedo, Ramírez D. Edgar I., Ruiz C. Osvaldo, Ortiz P. Armando

**Consumer Packaged Goods & Retail**
7. **RWTH Aachen University, Germany**
   D. Trauth, F. Klocke, R. Shirobokov, P. Mattfeld

**Education**
8. **Penn State, USA**
   Ivi Smid

**Energy, Process & Utilities**
9. **Technische Universität Braunschweig, Germany**
   Martin Baker, Aviral Shrot
10. **University of Alberta, Canada**
    Yi Zhang

**Industrial Equipment**
11. **Brandenburg University of Technology Cottbus-Senftenberg, Germany**
    Michael Lockan, Ivo Martin, Lennard Hartwig, Dieter Bestle
12. **KU Leuven, Belgium**
    Philip Eyckens

**Life Sciences**
13. **Anglia Ruskin University**
    Jennifer Boyd, Franziska Reisse, Sophie Bruneau, Xiang Chen, Josh Baxter, Paul Kim, Jonathan Deland,
14. **Ritsumeikan University, Japan**
    Zhongkui Wang, Shinichi Hirai
15. **Texas A&M, USA**
    Edwin Peraza Hernandez, Darren Hartl, Dimitris Lagoudas, Richard Malak, Ergun Akleman
16. **University of Alberta, Canada**
    Tanvir Mustafiy, Marwan El-Rich, Kodjo Moglo
17. **University of Alberta, Canada**
    Sadegh Naserkhaki, Tanvir Mustafiy, Marwan El-Rich, Kodjo Moglo
18. **University of Leeds, UK**
    Marlène Mengoni
19. **University of Oxford, UK**
    Jennifer Boyd, Cameron Brown, Andrew Price
20. **University of Oxford, UK**
    E.C Pegg, CRF Dodd, DW Murray, HG Pandit

**Marine & Offshore**
21. **The University of Western Australia, Dalian University of Technology, China**
    Pan Hu, Dong Wang, Mark Cassidy, Qing Yang

**Natural Resources**
22. **Leibniz Universität Hannover, Germany**
    Stefanie Zeumann, Andrea Hampel
23. **Leibniz Universität Hannover, Germany**
    Meike Bagge, Andrea Hampel
24. **University of Potsdam, Germany**
    Oliver Heidbach
25. **University of Potsdam, Germany**
    Moritz Ziegler, Karsten Reiter, Oliver Heidbach, Arno Zang

**Transportation & Mobility**
26. **Sheffield Hallam University, UK**
    Daniel Tunnicliff
SIMULIA COMMUNITY CONFERENCE ABSTRACTS

This program contains the abstracts of the papers submitted for presentation at the SIMULIA Community Conference held in Berlin, Germany, May 18-21, 2015. The abstracts are categorized by industry, followed by company name in alphabetical order. We are grateful to the authors for their efforts in preparing the papers that make up these proceedings.

The full manuscripts can be found on the SIMULIA Learning Community at www.3ds.com/slC.

AEROSPACE & DEFENSE

Numerical Analysis to Optimize Peen-Forming Process Parameters
Yasser Essa and Federico Martin de la Escalera (Aernnova Engineering Division S.A.)

Evaluation of Abaqus XFEM Capabilities for Crack Growth Analysis in Aeronautical Structures
Ismael Rivero Arevalo and Javier Gomez-Escalonilla Martin (Airbus Defence and Space)

Bird Strike Analysis for Impact-resistant Design of Aircraft Wing Krueger Flap
Sebastian Heimbs, Wolfgang Machunze and Gerrit Brand (Airbus Group)
Bernhard Schlipf (Airbus Operations GmbH)

Use of Abaqus Explicit for Composite Sandwich Impact Damage Prediction during Bird Impact
Marwan Al-Khalil (Airbus Operations Ltd.)
Richard Rigby and Erkan Kiritil (Airbus)

Design Against Distortion in the Aerospace Industry
Sjoerd van der Veen (Airbus Operations SAS)

Time Accurate Simulation of Aeroelastic Flap Deployment
Nicolas Reveles and George Antoun (ATA Engineering, Inc.)
James Fort (Dassault Systems SIMULIA Corp.)

Failure Analysis of CFRP Tubes with Integrated Rubber Layers Subjected to Transverse Low-velocity Impact Loading
Enrico Stelldinger, Arnold Kühhorn and Markus Kober (BTU Cottbus–Senftenberg)

Pressure Vessel New Design Subjected to Blast Loads
Laurence Cousin and Pierre Evrard (CER/DRM/DIF)

Predicting In- and Out-of-plane Damage Evolution in Fiber-reinforced Composites
Wouter Wilson (Fokker Landing Gear B.V.)

Nested Micromechanical and Structural Models for the Analysis of Discontinuous Long-fiber Thermoplastic Composite Materials and Structures
Hakan Kilic (Greene, Tweed & Co.)

An Approach Towards an Optimal Design of Composite Structures
Barbara Goller, Manuel Prackwieser and Manfred Gratt (INTALES GmbH)

Methodology for the Analysis of Tolerances in the Assembly Process of a Wing Torsion Box Through FE Simulations
Agustin Chiminelli, Ruban Breto Pontaque and Miguel Angel Jimenez (ITRINNOVAR)
Federico Martin de la Escalera and Luis Lizarduy (AERNOVAR)

Simulation of High Velocity Perforation using Various Plasticity and Fracture Models
Vladislav V Vershinin (Moscow State University of Civil Engineering)

Simulation Lifecycle Management in the Real World
Andres Belles Meseguer and Roland Zeillinger (Prime Aerostructures)

Modeling of Conduction Heat Transfer Across Bolted Joints in Abaqus
Volkan Coskun, Emir Ozkokdemir and Bulent Rcar (Roketsan Missiles Inc.)

Modeling of Forming Process of Composite Materials Based on Thermoplastic Matrix
Boris Fedulov, Mark Kantor, Alexander Safonov and Andrey Ushakov (Skolkovo Institute of Science and Technology)

Finite Element Simulation of Thermal Barrier Coatings in Rocket Engines
Martin Bäker, Torben Fiedler and Joachim Rösler (Technische Universität Braunschweig, Institut für Werkstoffe)

Residual Strength of the Carbon Fiber Panel with Delamination
Mikhail Pavlov and Ivan Zharenov (TsAGI - Central Aerohydrodynamic Institute)

Shrinkage Modeling of Thermoplastic Wing Rib
Nikita Rozin and Ivan Zharenov (TsAGI - Central Aerohydrodynamic Institute)
Alexander Safonov and Andrey Ushakov (Skolkovo Institute of Science and Technology)
Analysis of the Carbon Fiber Wing Torsion Box After Low Velocity Impact Damage
Ivan Zharenov and Mikhail Pavlov (TsAGI)

Enabling Multi-Fidelity Design Optimization Approaches in Isight
Timoleon Kipouros, Tiziano Ghisu and Jerome Jarrett (University of Cambridge)

A Simulation Tool for Fatigue Analysis and Lifecycle Prediction
Daniel Kujavski, Phani C. R. Sree, Deepak Abburi and Joshua Kuok (Western Michigan University)

ARCHITECTURE, ENGINEERING & CONSTRUCTION

Tangential Plasticity Effect on Buckling Behavior of a Thin Wall Pier under Cyclic Loading Conditions
Hideto Momii (ESTECH Corporation)

Study of Mechanical Compatibility at the Mortar-block Interface in a Heritage Building using the Abaqus Program and Experiment Model
Hicham Fihri Fassi (FSTS)

Improvement and Enhancement of Concrete Damage Plasticity Model
Yuji Nikaido (Kajima Corporation)

An Easy Procedure for Anisotropic Non-Linear Behavior of Short-Fiber-Reinforced Plastics
Sascha Pazour (PART Engineering GmbH)

Experience and Lessons Learnt from the SMART 2013 Benchmark
Maria J. Crespo, Javier Rodriguez, Luis Lacoma, Francisco Martinez and Joaquin Marti (Principia)

Finite Element Analysis of Deep Excavations using Abaqus
Yuepeng Dong (Singapore-MIT Alliance for Research and Technology)

Modeling of Confined Inflatable Structures
Eduardo Sosa (West Virginia University)

CONSUMER PACKAGED GOODS & RETAIL

Simplified Approaches for Modelling a Closed and Partly Fluid Filled Container
Oswald Valtiner, Daniel Pielmeier, and Claudius Oblasser (ALPLA Werke Alwin Lehner GmbH & Co KG)

Modeling the Mechanical Behaviour of Some Complex Polymeric Materials by Relying on the Nonlinear Capability in Abaqus
Colmar Wocke (Dow Chemical)

Analysis of Material-Related Instability Problems for Inelastic Thin Shells
Takaya Kobayashi and Yasuko Mihara (Mechanical Design & Analysis Corporation)

Application of Abaqus/Isight for Paper-based Packaging
Arnoud Dekker (Smurfit Kappa)

Use of Abaqus and True-Load to Determine Mountain Bicycle Loading Environment
Jay Maas (Trek Bicycle Corporation)

ENERGY, PROCESS & UTILITIES

Process integration between CAD and CAE applications for Isight based DOE analysis
Davide Pinna (Ransol Energia)

Hydraulic Fracturing Simulation for Fracture Networks
Stephan Arndt, Tobias Hoeink and Wouter van der Zee (Baker Hughes)

Fracture Stability Assessment during SAGD operations using 4D Geomechanical Models
Wouter van der Zee and Marc Holland (Baker Hughes)

Modeling of Cement Sheath for Thermal and HPHT Wells
Jueren Xie (C-FER Technologies)

Use of FEA and Radial Basis Functions for Reliability-based Design and Assessment of Connection Sealability
Jueren Xie, Thomas Dessein and Feng Rong (C-FER Technologies)

Simulation of Hydraulic Fracturing of Unconsolidated Sands using Fully Coupled Poro-Elastoplastic Models
Saurabh Puri, Juan Hurtado and Deepak Datye (Dassault Systèmes Simulia Corp)

Finite Element Validation of Non-homogenous Stress Fields during Composite Material Testing
Lars Pilgaard Mikkelsen (DTU Wind Energy)

Advanced Fracture Modeling for Cuttings Re-injection
Jorge Garzon (ExxonMobil)
Experimental Validation of Simulation Capabilities for Hydraulic-driven Fracture Propagating in Porous Medium
Jing Ning, Gilbert Kao, Nikolay Kostov and Kevin Searles (ExxonMobil Upstream Research Company)

Dynamic Hydraulic Fracture Modeling for Wellbore Integrity Prediction in a Porous Medium
Nikolay Kostov, Jing Ning and Pablo Sanz (ExxonMobil URC)
Shekhar Gosavi (ExxonMobil DC)
Piyush Gupta (University of Illinois Urbana Champaign)

A Numerical Investigation on the Bedding Resistance of Laterally Displaced Pipelines
Tim Geriach, Martin Achmus and Mandy Narten (Institute for Geotechnical Engineering, Leibniz University Hanover)

Abaqus Analysis Methods on Highly Restrained Pipeline with Soil Berm Formation
Shulong Liu and Emil Maschner (J P Kenny)
Teng Zhang and John Smyth (MCS Kenny)

Pipe Flexibility Analysis and The Development of PCS - Pipe Calculation System for Abaqus
Carlos Eduardo Alvarez de Castro e Sousa (RMC Consultoria)

Optimising Wind Turbine Bearing Performance Using Flexible Bearing Ring Analysis
Jose Torres (Romax Technology)

Design Optimization of a Curved Fir-Tree Root of Steam Turbine Using Computer Aided Optimization and Finite Element Analysis
Shang Jiong Tan, Wei Lin Shu, Gong Yi Wang (Shanghai Turbine Works Company Ltd)

Increasing the efficiency of offshore rigid pipeline lateral buckling assessments using a dedicated GUI and Isight
Carlos Charnaux, Graeme Roberts and Samuel Paul (Subsea 7)

Finite Element Simulation of Rotary Steerable Drilling Systems Advance Understanding of Damaging Downhole Shock and Vibration
Nader Abedrabbo and Liam Lines (Weatherford)

HIGH-TECH

Achieving Higher Productivity on Abaqus Simulations with HPC and Clustering Technologies
Pak Lui (HPC Advisory Council)

INDUSTRIAL EQUIPMENT

Robust and Optimal Strain Gauge Positions for Blade Vibration Testing
Piyawan Teufel (ABB Turbo Systems AG)
Jan Poland (ABB Corporate Research Center)

Prediction of Tin-Whiskers Generation Using Stress and Mass-Diffusion Analysis
Takeshi Terasaki (Hitachi, Ltd.)

Strength and Fatigue Analysis of Leaf Springs used in Heavy Duty Trucks
Mehmet Bakir, Murat Siklas, Serter Atamer (Mercedes-Benz Turk A.S.)

Coupled Analysis of Fracture Mechanics and Piezoelectricity in Active Layers in the Abaqus Code Operated Through the Isight Tool
Eugenio Brusa (Politecnico Di Torino - Dipartimento Di Ingegneria Meccanica e Aerospaziale)
Mehdi Mohammadzadeh Sari (Politecnico Di Torino)
Cosima Fiaschi, Mauro Parodi (EXEMPLAR)

XFEM Simulation of Radial-Median Crack Evolution in Knoop Indentation of Brittle Materials
Felix Rickhey (Sogang University)
Jin Haeng Lee (Korea Atomic Energy Research Institute)
Hyungyil Lee (Sogang University)

Evaluation of the Fatigue Strength of Threaded Connections using Local Concepts
Dominic Olveda (Technische Universität Darmstadt)
Ralf Waterkotte (Schaeffler Technologies GmbH & Co. KG)
Matthias Oechsner (TU Darmstadt)

Strip Shape Prediction using a Simplified Static Model of a 4-High Cold Rolling Process
Mario Alberto Bello-Gomez, Martha P. Guerrero-Mata, Luis A. Leduc-Lezama, Tania P. Berber-Solano (UANL)
Fernando Gonzalez (Ternium Mexico)
Validation of Flow Simulation on Abaqus/CEL™
Edgar Ramirez (Universidad Nacional Autónoma de México (UNAM))
Alan Saucedo, Osvaldo Ruiz, Victo Jacobo and Armando Ortiz (Engineering School, UNAM)

On the Numerical Implementation of a 3D Fractional Viscoelastic Constitutive Model
Gioacchino Alotta (University of Palermo - University of Oxford)
Olga Barrera, Alan C. F. Cocks (University of Oxford)
Mario Di Paola (University of Palermo)

Fractional Viscoelasticity, Constitutive Model, Abaqus/Standard, Creep, Relaxation Modeling of Hydrogen Embrittlement Mechanism in Metals
Olga Barrera, Alan C. F. Cocks and Edmund Tarleton (University of Oxford)

LIFE SCIENCES
Developing a Constitutive Model for a Highly Deformable Latex Rubber with Implementations in Lagrangian Frameworks
Christopher Basciano, Marcus Rademacher, Patrick Downie and Anita Bestelmeyer (BD)
Brian Croop (DatapointLabs)
Jorgen Bergstrom (Veryst Engineering LLC)

Developing a FE Head Model for Impact Simulation in Abaqus
Philippe Young (Simpleware)
Siddiq Qidwai, Alan Leung, Amit Bagchi and Nithyanan and Kota (Naval Research Laboratory)

Drop Test of a Hearing Aid
Morten Birkmose (Soendergaard GN Resound A/S)

Modeling of a Flatfoot using Abaqus for Improving Surgical Treatment
Zhongkui Wang and Shinichi Hirai (Ritsumeikan University)
Kan Imai, Masamitsu Kido and Kazuya Ikoma (Kyoto Prefectural University of Medicine)

TRANSPORTATION & MOBILITY
Efficient Evaluation of Material Cards for Non-reinforced and Reinforced Thermoplastics
Peter Reithofer (4a)

IDIADA’s Virtual Proving Ground (VPG) for Durability Analysis
Jordi Arbiol, Xavier Armengol, Jose Antonio Munoz and Ines Lama (APPLUS IDIADA)

Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach
Vinod Kumar, Vinay Somashekhar and Srivathsa Jagalur (Autoliv India Pvt. Ltd.)

Development of a Separator Plate for Valve Bodies in a Vehicle Automatic Transmission
Harald Rebien, Tobias Tauscher and Rohit Ramkumar (DANAR)

System Modeling and Simulation at Dana Holding
Frank Popielas (Dana Holding Corp)

Modeling the Draping of NCF Composite Preforms
Muhammad-Ali Siddiqui and Colmar Wocke (Dow Europe GmbH)

New Constitutive Model for Woven Thermoplastic Composite Materials
Pieter Volgers, Helga Kuhlmann and Zhenyu Zhang (DuPont Engineering Technologies)

Durability Assessment of Heavy Duty Diesel Engine Cylinder Head
Ahmet Hamdi Guzel (Ford Otosan)

Prediction of Compressive Characteristic of Hat Section High-strength Steel Frame Taking Bauschinger Effect into Account
Kenji Takada (Honda R&D)

The Effect of Rubber in Strength Analysis for Insulator Assembly
Hoo-Gwang Lee, Ji-Hoon Kim and Sung-Jin Yoon (Hyundai Motor Company)

Study on Body Rigidity, Strength and Fatigue Performance for Carbon Fiber Reinforced Plastics Vehicles
Oh Se-Hee and Son Yu-Na (Hyundai Motor Company)

Parametric study of wear of variable displacement vane pumps using finite element analysis
Ju Hwan Lee, Seoung Joon Kim, Tae Gyun Shin, Jeong Soo Lim and Dong Il (Hyundai Motor Company)
Simulation of Pipe Bending Process with Abaqus
Satoshi Ishikawa (IDAJ Co., Ltd.)
Yoshihiro Ishikawa (SANGO Co., Ltd.)

Development of the Automated Weight Optimization Process for Automotive Wheel Bearing
Jungho An, Inha Lee, Seungpyo Lee (ILJIN Global Corporation)
Youngmin Cho and Sangkyu Park (Dassault Systèmes SIMULIA)

Automated Analysis in Abaqus for CATIA V5
Stephen Wallis (Jaguar Land Rover)

Simulation of Fully Coupled Thermo-Mechanical Effects in a Disc Brake Rotor
Ganesh Karunakar, Julian Oscroft and Richard Taulbut (Jaguar Land Rover Limited)

Simulation of Combined Forming and Injection Molding Processes
Timo Mueller (Karlsruhe Institute of Technology (KIT))

Application of Predictive Engineering Tool (Abaqus) to Determine Optimized Automotive Rubber Door Harness Grommet Design
Praveen Mishra and Dayananda Gowda (Mercedes Benz India R & D Pvt Ltd)

Structural Analysis of an Automotive Forming Tool for Large Presses within Dynamic FEM simulation using Abaqus/Standard
Krzysztof Swidergal and Marcus Wagner (OTH Regensburg)
Christian Lubeseder and Ingo von Wurmb (BMW Group)
Steffen Marburg (Universität der Bundeswehr München)

Strength Assessment of Injection Molded Short-fiber-reinforced Plastic Components
Wolfgang Korte, Marcus Stojeck, Sascha Pazour (PART Engineering)

Simulation Driven Design for Manufacture of a Pressed Composite Automotive Tailgate
Andy Ngai and Mark Arnold (Penso Consulting)

Brake System Model Reduction for Squeal Noise Study
Samuel Nacivet and Olivier Fazio (PSA Peugeot Citroën)
Jean-Jacques Sinou (Ecole Centrale de Lyon)

Application of Coupled Eulerian Lagrangian Approach in Finite Element Simulation of Friction Stir Welding
N. K. Sanjeev and B. P. Ravikiran (Robert Bosch Engineering and Business Solutions Limited)

Innovative Anisotropic Material Modeling Approach for Fiber Reinforced Thermoplastics
Recep Yaldiz (SABIC)
Gertjan Kloosterman (Dassault Systèmes B.V. 3DS/SIMULIA)

Topology Optimization of a Motorcycle Swing Arm Under Service Loads using Abaqus and Tosca
Roberto Saponelli (Sacmi/Protesa)
Massimo Damasio (Exsemplar)

Consideration of Manufacturing Pre-history Influence of Fatigue Life of Sheet Metal Formed Automotive Parts
Dmitry Buzlaev (TESIS)
Viktor Kalaxe and Dmitry Zheleznyakov (GAZ UEC)
Anders Winkler (Dassault Systèmes)

Development of a Multifunctional Truck Cross Member Made of Continuous Fiber Reinforced Plastics
Herwig Kirchberger (TEUFELBERGER Composite Ges.m.b.H.)
Stefan Fischmeister and Helmut Dannbauer (Engineering Center Steyr GmbH & Co KG)

Correlation Study Between FE Analysis and Physical Test about Misuse on Light Commercial Vehicles Bonnet with Abaqus Explicit
Tamer Aydiner and Halil Bilal (Tofaş Türk Otomobil Fabrikası A.Ş.)
AEROSPACE & DEFENSE

Numerical Analysis to Optimize Peen-Forming Process Parameters
Yasser Essa and Federico Martin de la Escalera (Aernnova Engineering Division S.A.)

Peen-forming is a practical technique for forming of non-ferrous metallic aeronautical components. The method performance of this cold-working procedure mainly depends on the efficient selection of process parameters and the method performance. Currently, trial and error procedures, based on operators’ experience are used to determine peen-forming intensity patterns. This expensive and time consuming method could be avoided by the accurate estimation of peen-forming process parameters. The absence of a predictive tool to determine these parameters makes difficult to give a quick response to market for the industrialization of new components. A virtual tool is developed to help designers to evaluate the feasibility of applying the peen-forming technique and process parameters to minimize drastically the number of rejected components in the process setup. Peen-forming intensity is usually quantified by means of surface coverage, pre-stress condition, ball size, shot velocity, component thickness, etc. The virtual software is called AERnnova Numerical Optimization of Peen-Forming Process (AERNOP). The scope of the present study is to determine the process parameters optimization of peen-forming process using a 3D finite element method (FEM) methodology.

Keywords: Virtual tool, flat pattern, residual stresses, optimization, AERNOP

Evaluation of Abaqus XFEM Capabilities for Crack Growth Analysis in Aeronautical Structures
Ismail Rivero Arevalo and Javier Gomez-Escalonilla Martin (Airbus Defence and Space)

In the Fracture Mechanics field, standard industrial methods for crack growth analysis are mainly based on analytical calculations as classical Finite Element approaches are not practical to deal with discontinuities such as fatigue cracks due to the associated high computational costs. XFEM is one of the methodologies that are being developed in the recent years in order to overcome the limitations associated to classical approaches, especially for complex analysis. However available industrial experience is too limited to adopt this new methodology as a standard practice in the aircraft industry. XFEM capabilities available in Abaqus 6.14 have been evaluated by the authors to assess the reliability and feasibility of the implementation of XFEM methodology to perform crack growth analyses in aeronautical structures. A set of different cases, common in aeronautical structures, have been selected for this purpose, from the most simple cases for basic validations to more complex configurations to check Abaqus capabilities on more demanding situations. In the paper authors present the results obtained for the evaluated cases and the validation of these results using alternative methodologies for comparison. Current limitations in Abaqus capabilities are also discussed and future developments are proposed for implementation in next Abaqus versions.

Keywords: Durability & Fatigue

Bird Strike Analysis for Impact-resistant Design of Aircraft Wing Krueger Flap
Sebastian Heimbs, Wolfgang Machunze and Gerrit Brand (Airbus Group)
Bernhard Schlipf (Airbus Operations GmbH)

Bird strike is a severe high velocity impact load case for all forward-facing aircraft components and a major design driver due to the high energies and the strict safety requirements involved. This paper summarizes an experimental and numerical study to design a bird strike-proof lightweight metallic Krueger flap as a high-lift device concept for a laminar wing leading edge of a single aisle short range aircraft. The whole design process was based on numerical optimizations for static load cases in combination with high velocity bird impact simulations with Abaqus/Explicit, with the focus on accurate modelling of the fluid-like bird projectile, the plasticity of the aluminum material and the failure behavior of the structural hinges and fastened joints. Finally, a full-scale Krueger flap prototype was manufactured and tested under bird impact loading, successfully validating the numerical predictions and impact resistance.

Keywords: Bird strike, impact simulation, aircraft Krueger flap, gas gun test

Use of Abaqus Explicit for Composite Sandwich Impact Damage Prediction during Bird Impact
Marwan Al-Khalil (Airbus Operations Ltd.)
Richard Rigby and Erkan Kirtil (Airbus)

A method development program of testing and simulations was carried out to develop bird impact NLFEA capabilities of composite sandwich damage prediction and bird dispersion after penetration of the primary layer using Abaqus/Explicit. Curved composite honeycomb panels representing typical composite wing Fixed Leading Edge (FLE) structure were subjected to bird strike to generate data for method validation. The test campaign was tailored to produce various levels of damage and modes of failure, from minor localized core/skin damage to panel perforation. In all tests high, speed video was installed to capture the behavior of the structure during the impact. In some tests, a fully instrumented witness panel was installed to enable measurement of deformation as a measure of bird residual energy and dispersion after bird penetration of the sandwich panel. FE models were created of the tested structure and analyzed using Abaqus/Explicit and the results were compared with tests. The analysis performed used a standard Lagrangian bird model developed and validated at Airbus over many years. In general, good correlation was obtained between tests and simulations results. It is known that Lagrangian bird models suffer from numerical instability in case of impacting sharp edges due to element distortion. To avoid such numerical difficulties, the SPH capability available in Abaqus/Explicit was used to generate a representative SPH bird model. Test cases were re-analyzed with the SPH bird model and results compared to the Lagrangian approach. A summary of bird impact simulations and correlation against tests is presented in this paper.

Keywords: Aircraft, Wing, Leading, Edge, Bird, Damage, Impact, Composites, Sandwich
Design Against Distortion in the Aerospace Industry
Sjoerd van der Veen (Airbus Operations SAS)
Most manufacturing processes used in the aerospace industry induce residual stress. This is true for thermoset carbon-fiber reinforced composites and for aluminum alloys. The level of stress may be low compared to the strength of the material, but the consequences for part distortion can be great. The main challenge of the aerospace industry today is to increase productivity and reduce recurring cost, while continuing to reduce weight. In this context, future aerostructures must be more integrated and have more optimized designs. Integration leads to larger, more distortion-prone components with locally strict tolerance requirements. Topology-, shape- and composite lay-up optimization can lead to increased sensitivity to process-induced residual stress and can therefore lead to increased risk of part distortion. Abaqus multi-physics non-linear simulations of manufacturing processes allow to predict the sensitivity of new designs to distortion. Examples will be shown for machining of metal, the curing of thermoset carbon-fiber reinforced composites and additive layer manufacturing. In a new European CleanSky2 project “Design Against Distortion”, numerical process simulations will be linked to design optimization techniques in order to design against distortion from the outset.

Keywords: Manufacturing

Time Accurate Simulation of Aeroelastic Flap Deployment
Nicolas Reveles and George Antoun (ATA Engineering, Inc.)
James Fort (Dassault Systems SIMULIA Corp.)
Today, numerous aeroelastic structures are deployed over a finite time period, such as flaps, spoilers, control surfaces, wheel bay and bomb bay doors. These devices must be both divergence and flutter free. Current state-of-art simulations do this using a quasi-steady assumption that does not require the system to physically deploy within the computations. That is, the structure is assumed elastic but does not possess the “large” motion associated with the device’s path of travel. While these simulations have strong technical merit, especially in the case where a structure is inadvertently locked in a partially deployed position, they are unable to capture all of the relevant physics. When a component is deployed over a finite period, the flow physics include additional unsteady aerodynamic effects that are lost without considering the large motion of the device. In addition to the unsteady aerodynamics, the structure has inertial terms that cannot be correctly accounted for with the quasi-steady assumption. It will be demonstrated that by coupling Abaqus/Standard, which simulates the elastic response and flexible multi-body dynamic articulation, to the CFD solver Star-CCM+ via SIMULIA’s Co-Simulation Engine (CSE), a time accurate response to flap deployment can be achieved. This is compared to results from a simulation utilizing the quasi-steady assumption.

Keywords: Aeroelasticity, CFD Coupling, Connectors, Coupled Analysis, Dynamics, Elasticity, Flap, Free Play, Multi-Body Dynamics

Failure Analysis of CFRP Tubes with Integrated Rubber Layers Subjected to Transverse Low-Velocity Impact Loading
Enrico Stelldinger, Arnold Kühhorn and Markus Kober (BTU Cottbus–Senftenberg)
Among the numerous advantages of CFRP laminated structures there are some drawbacks, such as the high sensitivity in terms of impact loading. Particularly low-velocity impacts result in non-visible and barely-visible impact damage respectively. The three-dimensional state of stress in the impact zone leads to matrix cracking and delamination inside the composite laminate and possibly to fibre failure for higher impact energies. It can be very difficult to detect such damage by visual inspection, and additionally, in most cases a significant reduction of strength and stiffness can be expected. To improve the damage tolerance a rubber layer, named KRAIBON®, is integrated in the composite layup of cylindrical carbon/epoxy tubular specimens. Numerous low-velocity impact tests, using different rubber compounds and layups, are carried out. The degree of damage is examined using microsectioning. A reliable prediction of the initiation and propagation of failure, caused by impact loading, requires suitable failure criteria such as the theories of Puck, Cuntze or the LaRC05 criterion. Within this work a theory for initial and progressive failure, based on the Puck failure criterion, is implemented in Abaqus/Explicit using a user material subroutine (VUMAT). The initiation and propagation of interface delamination is modelled using cohesive layer elements. A constitutive hyperelastic material law provided by Abaqus is used to model the constitutive behaviour of the rubber by fitting the hyperelastic constants to experimental test data. The numerical predictions of the local pattern of damage and the impactor reaction force histories, computed by explicit finite element calculations, have been compared with the experimental results.

Keywords: Composites, Delamination, Failure, Impact

Pressure Vessel New Design Subjected to Blast Loads
Laurence Cousin and Pierre Evrard (CER/DRM/DIF)
A new design of containment vessel has been proposed to conduct confined detonation experiments. In order to guarantee the confinement of the detonation products, the risk expressed as a probability of failure has to be quantified. This quantification is done using probabilistic analyses which require data from experiments and simulations in order to be sustained. When a blast stresses a spherical vessel, many different mechanical phenomena appear and have to be studied using adapted models. A blast leads to reflected waves in the structure. A numerical chaining is used to access the vessel dynamic structural response. This chaining consists in linking Abaqus for CATIA V5 with Abaqus/CAE to realize the analysis models and in linking our hydrodynamic code to Abaqus/Explicit in a weak coupling. High speed hydrodynamic simulations provide transient pressures...
which are used to act on the inner shell of the vessel. Two mechanical phenomena and their associated finite-elements models are of interest. First, we focus on the nominal model used to study the dynamic response of the vessel itself, underlining the most important components (internal furniture, material laws, type of mesh...) to be taken into account. The spherical vessel has five ports. Each of them has a cover bolted to the vessel. Then, we review a model the aim of which is to understand the bolts behavior when the vessel is subjected to high dynamic stresses. This model consists in a first implicit analysis followed by an explicit analysis which continually takes into account the bolts pre-tension loads. Experiments were performed in order to validate this weak coupling. We end up showing the good agreement between experimental and numerical results.

**Keywords**: Bolt Loading, Connectors, Constitutive Model, Coupled Analysis, Dynamics, Experimental Verification, Explosive, Impact, Safety, Vibration.

---

**Predicting In-and Out-of-plane Damage Evolution in Fiber-reinforced Composites**

Wouter Wilson (Fokker Landing Gear B.V.)

In order to successfully design and qualify composite landing gear parts it is essential to be able to reliably predict the mechanical and failure behavior of the composite material. These predictions rely on the availability of accurate test data, but mechanical tests do not always generate the expected material properties. The goal of this study was to check the validity of test results from physical tests by using virtual testing. For this purpose a new material model has been developed. In this material model the fibers and resin are modelled as separate materials with their own specific material and failure behavior. The interaction between the fibers and resin is accounted for using a modified Mori-Tanaka approach. For the resin both the plasticity and damage behavior is included. For fiber failure new damage laws have been developed. Cohesive surfaces are used to model the delamination behavior. The implementation of this material model and cohesive damage laws is done in Abaqus/Standard. Damage evolution simulations using this material model has led to a much better understanding of unexpected results of several coupon and element tests.

**Keywords**: Composites, Constitutive Model, Damage, Delamination, Fabrics, Failure, Landing Gear, User-Defined Material.

---

**Nested Micromechanical and Structural Models for the Analysis of Discontinuous Long-fiber Thermoplastic Composite Materials and Structures**

Hakan Kilic (Greene, Tweed & Co.)

This study presents a three-dimensional (3D) micromechanics-based nonlinear framework for the analysis of discontinuous long fiber (DLF) thermoplastic composite materials and structures. The proposed material modeling framework is a nested micromechanical approach that explicitly recognizes the composite system within the cross-section of a DLF composite member. This model consists of a unit-cell split into two weighted parts representing fiber and matrix dominated responses. The fiber and matrix responses are explicitly recognized in the micromodel. The overall modeling approach is able to predict both the elastic and nonlinear response of the composite material based on the in-situ properties and response of the fiber and matrix constituents. The proposed modeling framework is applied for a DLF composite material system made from AS4/PEEK. Coupon tests are performed to calibrate the in-situ linear and nonlinear properties of the fiber and matrix. The nonlinear calibration of the matrix is done by using the overall axial shear stress-strain response generated from V-notch tests. Experimental data is used to verify the proposed modeling approach. The 3D modeling framework shows good prediction capabilities for the overall effective elastic constants, as well as the nonlinear multi-axial stress-strain response. The multi-scale constitutive framework is implemented in a displacement-based finite element (FE) code, Abaqus, for the analysis of DLF composite structures. Several examples are presented to demonstrate the coupled multi-scale material and structural analysis. The overall structural modeling framework is numerically efficient and accurate when compared with experimental response of DLF structural components.

**Keywords**: Composites, Constitutive Model, Micromechanics, Experimental Verification, Progressive Damage, Thermoplastic, Failure, Discontinuous Long-fiber, Multi-scale.

---

**An Approach Towards an Optimal Design of Composite Structures**

Barbara Goller, Manuel Prackwieser and Manfred Gratt (INTALES GmbH)

Many engineering fields pose the challenging need for structures with low mass targets in order to be competitive with respect to material costs and fuel consumption. At the same time, multiple constraints regarding allowable values such as stresses and strains have to be fulfilled. Mathematically speaking, a constraint optimization problem has to be solved in the high-dimensional space, where the design variables to be optimized are the number of plies, their orientations and also the stacking sequence. The dimensionality of the problem might soon lead to the limit of computational feasibility if classical optimization algorithms are applied. For this reason, a novel approach is proposed where the optimization problem is tackled by a heuristic adaption procedure on element level. During the FE-analysis, the structural requirements like e.g. allowable stresses or strains are checked and the layup of the element is changed by adding plies and aligning the fiber direction such that the mechanical properties can be optimally used. Access to the element routines in the FE-analysis is a requirement for this strategy for which reason a number of state-of-the-art element routines have been developed and adopted as user element routines. The main advantage of this approach is the applicability to large and complex FE-models, where also several million DOFs do not lead to algorithmic limitations. In order to handle the large amount of data involved in the optimization process, a SQLITE-database is used, which acts as a communication medium between the algorithm and the FE-model.

**Keywords**: Aircraft, Composites, Design Optimization, Minimum-Weight Structures, Optimization, Output Database, Post-processing, Shell Structures.
**Methodology for the Analysis of Tolerances in the Assembly Process of a Wing Torsion Box Through FE Simulations**

Agustín Chiminelli, Ruban Breto Pontaque and Miguel Angel Jimenez (ITAINNOVA)  
Federico Martín de la Escalera and Luis Lizarduy (AERNOVA)

This work presents a methodology for the analysis of deviations that might be produced during the assembly process of aircraft components using the finite element method as calculation tool. The methodology allows determining the effect that different deviations in the constituent parts may have on the final tolerances of an assembly considering the influence of the joining techniques and the different operations that are usually carried out during such processes. This permits evaluation if certain deviations in some parts would maintain the final assembly within the specified tolerances or, conversely, may accumulate/propagate negatively generating a non conformity. In addition, the simulations performed allow analyzing the possible effect of deviations coming from the assembly tooling or different fastening sequences. In this sense, the work performed demonstrates that the finite element method can be used effectively to improve the processes commonly used in the manufacturing of components constituted by several parts and with tight tolerances. This work is part of the activities performed within the project “284961 CLEAN SKY, Simulation of the assembly tolerances for composite aircraft structures – SATCAS”, led by ITAINNOVA and performed under the supervision of Aernnova Engineering Division as Topic Manager. SATCAS is a sub-project associated to the activity of Assembly Simulation defined in the work-package WP 3.2.1.2 of BLADE – SFWR - Clean Sky program, led by Airbus.

**Keywords**: Simulation Methodology, Assembly Processes, Tolerance Analysis, Deviations, Aircraft Structures.

**Simulation of High Velocity Perforation using Various Plasticity and Fracture Models**

Vladislav V. Vershinin (Moscow State University of Civil Engineering)

To evaluate the validity of several plasticity and fracture models high velocity perforation of circular plates by hard steel spheres is considered. As a benchmark target material 2024-T3(51) aluminum alloy is chosen. Experimental data on ballistic limit velocity and residual velocity are used as a reference solution of the considering problem. Models calibration is performed utilizing comprehensive test results available in literature. The sensitivity of major qualitative and quantitative characteristics to plasticity and fracture models elaboration (presence of pressure and Lode dependence), values of parameters of these models, and problem formulation (axisymmetric or 3-dimensional) is assessed independently via simulation of the ballistic limit velocity tests. Then plasticity and fracture models, which give the most accurate results, are used in simulation of the residual velocity tests, that also confirms the validity of these models.

**Keywords**: Plasticity model, fracture criterion, plate perforation, numerical simulation.

**Simulation Lifecycle Management in the Real World**

Andres Belles Meseguer and Roland Zeillinger (Prime Aerostructures)

Simulation has become a key factor in the development of competitive and innovative products. However, the implementation of simulation in an efficient way goes further than choosing the most powerful analysis tools or working with the best engineers: Managing simulation data, the usage of non-standard methods or communication problems, are some of the issues that need to be solved. In Prime, the implementation of the Simulation Lifecycle Management System from SIMULIA has given numerous benefits. Sometimes it is difficult to see the necessity of SIMULIA SLM if not from a practical application. Therefore the objective of this discussion is to show through some of our projects from the aeronautical and automotive industries how SIMULIA SLM is being used and how it has improved our simulation processes. During the development of airplane components with composite collaborative work between design and simulation is needed. This is accomplished by the composites modeling tools from CATIA and Abaqus, which offer bi-directional data transfer links between design and analysis. SLM guarantees an easy up-to-date management of these data, including geometry revisions, laminate definitions or testing results among others, with total traceability. We also present our success analyzing sports car exhaust systems, where different analyses such as a heat transfer, a thermo-mechanical or a dynamic simulation are performed for every new project. Scenario definition tools help us standardizing the methods, giving the engineer the steps to follow. In this way it is easier to follow the state of a project and accessing to its data.

**Keywords**: Simulation Lifecycle Management, SLM, FEM, Composites.

**Modeling of Conduction Heat Transfer Across Bolted Joints in Abaqus**

Volkan Coskun, Emir Ozkokdemir and Bulent Rcar (Roketsan Missiles Inc.)

Bolted joints are frequently used connections in many engineering systems, and crucial parts of the heat transfer path in structures. During the design phase, heat transfer across the joints must be modeled properly in order to determine accurate temperature distribution. Usually in finite element models, perfect thermal contact between mating surfaces is assumed. However, this approach is inadequate and may bring on misleading results owing to the equality of temperature assumption between surfaces. Complex configuration of the joints and non-uniform contact pressure distribution due to bolt preload make the prediction of transient heat transfer characteristics across the joints arduous. In this study, transient conduction heat transfer through the bolted joints was modeled in Abaqus by defining gap conductance between the abutting surfaces. Several available models relating the surface roughness and contact pressure to contact conductance were utilized to estimate the thermal contact resistances at the joints. Impacts of the bolt preload and resulting interfacial pressure distribution on heat transfer and temperature distribution were investigated. Experiments were performed to verify the analyses results by making various temperature measurements.

**Keywords**: Bolted Joints, Gap Conductance, Thermal Contact Resistance.
Modeling of Forming Process of Composite Materials Based on Thermoplastic Matrix
Boris Fedulov, Mark Kantor, Alexander Safonov and Andrey Ushakov (Skolkovo Institute of Science and Technology)

Composite materials based on thermoplastic matrix became a popular choice as a material for modern structures. Nevertheless, the manufacturing process of this type of materials have many technology parameters, which have to be determined before the first composite part is produced. The study of influence of all parameters on final quality of composite part by means of technological experiments are time and cost consuming. This makes engineers to study the way of modeling of thermoplastic composite forming process. This research is dedicated to modeling of thermoplastic material under different conditions. Several approaches to capture specific for this material features are performed. Method to model the influence of crystallization on mechanical properties of composite material and on final residual stresses is analyzed. The way of estimation of consolidation of thermoplastic composite layers is studied. An approach to model shear nonlinearity in composite prepregs is performed. The analysis of initiation of defects in thermoplastic composites under technological temperature cycle is also performed. All numerical procedures and special subroutines based on Abaqus software are presented. Eventually a complete set of engineering tools using Abaqus software needed to model forming process of thermoplastic composite details is realized.

Keywords: Composite material, Thermoplastic composite material, Thermoplastic material forming, Thermoplastic composite residual stress, Technology process thermoplastic material, Failure due to technology residual stress.

Finite Element Simulation of Thermal Barrier Coatings in Rocket Engines
Martin Bäker, Torben Fiedler and Joachim Rösler (Technische Universität Braunschweig, Institut für Werkstoffe)

Rocket engines need to withstand extreme gas temperatures. To achieve this, the engine is lined with copper and cooled with liquid hydrogen. Nevertheless, creep processes can lead to damage and failure of the component. To avoid this, rocket engines can be protected with thermal barrier coatings. Standard coating systems as used in gas turbines are problematic because of large thermal stresses between the copper substrate and the coating due to a large thermal mismatch. Finite element simulations can be used to select more promising coating materials, by calculating the thermal stresses for different alloy systems that can be used as coatings. In addition, these simulations can also be used to study crack propagation in thermal barrier coatings. For this, the Abaqus software is used together with a user-programmed pre-processor to allow frequent remeshing of the crack front and dynamic crack propagation.

Keywords: Coatings, Thermal Stress, Aerospace, Rocket Engine

Residual Strength of the Carbon Fiber Panel with Delamination
Mikhail Pavlov and Ivan Zharenov (TsAGI - Central Aerohydrodynamic Institute)

Carbon fiber reinforced materials are getting widely spread. But at present time method of the calculation of impact residual strength for carbon fiber parts is poorly developed. Especially this method is underdeveloped in case of the delaminated composite parts. Residual strength is very essential for any project where the composite material parts are used. The possibility to estimate the value of residual strength of the carbon fiber reinforced parts is highlighted in terms of safety for Aerospace structures. In this work we propose a method to calculate the carbon fiber panel residual strength of the civil plane wing prototype with respect to delamination progress simultaneously with degradation law of the material properties. This approach has been developed and realized based on Abaqus software. All numerical models are presented and the dependence of panel residual strength on the delamination size was shown based on these models.

Keywords: Carbon fiber, Delamination, Abaqus

Shrinkage Modeling of Thermoplastic Wing Rib
Nikita Rozin and Ivan Zharenov (TsAGI - Central Aerohydrodynamic Institute)
Alexander Safonov and Andrey Ushakov (Skolkovo Institute of Science and Technology)

During manufacture of parts made of thermoplastic composites in the process of thermoforming liquid of resin transfers to solid-state with occurrence of residual stresses in part which may lead to distortion of the shape of the product. Transition from liquid to solid is called the crystallization process, the reverse process – melting. In order to predict products distortion mathematical modeling techniques are developed. Methods are allowed to evaluate the degree of warpage of the selected design solutions and on the basis of these data to change the process parameters or product design. Behavior model of thermoplastic material has been developed by Skolkovo Institute of Science and Technology and then realized in Abaqus user subroutine UMAT. The paper presents an example of the application of this material model for thermoforming process of the wing rib on the last states of the process – without drooping process. The material of the wing rib is Tenax®-E TPCL PEEK-HTA40. Every ply was modeled by own solid element. The feature of modeling of plies is that one ply may differ in thickness in 2,3 times in construction that’s why they were modeled in Abaqus for CATIA V5. Geometry distortion in modeling has good agreement with the experimental data.

Keywords: Composite Materials, thermoplastics, thermoforming

Analysis of the Carbon Fiber Wing Torsion Box After Low-velocity Impact Damage
Ivan Zharenov and Mikhail Pavlov (TsAGI)

Carbon fiber reinforced materials are getting widely spread. But at present time method of the calculation of impact residual strength for carbon fiber parts is poorly developed. Especially this method is underdeveloped in case of the delaminated composite parts. Residual strength is very essential for any project where the composite material parts are used. The possibility to estimate the value of residual strength of the carbon fiber reinforced parts has a special importance in terms of safety for Aerospace structures. In this work we propose a method for calculating residual strength of the carbon fiber panel of the civil
plane wing prototype with respect to delamination progress simultaneously with degradation law of the material properties. This approach has been developed and realized based on Abaqus software. All numerical models are presented and as a result, the dependence of panel residual strength on the delamination size was shown.

**Keywords**: Composite Materials, residual strength, VCCT

---

**Enabling Multi-Fidelity Design Optimization Approaches in Isight**

**Hideto Momii (ESTECH Corporation)**

In an advanced engineering design environment, such as jet engine design, there are several levels of the design description and process. The engineering paradox is that at the early stages of the design process, crucial decisions are required and certain design parameters have to be defined with very limited knowledge of their impact on the final design. With specific multi-fidelity design optimization approaches we are able to facilitate more informative selections of these parameters at the early stages of the design process. We can achieve this by making high-fidelity information available at the preliminary, or even conceptual stage of the design by means of carefully tailored multi-fidelity design approaches. It is very clear that this has great benefits to the vendors (industrial companies). At the same time, in-house design strategies and methodologies that were developed at academic level to tackle such challenges have to be adapted now in the industrial context. Isight is a framework flexible enough to allow such implementations and developments. The case study presented in this paper is the design of the core compression system of an air-breathing engine. The description of this design problem is expressed with a large design space, which makes the application of high-fidelity analysis within the computational design context particularly challenging. With the presented multi-fidelity design approach we are able to reduce significantly the computational cost, and additionally, achieve a much improved design solution.

**Keywords**: Optimization

---

**A Simulation Tool for Fatigue Analysis and Lifecycle Prediction**

**Daniel Kujawski, Phani C.R. Sree, Deepak Abburi and Joshua Kuok (Western Michigan University)**

This paper presents an interactive web-based simulation tool for a modern fatigue analysis and lifecycle prediction methodology of smooth and notched components. Users around the globe with diverse degree of fatigue familiarity may access it via Internet by means of multiple platforms such as desk- and lap-top computers, tablets and/or smart-phones. In particular, the users with a limited fatigue analysis background would benefit from “on-the-fly” fatigue learning experience. This is accomplished by means of proper guidance through a step-by-step process and providing specific details and explanations without the need of a tutorial handbook. It makes a self-explanatory and frustration free web-based simulation software, which allows users to learn the fatigue fundamentals while expanding their knowledge on modern fatigue analysis methods. For variable amplitude loading a dedicated spectrum analysis tool is provided for a potential clean-up and desire modifications of a raw spectrum data. Subsequently, a rainflow method is used and the corresponding hysteresis loops at the notch-root are determined. Then, the relevant interactive graphs, calculated values and tables are displayed.

**Keywords**: Web-based software, fatigue analysis, notch stresses, Neuber’s rule, spectrum loading

---

**ARCHITECTURE, ENGINEERING & CONSTRUCTION**

**Tangential Plasticity Effect on Buckling Behavior of a Thin Wall Pier under Cyclic Loading Conditions**

**Hideto Momii (ESTECH Corporation)**

As have been mentioned in various experimental studies, low cycle fatigue life depends not only on strain amplitude but also on strain path as well, in fact it changes if proportional or non-proportional cyclic loading conditions are applied. Therefore, it is important to accurately predict the accumulated plastic deformations due to non-proportional cyclic loading. The conventional elastoplasticity models, based upon the premise that the interior of yield surface is a pure elastic domain, have contributed to the predictions of elastoplastic deformation of solid structures such as metals and geomaterials. They, however, cannot describe cyclic loading behavior below the yield stress of materials due to the neat distinction into two separate domains: an elastic and a plastic ones. An additional drawback is the unrealistically stiff response that might be generated in multiaxial loading problems, especially for those where a non-proportional loading path is considered. The main reason for this is the assumption that the inelastic strain is not induced by the stress rate component tangential to the yield surface but it is uniquely directed outward along the normal to the plastic potential surface. In this work, we adopted an unconventional elastic-plastic model, which can consider the plastic deformations within the elastic domain and tangential inelastic strain rate due to the stress rate tangential to the yield surface, to analyze the behavior of material under non-proportional cyclic loading conditions. We discuss the numerical simulation results under cyclic non-proportional loadings.

**Keywords**: Non-proportional Cyclic Loading, Elasto-plastic FE Analysis, Tangential Plasticity, Low Cycle Fatigue Life, Constitutive Equation.

---

**Study of Mechanical Compatibility at the Mortar-block Interface in a Heritage Building using the Abaqus Program and Experiment Model**

**Hicham Fihri Fassi (FSTS)**

This research work aims at modeling the mortar-block assembly for analyzing the potential occurrence of mechanical problems at interfaces as the consequence of an eventual incompatibility. The study considers a heritage building located in Morocco. The authors focus their attention on a representative part of the building and perform numerical simulations using Abaqus program, studying the stress repartition inside the masonry. For carrying out efficient analysis, the mechanical characteristics
(Modulus of Resistance, Modulus of Elasticity) are collected through non-conventional testing techniques dedicated to the field of Heritage Conservation that are briefly described. The scenario considered in the proposed study concerns a bearing wall restored with a non-adequate mortar. The goal of the modeling works is to compare the reparation of internal solicitations between the original situation and the restored one in order to quantify the risk, for materials that should be conserved, associated with the “stress shielding” phenomenon and to justify the particular attention to be paid for the choice of a specific mortar for interventions to be carried out on the concerned building.

Keywords: Masonry, Numerical simulation, Compatibility, Stress shielding, Abaqus

Improvement and Enhancement of Concrete Damage Plasticity Model
Yuji Nikaído (Kajima Corporation)

In order to assess the safety margins of the reinforced concrete structures under a cyclic loading, it is necessary to properly simulate damage process of the concrete; however, concrete damage models in commercial analysis codes are rarely practicable in terms of a transition behavior from tension to compression, and vice versa. In Abaqus, elasto-plastic concrete models have been introduced based on plastic damage theory from V6.3. Though they have enabled elasto-plastic evaluations under a single loading in practical use, it is hard to say that they can simulate an appropriate behavior under a cyclic loading. The purpose of the present paper is to make an improved concrete damage plasticity model that focuses on a cracking behavior. First, we constitute the material model that simulates a process of closing the large tensile cracks and recovering the stiffness. Second, the model is enhanced by adding another feature that simulates a transition from compression to tension again.

Finally, experimental simulation analyses are conducted for validation of efficiency.

Keywords: Material Modeling

An Easy Procedure for Anisotropic Non-Linear Behavior of Short-Fiber-Reinforced Plastics
Sascha Pazour (PART Engineering GmbH)

An easy way to do structural analysis with anisotropic non-linear material properties is shown. For this procedure no user subroutine is required, so there is no limitation in possibilities and speed. It will be shown with industrial examples how the required material can be created, used, and analysed.

Keywords: Composites, Fiber-Reinforced, Injection molding, Process-Structure-Interaction, Anisotropic

Experience and Lessons Learnt from the SMART 2013 Benchmark
Maria J. Crespo, Javier Rodriguez, Luis Lacoma, Francisco Martinez and Joaquin Marti (Principia)

The French CEA, together with EDF and the IREA, recently organised an international benchmark to evaluate the ability to model the mechanical behaviour of a typical nuclear reinforced concrete structure subjected to seismic demands. The participants were provided with descriptions of the structure and the testing campaign; they had to propose the numerical model and the material laws for the concrete (stage #1). A mesh of beam and shell elements was generated; for modelling the concrete a damaged plasticity model was used, but a smeared crack model was also investigated. Some of the initial experimental results, with the mock-up remaining in the elastic range, were provided to the participants for calibrating their models (stage #2).

Predictions had to be produced in terms of eigen-frequencies and motion time histories. The calculated frequencies reproduced reasonably the experimental ones; the time histories, calculated by modal response analysis, also reproduced adequately the observed amplifications. The participants were then expected to predict the structural response under strong ground motions (stage #3), which increased progressively up to a history recorded during the 1994 Northridge earthquake, followed by an aftershock. These results were produced using an explicit solver and a damaged plasticity model for the concrete, although an implicit solver with a smeared crack model was also investigated. The paper presents the conclusions of the pre-test exercise, as well as some observations from additional simulations conducted after the experimental results were made available.

Keywords: Concrete, damage, time history.

Finite Element Analysis of Deep Excavations using Abaqus
Yuepeng Dong (Singapore-MIT Alliance for Research and Technology)
Harvey Burd and Guy Houlsby (University of Oxford)
Andrew Whittle (Massachusetts Institute of Technology)

Deep excavation, widely used for underground constructions, is a very complex soil-structure interaction problem in geotechnical engineering. Its performance is influenced by the sophisticated soil behavior, complex retaining structures, and variable construction sequences. As the excavation goes deeper and larger in scale, challenges arise for both practical applications and research. Finite element analysis is an effective tool to study its mechanism, which can consider both geotechnical and structural aspects such as soil behavior, details of structures, and construction sequences. It provides necessary information on the performance of deep excavations for design purpose, and can also be used to predict the behavior of deep excavation and provide guidance for the construction. This paper shows an example of such detailed analysis of deep excavations using Abaqus. A multiple yield surface plasticity soil model is implemented into Abaqus to represent the small-strain nonlinearity of the soil through UMAT subroutine. A deep excavation case history is investigated through 3D analysis. Important aspects such as (i) thermal effects of concrete, (ii) construction joints in the retaining wall, and (iii) initial ground conditions, are also considered in the analysis. The computed results are correlated with the field measurement during the construction. Results showed that the finite element analysis can capture the performance of the deep excavation satisfactorily.

Keywords: Deep excavations, Finite element analysis, Case histories
Modeling of Confined Inflatable Structures
Eduardo Sosa (West Virginia University)

This work presents the development of Finite Element (FE) models to simulate different stages of the design and operation of constrained inflatable structures. Such structures can be used, for example, for sealing segments of underground civil infrastructure such as large pipes or tunnels. In such applications, the confined inflatable structure is prepared for placement, either permanently or temporally, and left ready for deployment, inflation, and pressurization when needed. Once positioned and in operation, the inflatable may be subject to a number of possible loading scenarios and interaction with the surrounding confining environment, which must be evaluated beforehand. FE models developed for this work include models for folding, positioning, deployment, pressurization, and in-service loading of constrained inflatable structures. Models were created and analyzed with the SIMULIA Abaqus simulation suite. Membrane materials, relaxation techniques, mass scaling, as well as a multi-chamber approach were evaluated and implemented in order to better correlate predicted results to available experimental data. The result is a modeling procedure that can be used to predict the performance of an inflatable structure for a given confining environment. Thus, the models can be used as a predicting tool for optimization of different aspects of the design and field operation of the inflatable structure.

Keywords: Abaqus/CAE, Abaqus/Explicit, Confined Inflatable, Deployment, Folding, Membrane

CONSUMER PACKAGED GOODS & RETAIL

Simplified Approaches for Modelling a Closed and Partly Fluid Filled Container
Oswald Valtiner, Daniel Pielmeier, and Claudius Oblasser (ALPLA Werke Alwin Lehner GmbH & Co KG)

Containers (bottles) made of polymers are exposed to manifold loads during their life cycle. They are open or sealed as well as empty or filled. Depending on the mentioned conditions a container behaves different under loading. This effect can be observed in everyday life. Imagine opening a shaken plastic bottle filled with a carbonated beverage. Abaqus provides simulation methods for completely filled containers with either a pneumatic or a hydraulic fluid. Real containers barely are totally filled with a hydraulic fluid. In general there is always a so called head space filled with gas (pneumatic fluid). This paper presents two methods for modelling the two-phase system (liquid and gas) inside a partly filled container using a fluid cavity instead of meshing the volume inside the container. Therefore a one-phase system equivalent to the two-phase system is described and applied to the model using two different methods – an explicit and an implicit one. These methods are implemented by using subroutines which allow extending the functionality of Abaqus. Applying these approaches allows cost efficient simulations of various loads like excess pressure, vacuum, temperature change, top load, squeezing etc. For sealed, partly filled containers. The paper describes the background and assumptions necessary to define the one-phase system, verifies the model by comparing the FEA results with analytical solutions and shows worked examples.

Keywords: Container, Bottle, Filled Container, Partly Filled Container, Consumer Packaged Goods, Hot Fill, Fluid-Structure Interaction, Fluid Cavity, Multi-Phase Simulation, Equation of State, Ideal Gas, Bulk Modulus, Fluid Pressure Compliance and Fluid Temperature Compliance

Modeling the Mechanical Behaviour of Some Complex Polymeric Materials by Relying on the Nonlinear Capability in Abaqus
Colmar Wocke (Dow Chemical)

Polymeric materials are complex in their mechanical behaviour; inter alia, they exhibit large strain, anisotropic and irreversible response which is often accompanied by stable localized necking behaviour. When developing suitable computational models for such materials, one cannot do this without the underlying support of a FEM program that offers inherently non-linear solution capabilities. Within DOW there has been some internal activity, concerned with the development of such a polymeric material computational model. Excellent agreement has been obtained in modeling simple tension tests of Low density polyethylene (LDPE) and Polyethylene terephthalate (PET) films. In addition to showing the results of these model calibrating efforts, some simulation results from application areas for the developed model are presented. These application areas typically come from within advanced, flexible packaging applications where minimum usage of material has an environmental benefit. The severe deformation and generally-complex behaviour able to be captured by the model highlights how indispensable the core-capabilities of Abaqus were in developing the DOW modeling capability.

Keywords: Abaqus, Polymers, Consumer Packaging, Anisotropic, Necking, Non Linear, Fracture, Environmental, Wastage, Plastic

Analysis of Material-Related Instability Problems for Inelastic Thin Shells
Takaya Kobayashi and Yasuko Mihara (Mechanical Design & Analysis Corporation)

Geometrically nonlinear static problems sometimes involve buckling or collapse behavior, in which the load-displacement response shows a negative stiffness and the structure must release strain energy to remain in equilibrium. If the analysis process traces unstable paths under the global load-displacement response with negative stiffness, the arc-length method is effectively usable. However, if the instability is localized (e.g., surface wrinkling, material instability, or local buckling), there will be a local transfer of strain energy from one part of the model to the neighboring parts, and global solution methods may not work. This class of problems must be solved either dynamically or with the aid of artificial damping. This paper describes the application of the artificial damping technique for the generation of singularities in large-deformed viscoelastic shells, which is expected to be utilized in the blow-forming of plastic containers or the formation of polymer thin films. We performed experimental and analytical investigations of viscoelastic cylindrical shells under axial compression. In general, a change in elastic modulus due to viscoelastic relaxation has a range of ten to the third power or more. The artificial damping method overcomes the material-related instability because the instantaneous stiffness change can be reflected in the global
equilibrium equations with appropriately adjusting the damping force so as to correspond to the states of the local and individual instabilities in the model. The study accomplished a fully automatic and seamless simulation in the deep post-buckling region and showed good agreement with the experimental results.

**Application of Abaqus/Isight for Paper-based Packaging**

Arnoud Dekker (Smurfit Kappa)

The presentation is about the application and impact of Abaqus and Isight simulations for corrugated board and packaging within Smurfit Kappa. Two examples will be shown. The first example is the use of Abaqus and Isight for the development of the prediction formulas, used in our prediction tool PaperToBox. PaperToBox is considered as industry leading. This tool is internally used by 1800 colleagues, resulting in >500,000 calculations per year. Also this tool provides the target values for the newly rolled-out quality measurement system, the “Board Referee”. The second example is the use of Abaqus for the optimization of flute profiles (the shape of the wavy paper in the middle), which led to the development and implementation of the flute profile standard within Smurfit Kappa. Smurfit Kappa is one of the leading producers of paper-based packaging in the world, with around 41,000 employees in approximately 350 production sites across 32 countries and with sales revenue of €7.9 billion in 2013.

**Keywords**: Corrugated Board, Bending strength, Design Of Experiments, Response Surface Creation, Optimization

**Use of Abaqus/CAE and True-LoadTM to Determine External and Internal Loading of a Full Suspension Mountain Bicycle**

J. Maas (Trek Bicycle Company)

Trek Bicycle Corporation has long been at the leading edge in the bicycle industry. Treks bicycles are subjected to the most rigorous testing in the industry and their frames are covered by a lifetime warranty. To maintain the highest level of safety and quality, real world loads need to be properly understood. Trek Factory Racing professional athletes are pushing that understanding, most recently attempting a front flip across a 72 foot canyon. The loads generated for such an event are certainly beyond current understanding and testing protocols.

A process using Abaqus and the True-Load™ plug-in (Wolf Star Technologies) will be presented that effectively converts a full-suspension mountain bicycle into a load transducer. A fully instrumented Trek Aluminum Session 29er mountain bike (strain gauges, accelerometer, shock sensors, GPS, etc.) is used with high speed video to record a high load event. These data, in conjunction with the Abaqus / True-Load process, are used to determine the external loads on the bicycle. A comparison of laboratory testing loads to the external loads is made. The implementation of the Abaqus / True-Load process increases Treks knowledge of loading environments and aligns with Treks history of innovation and reliability.

**Keywords**: Abaqus CAE, True-Load, Bicycle, External Loads, Internal Loads, Experimental Verification

**ENERGY, PROCESS & UTILITIES**

**Process integration between CAD and CAE applications for Isight based DOE analysis**

Davide Pinna (Rinsaldo Energia)

Product development engineers often use simulation to evaluate the performance of a single design - a part or a system - characterized by a specific CAD representation, specific input data and materials properties. Sometimes simple parametric analyses and “what-if?” studies are performed in order to try to improve the design. The use of optimization and Design-Of-Experiments techniques, that allow to efficiently and scientifically explore the whole project space to establish the correlation between the various parameters and to identify optimal solutions, is instead less frequent, especially when complex CAD models and various CAE tools are involved. This paper is focused on CAD and CAE workflow integration for the design of a Gas Turbine blade. The model of the complete process, implemented in SEE-Fiper environment and involving all the calculation tools, is described. The target of the project is the exploration of the Design Space (DOE Analysis) to identify an optimized geometry in order to achieve a blade design that implies simpler machining and that satisfy both minimum creep life and minimum fatigue life criteria.

**Keywords**: CAD, CAE, Collaborative Engineering, Gas Turbine Blade, Gas Turbine Vane, Isight, Multidisciplinary Process, Data and Process Management, Process Automation, Turbomachinary.

**Hydraulic Fracturing Simulation for Fracture Networks**

Stephan Arndt, Tobias Hoeink and Wouter van der Zee (Baker Hughes)

The term ‘Unconventional Reservoirs’ is used in the Oil & Gas industry for hydrocarbon reservoirs that have very low permeability in the magnitude of microdarcy (μd) and therefore rely on artificially introducing pathways for fluids and gases, commonly by using hydraulic fracturing techniques, to enable economic production. This approach of using fracturing stimulation campaigns is also increasingly used to improve production in mature oil and gas fields. Industry estimates for North America indicate that more than half of all stimulation treatments have no impact on production, despite this technique having been in use in numerous onshore wells for more than a decade. Creating fractures connecting the source rocks to the well whilst allowing fluids to flow, either with new hydraulic fractures and/or activating existing natural fracture sets can have a large impact on production. Optimising the current approach based on determining its many parameters and adapting experience based knowledge to other regions in the world poses a complex challenge with the goal to understand and predict the effectiveness of the stimulation campaigns.
Significant progress in the simulation of fluid driven crack propagation for a single fracture occurred with recent updates of Abaqus capabilities including coupled fracture and pore fluid flows with porous medium deformation. This paper looks at the challenges using different modelling approaches using Abaqus such as Cohesive Elements, Extended Finite Element Method (XFEM), Smooth Particle Hydrodynamics (SPH) and coupled Eulerian Lagrangian methods (CEL) with the goal to extend applications to more complex geometries with interaction of multiple fractures and stress shadowing effects.

**Keywords:** Fracture Mechanics, Geomechanics (Oil&Gas)

### Fracture Stability Assessment during SAGD operations using 4D Geomechanical Models

**Wouter van der Zee and Marc Holland (Baker Hughes)**

In the oil and gas industry, steam assisted gravity drainage (SAGD) is an operation where steam is injected into a reservoir to lower the viscosity of heavy oil to allow its production. The injection of the steam leads to thermally induced stresses and pore pressure changes in the reservoir and its proximity. Although the pressure and temperature evolution can be modelled with reservoir simulators, they neglect the change of the stress field in relation to these properties, which can impact the stability of faults and fractures. These active faults and fractures in turn impact on the fluid transport properties. In order to assess the potential for fault and fracture (re-)activation a model of a SAGD operation was created by coupling the results of a reservoir simulation to a 4D geomechanical finite-element model using Abaqus. The stresses calculated in the Abaqus model are used as the input for fracture stability calculations. First, a conservative analysis uses a directionless assessment of fracture stability without the need for discrete fractures. It shows extensive critically-stressed regions above and below the injection well, which extend into the cap rock. A second analysis using the available fault models reveals that the bounding faults have critically stressed patches at the final stage of the stimulation. Quantitative risk assessment is performed to investigate the influence of a range of input parameters on the fault stability and to quantify the risk of fault reactivation. The outcome of the study finds there is need for a detailed analysis of the cap rock properties such as thickness, strength and fracture toughness and the impact of potential fractures.

**Keywords:** Oil and Gas, Geomechanics, Steam Assisted Gravity Drainage, SAGD, Reservoir, Thermal Expansion, Abaqus, JewelSuite, Stress, Displacement, Permeability, Cap Rock, Seal integrity, Faults, Fractures, Stability, Critically Stressed, Reactivation

### Modeling of Cement Sheath for Thermal and HPHT Wells

**Jueren Xie (C-FER Technologies)**

One of the significant challenges in oilwells is to maintain adequate structural and sealing capacities of casing and tubing connections. For example, in HPHT wells, the high pressure (greater than 70 MPa) and high temperature (greater than 177°C) well environments require elevated standards for the sealability of tubular connections. However, the deterministic design approach appears to be overly conservative as it does not take into account the uncertainties related to the variability in tubular connection geometry, material mechanical properties (e.g. yield and tensile strength), and operational loads. Therefore, Reliability-based Design and Assessment (RBDA) is recommended to achieve a balance between operational safety and cost. This paper presents a RBDA methodology that characterizes system safety using a quantitative estimate of reliability. In the RBDA approach, Finite Element Analysis (FEA) using Abaqus is employed to determine the connection sealing capacities as functions of several design parameters, such as connection geometry, material mechanical properties, and make-up and operational conditions. Based on the FEA results of design parameters and variations, an implicit limit state function can be established using Radial Basis Functions (RBF). This paper also presents analysis examples using generic premium connections to demonstrate the use of the proposed methodology.

**Keywords:** Design Optimization

### Use of FEA and Radial Basis Functions for Reliability-based Design and Assessment of Connection Sealability

**Jueren Xie, Thomas Dessein and Feng Rong (C-FER Technologies)**

One of the significant challenges in oilwells is to maintain adequate structural and sealing capacities of casing and tubing connections. For example, in HPHT wells, the high pressure (greater than 70 MPa) and high temperature (greater than 177°C) well environments require elevated standards for the sealability of tubular connections. However, the deterministic design approach appears to be overly conservative as it does not take into account the uncertainties related to the variability in tubular connection geometry, material mechanical properties (e.g. yield and tensile strength), and operational loads. Therefore, Reliability-based Design and Assessment (RBDA) is recommended to achieve a balance between operational safety and cost. This paper presents a RBDA methodology that characterizes system safety using a quantitative estimate of reliability. In the RBDA approach, Finite Element Analysis (FEA) using Abaqus is employed to determine the connection sealing capacities as functions of several design parameters, such as connection geometry, material mechanical properties, and make-up and operational conditions. Based on the FEA results of design parameters and variations, an implicit limit state function can be established using Radial Basis Functions (RBF). This paper also presents analysis examples using generic premium connections to demonstrate the use of the proposed methodology.

**Keywords:** Design Optimization
Finite Element Validation of Non-homogenous Stress Fields during Composite Material Testing

Lars Pilgaard Mikkelsen (DTU Wind Energy)

Uni-directional glass fibre reinforced polymers play a central role in the task of increasing the length of the wind turbines blades and thereby lowering the cost of energy from the wind turbine installations. During this, optimizing the mechanical performance regarding material stiffness, compression strength and fatigue performance is essential. Nevertheless, testing composites includes some challenges regarding stiffness determination using conventional strain gauges and achieving correct material failure in the gauge section unaffected by the gripping region. Challenges, which in the present study, has been addressed using the finite element method. During this, a verification of the experimental observations, a deeper understanding on the test coupon loading and thereby improved test methods has been achieved.

Keywords: Composites, Experimental Verification, Fatigue, Scripting

Simulation of Hydraulic Fracturing of Unconsolidated Sands using Fully Coupled Poro-Elastoplastic Models

Saurabh Puri, Juan Hurtado and Deepak Datye (Dassault Systèmes Simulia Corp) Ganesh Dasari, Kevin Searles and Pablo Sanz (ExxonMobil Upstream Research Company)

Many deep water reservoir rocks are soft and consist of unconsolidated or weakly consolidated sands. Fractures may have to be created in these soft reservoirs for a variety of purposes, such as the injection of fluids to maintain reservoir pressure, constructing Frac Pack sand control mechanisms, etc. It is now recognized that fracturing in unconsolidated sands is very different from fracturing in hard rocks. Fracturing in unconsolidated sands is dominated by shear failure as opposed to Mode I tensile failure in hard rocks, and the higher ductility of unconsolidated sands leads to significantly higher net fracturing pressure compared to that of hard rocks. The simulation of these complex failure mechanisms can only be achieved with suitable models that account for the coupling of soft rock deformation and fluid pressure changes. In this paper we discuss the implementation in Abaqus of a poro-elasto-plastic constitutive model for soft rock and its application to some practical problems. First we present a critical-state based constitutive model for soft rock that follows the work of Crook et al. (2006). The model was specifically designed to describe the evolution of sand and weakly consolidated rock from the intact state to the fully damaged state. It includes pressure dependency of the elastic response as well as the yield surface. A softening regularization scheme is also introduced to minimize mesh sensitivity due to strain localization effects. The model is implemented in Abaqus/Standard and Abaqus/Explicit using an implicit backward-Euler integration scheme. Numerical examples are presented in the second part of the paper in the context of applications of practical interest to the Oil and Gas industry. First, we demonstrate the model’s ability to predict shear band formation and localization in a borehole stability analysis. Second, we investigate the evolution of a fracture due to injection of fluids in unconsolidated sand.

Keywords: Geomechanics (Oil&Gas)

Advanced Fracture Modeling for Cuttings Re-injection

Jorge Garzon, Matias Zielonka, Kevin H. Searles and Pablo Sanz (ExxonMobil Upstream Research Company)

An environmentally friendly and cost-effective method for waste-disposal operations commonly used worldwide in the oil and gas industry is known as Cuttings Re-Injection (CRI). It consists of injecting drill cuttings into a suitable geological formation through one or more hydraulically-induced fractures. In this paper, some case studies assessing the risks associated with CRI are introduced using recently co-developed capabilities in Abaqus Standard for simulating fully-coupled hydraulic fracturing. Finite element models (FEM) are constructed using material properties and in situ stress state based on well log information. Furthermore, models are calibrated using data from actual field injection tests. This new 3D simulation capability in Abaqus Standard overcomes some of the limitations of traditional tools and gives engineers an opportunity to assess the impact of planned CRI operations on predicted fracture geometry and pore pressure changes. These variables are very important in understanding risk to formation top seal integrity and possible interaction with pre-existing discontinuities, both of which may lead to loss of fluid containment and non-compliance with environmental regulations.

Keywords: Cuttings re-injection, CRI, Hydraulic fracturing, Pore pressure, Injection, Geomechanics

Experimental Validation of Simulation Capabilities for Hydraulic-driven Fracture Propagating in Porous Medium

Jing Ning, Gilbert Kao, Nikolay Kostov and Kevin Searles (ExxonMobil Upstream Research Company)

The problem of a hydraulically driven fracture propagating in a porous medium is studied using the recently co-developed SIMULIA Abaqus fully-coupled hydraulic fracturing modeling capabilities and validated against lab scale experimental results. A PolyaX Test Cell (PTC) is set up for controlled hydraulic fracturing in fabricated rock samples allowing different testing conditions of confining stress, injection rate, as well as rock and fluid properties. Two representative categories of propagation regime test results (the toughness/storage dominated and viscosity/leak-off dominated) are selected to compare against numerical results carried out in SIMULIA Abaqus. Fracture propagation and fracturing fluid flow are modeled using coupled pressure/deformation cohesive elements in SIMULIA Abaqus, while the porous medium deformation and pore-fluid flow in the rock are modeled with coupled “pore-pressure/deformation” continuum finite elements. Injection pressure response and fracture and leak-off geometries agree well with corresponding experimental results. The lab experiment validation of the newly co-developed hydraulic fracturing capabilities within SIMULIA Abaqus provides confidence in its ability to accurately model field-scale fluid driven fracturing applications including cuttings and water injection, stimulation for unconventional reservoirs, lost returns and other drilling operations.

Keywords: Geomechanics (Oil&Gas)
Dynamic Hydraulic Fracture Modeling for Wellbore Integrity Prediction in a Porous Medium

Nikolay Kostov, Jing Ning and Pablo Sanz (ExxonMobil Upstream Research Company)

Piyush Gupta (University of Illinois Urbana Champaign)

We present a workflow for modeling dynamic growth of hydraulically induced fractures in a porous medium in the region near a wellbore. The focus is on the early time fracture growth behavior and the stress state near the wellbore as a result of it. The model is based on the Cohesive Zone Modeling (CZM) elements with pore pressure degrees of freedom which were co-developed with the ExxonMobil Upstream Research Company and recently implemented into the Abaqus finite element package. The model allows for fluid leak-off from the wellbore and fracture faces into the porous medium and an arbitrary injection schedule in addition to the physics of dynamic fracture growth and pore pressure diffusion. The dynamic fracture growth and interaction between the stress concentration region near the tip and the wellbore result in a non-linear behavior of the near-wellbore stress state during early time fracture propagation. This analysis allows for recommendation of optimum fracture width for maximum increase in wellbore integrity for the target formation. The model also provides an effective tool to improve the treatment selection, parameters like particle size distribution for effective treatment, field diagnostics for various lost circulation treatments and lost return events. The model is incorporated into a fully automated workflow which allows the user to setup the analysis without any previous experience with Abaqus.

Keywords: Geomechanics (Oil&Gas)

A Numerical Investigation on the Bedding Resistance of Laterally Displaced Pipelines

Tim Gerlach, Martin Achmus and Mandy Narten (Institute for Geotechnical Engineering, Leibniz University Hanover)

The soil resistances in axial and lateral direction strongly affect the behavior of earth-covered district heating pipes under variable operating temperatures. The state of knowledge regarding these resistance forces is summarized, and it is shown that almost no information exists regarding the cyclic effects on lateral soil resistances. But, for the safe design of a district heating network, the forces that soil can exert on the structure due to relative soil-pipe displacements are of particular interest. So, a numerical model has been developed, using the concept of hypoplasticity as an advanced constitutive model for noncohesive, granular materials. Additionally the intergranular strain overlay comes into use, enabling a realistic prediction of soil behavior under cyclic loading. The material model is included using the Abaqus umat interface. Results gained from the finite element model are compared to results from an experimental test with cyclic lateral loading of a pipe. Due to the satisfying agreement, numerous variations on geometry and loading device have been done. The investigations give a first insight into the behavior of cyclic laterally loaded district heating pipes. One important result is that the change of bedding resistance under cyclic loading is independent from geometric properties, such as pipe diameter or cover ratio. It is mainly dependent on the range and mean of displacement, as well as on the number of cycles. This finding offers the possibility to formulate a design approach for laterally loaded pipelines, where the cyclic impacts can be normalized on the pipes static bearing capacity.

Keywords: Abaqus, hypoplasticity, intergranular strain concept, soil-structure-interaction, slave-master-concept, district heating networks, pipelines.

Abaqus Analysis Methods on Highly Restrained Pipeline with Soil Berm Formation

Shulong Liu and Emil Maschner (J P Kenny)

Teng Zhang and John Smyth (MCS Kenny)

Wood Group Kenny has developed an advanced pipe-soil interaction subroutine to accompany the pipeline lateral buckling and axial walking analyses in Abaqus. The pipe-soil interaction subroutine takes into consideration both axial friction variation and the lateral soil berm formation mechanism for pipeline lateral buckling and walking analyses subject to cyclic loading. The lateral soil berm formation includes consideration of differential berm growth on the soil berm front and the rear faces, berm resistance accumulation and mobilisation distance and the residual ‘sweep’ friction variation in production operation cycles. It can envelop nodal displacement against its past displacement history around the buckle periphery. Particularly in regard to through life oil and gas pipeline design aspects it can simulate realistic cyclic stress ranges, the soil berm breakout mechanism during rare high temperature /pressure loading events and lateral soil restraint influence in global pipeline walking studies.

Due to the complexity of the berm formation mechanism it has been found that small differences in the initial lateral berm profile generated using various Abaqus analysis methods can have a magnification impact during subsequent cyclic loading. This can be important as it has potential to alter the berm accumulation profile around the developing lateral buckle and increasingly impact on the analysed pipeline behaviour with the number of cycles. To identify the actual pipeline displacement behaviour in production operation cycles and the following mid-life or late life hot oiling events, sensitivity checks using the various analysis numerical methods available to Abaqus users are compared in this paper. The buckle formation sensitivity checks include stabilizer with various stabilizer values, damping factor with various damping values and the less-favoured RIKS in static analysis / quasi-dynamic analysis.

Keywords: Abaqus FEA, Pipe-Soil Interaction, Soil Berm Formation, Pipeline Lateral Buckling, Cyclic Loading, Statics, Dynamics

Pipe Flexibility Analysis and The Development of PCS - Pipe Calculation System for Abaqus

Carlos Eduardo Alves de Castro e Sousa (RMC Consultoria)

The current framework for piping stress analysis is based on a simplified calculation method directly derived from experimental research performed over 60 years ago in the 1940s and 1950s. This framework was originally intended for hand calculations and, apart from minor changes and amendments, has been successfully employed by piping engineers since its development.
As computational power increases and finite element analysis becomes accessible for piping engineers, it has become clear that this framework is not well suited for complex finite element evaluation of piping. Advanced finite element procedures enable engineers to perform in depth evaluation of piping systems that are extremely difficult or even impossible to evaluate through traditional methods. Contrasting to traditional methods, a finite element simulation allow engineers to evaluate creep-fatigue interactions, advanced material models, complex loadings, complex geometries, complex support conditions, as well as the inherent evaluation of fatigue stress intensification factors. A major difficulty for performing piping finite element simulation is that currently there are no high performance finite element packages capable of efficiently generating the required analysis model for complex piping systems. This paper presents the development of PCS - Pipe Calculation System, a pipe flexibility analysis tool for Simulia Abaqus as well as the comparison of its results to existing literature.

**Keywords:** Assemblies, Process Automation

---

**Optimising Wind Turbine Bearing Performance Using Flexible Bearing Ring Analysis**

**Jose Torres (Romax Technology)**

Romax Technology is a global provider of integrated software and services for the design, analysis and optimization of gearbox, bearings and driveline systems across automotive, wind energy, bearings, off road, rail, marine and aerospace. A common assumption when analysing bearing performance is that both the inner and outer bearing rings are rigid. However, this assumption is not valid for all bearings, especially those with relatively narrow cross sections which are housed within flexible structures. For these types of bearings, ring flexibility must be considered in order to accurately predict bearing and housing performance. One application where bearing ring flexibility can have a large influence on performance is in the case of wind turbine blade pitch bearings. To illustrate this, a blade pitch bearing was modelled and analysed using Romax software. The increased accuracy offered through Romax analysis enabled the prediction of phenomena not covered by conventional standards. Thus, design iterations were then optimized. Additionally, for the first time flexible bearing models can now be included as part of a full gearbox analysis within Romax software as modelling and analysis times are extremely short. In this presentation, we will highlight comparisons between a rigid ring and the flexible bearing ring predictions. We will also present recent case studies through our work with major wind turbine OEMs and bearing manufacturers.

**Keywords:** Optimization, Wind turbine bearing, Flexible bearing ring analysis

---

**Design Optimization of a Curved Fir-Tree Root of Steam Turbine Using Computer Aided Optimization and Finite Element Analysis**

**Shang Jiong Tan, Wei Lin Shu, Gong Yi Wang (Shanghai Turbine Works Company Ltd)**

This paper is concerned with the optimization of a turbine blade curved fir-tree root and rim by integrating a knowledge of computer aided optimization (CAO) and finite element analysis (FEA). The optimization software Isight and FEA software Abaqus are applied in an effort to optimize the shape of the curved fir-tree root of a given turbine blade. During this process, root profile from axial perspective is fixed, while attention is devoted to examining the effect of geometrical feature from the radial perspective. Several critical geometrical parameters such as the radius of root centerline, x, y coordinate of center of the centerline, and distances from blade center to leading and trailing edge of the platform, are taken as design variables and the maximum equivalent stress of root and rim as objective function. Compared with the original structure, the position of the maximum equivalent stress changes while the maximum equivalent stress of both root and rim are reduced significantly, resulting in increased operation reliability of turbine blade. A product model from STWC is used as a base design in the optimization.

**Keywords:** Steam turbine blade, Curved root, Finite element analysis, Multi-Island Genetic Algorithm, Downhill Simplex

---

**Increasing the efficiency of offshore rigid pipeline lateral buckling assessments using a dedicated GUI and Isight**

**Carlos Charnaux, Graeme Roberts and Samuel Paul (Subsea 7)**

The design phase for offshore rigid pipelines can be lengthy and demanding. A significant number of engineering hours can be burned before the various analyses attain target results. The assessment of the lateral buckling phenomenon is one aspect of the pipeline design that requires numerous finite element simulations. The purpose of such simulations is to determine if the pipeline has a genuine tendency for lateral buckling and, if so, whether control of the buckling behaviour is required to ensure that the pipeline stresses and strains are within allowable limits. The process map for finite element based lateral buckling assessments follows a distinct procedure distributed into four generic steps: (1) gathering input information, (2) constructing finite element models, (3) running the simulations and (4) results post-processing. Typically, the process map is followed manually, and requires engineering analysts to develop quite complex FE models using typed data input with subsequent use of batched Python scripts to acquire model results, verify and validate the analysis runs and make tabular and graphical output. Depending on project requirements and other circumstances such as availability of license tokens and disk space, the simulation process can be relatively inefficient and places a limit on the number of analysis runs that can be made and, therefore, the range of input that can be assessed. The latter point is important because the results of lateral buckling assessments depend heavily on input uncertainties (e.g. pipe-soil interaction data) and limiting the number of analysis runs may lead to conclusions that are not based on sound engineering judgment. In modern offices, it is rare to find engineers who are conversant and comfortable with developing batch processes or writing computer code that would enable a large range of jobs to be completed and post-processed efficiently. However, Windows-based applications such as the user-friendly Isight program allow engineering processes to be automated in order to enhance the results fields and to better understand the influences of the uncertainties of some input parameters. Subsea 7 has developed a Graphical User Interface (GUI) that works with Isight and which fully automates lateral buckling assessments, from the input gathering stage through model generation, post-processing, graphical output and model verification. This paper demonstrates the current status of...
the automation process which is built around the “Splatter” GUI, Isight workflows and in-house Python scripts. The process enables engineers with relatively little experience of Abaqus to create, analyze and post-process Abaqus models for lateral buckling, pipeline walking and on-bottom roughness assessments.

**Keywords**: Design of Experiments, Exploration, Lateral Buckling, Optimization, Pipeline, Pipe-Soil Interaction, Post-processing, Probabilistic Design, Scripting.

**Finite Element Simulation of Rotary Steerable Drilling Systems Advance Understanding of Damaging Downhole Shock and Vibration**

Nader Abedrabbo and Liam Lines (Weatherford)

The latest generation of rotary steerable drilling systems make it possible to reach hydrocarbons located deeper and in more hostile environments than ever before. These systems give the directional driller, standing some 10,000ft above on the rig floor, the ability to steer the well path in real-time to reach the elusive ‘pay zone’. Rotary steerable drilling systems, which are located at the end of the drilling assemblies, are exposed to extreme conditions. In addition to temperatures in excess of 390°F and pressures up to 25,000 psi, they must also convey compressive and bending loads, torque and rotation to the drill bit from the surface, all while controlling the 3D trajectory of the wellbore. Shock and vibration sensors highlight the severity of the downhole environment, but the root cause of vibration is difficult to determine based on sensor data alone. Industry standard predictive analysis techniques are incapable of detecting the effects of contact dynamics and real-world testing has shown that these are among the most damaging to downhole drilling tools, significantly reducing efficiency of the drilling process. With some deepwater rig rates exceeding $500,000 per day, the most successful systems are those that drill fastest with the lowest number of downhole failures. Advanced finite element transient dynamic analysis and the manner in which it yields new insight into downhole shock and vibration shows that a combination of 1D beam elements and real 3D part geometries is critical to accurately predict the drilling assembly’s dynamic signature, while still allowing for a timely solution process. This analysis enables the complex drilling assembly to be modelled within the confines of the wellbore. The analysis also assesses dynamics associated with rotating contact, such as lateral and torsional vibration, as well as the excitation of the drilling assembly’s structural resonant modes. Transient dynamic analysis enables drilling assemblies and downhole tools to be designed from the outset to minimize contact dynamics and avoid excitation of damaging structural resonances. In addition to improved design of downhole tools, the results of this analysis also show the drilling team what drilling parameters are optimum to minimize and mitigate shock and vibration while drilling.

**Keywords**: Vibration, Drilling Dynamics, Contact Dynamics, Rotary Steerable, Downhole Tools, Bottom Hole Assemblies, Oil & Gas.

**HIGH-TECH**

**Achieving Higher Productivity on Abaqus Simulations with HPC and Clustering Technologies**

Pak Lui (HPC Advisory Council)

Engineers from wide ranges of industries face an ever increasing need to run virtual tests to simulate complex models for improving reliability as well as reducing product development time and costs. Abaqus is designed to tackle these challenges by producing high-quality realistic simulation solutions, while delivering high performance in productivity by utilizing efficient use of modern compute cluster available in High Performance Computing (HPC). Many organizations can now deploy with HPC cluster to process such complex analyses on Abaqus with computer cluster to render such high-quality models, while reducing simulation time from days to just hours. Behind this type of computational and clustering technologies that enables Abaqus to perform, it involves complex calculations and data exchanges among computational systems. The HPC Advisory Council has performed a deep investigation on Abaqus to evaluate its performance and productivity capabilities and to explore potential optimizations. This study presents the techniques and profiling results to further understand Abaqus dependencies on the CPUs, network, IO storage, and the other underlying hardware. The paper will review the effects by comparing various hardware using different simulation models on Abaqus.

**Keywords**: Abaqus/Standard, Abaqus/Explicit, clusters, High Performance Computing, HPC, Benchmarking, Ethernet, Tuning, CPU, Power, network, InfiniBand, Turbo Mode, MPI Profiling, IO Profiling

**INDUSTRIAL EQUIPMENT**

**Robust and Optimal Strain Gauge Positions for Blade Vibration Testing**

Piyawan Teufel (ABB Turbo Systems AG)

Jan Poland (ABB Corporate Research Center)

One important and widely used instrumentation to measure blade vibration are strain gauges. To find an optimal application for a small number of gauges to capture the modal strain amplitude over a wide range of natural frequencies is a major challenge. This paper focuses on the strategy for an automatic selection of optimum gauge positions and orientations with strain sensitivity, position overlapping, explicit mode assignment and robustness as criteria. The strategy assures that all required vibration modes are well detected from at least 2 strain gauges in case one strain gauge fails during testing. This automatic procedure is written in the Abaqus Scripting Interface. The selected location and orientations of strain gauges can be visualized within the Abaqus odb. The measured strain in gauge direction is shown within a gauge size.

**Keywords**: strain gauge, high cycle fatigue, blade vibration, automatic pre- and post-processing, Abaqus scripting language, Python, optimization, hill climbing, tabu search
Prediction of Tin-Whiskers Generation Using Stress and Mass-Diffusion Analysis

Takeshi Terasaki (Hitachi, Ltd.)

A spreading environmental awareness has prompted a global movement towards lead-free electronic products, in which lead-free tin coating is included. Lead-free tin coating provides good solderability equal to tin-lead coating, however the spontaneous whisker growth on lead-free tin coating is a serious reliability issue. It’s imperative to clarify the whisker generation and growth mechanism. So we have developed a multi-scale simulation technique for calculating tin atom diffusion caused by stress gradient induced by temperature change and crystal orientation distribution in polycrystalline tin coating. This multi-scale simulation technique uses molecular dynamics (MD) simulation and a finite element method (FEM). The FEM simulates macro-scale stress and stress-induced diffusion, and the MD simulates nano-scale atomic diffusion. The stress analysis model considers elasticity anisotropy, thermal-expansion anisotropy, and the crystal orientation of tin. A thermal cycling test was conducted to induce whisker generation on tin-plated specimens, and the crystalline orientations around the whiskers were evaluated using electron back-scattering diffraction pattern (EBSP) measurement. The hydrostatic pressure distribution and tin-atomic-density distribution in the specimens were calculated using our technique. The whisker locations corresponded to the areas of higher tin-atomic density and lower hydrostatic pressure on the tin-plated surface, indicating that our technique can predict whiskers generation in tin coatings.

Keywords: Tin whisker, Tin plating, Electronic product, Lead-free, Finite element analysis, Atomic diffusion, Stress gradient, Crystal orientation, Elasticity anisotropy

Strength and Fatigue Analysis of Leaf Springs used in Heavy Duty Trucks

Mehmet Bakir, Murat Siktas, Serter Atamer (Mercedes-Benz Turk A.S.)

In the world of today, there are a noticeable number of weight and cost reduction projects within the vehicle engineering development activities. Regarding this phenomenon, a complete optimization study is applied to a leaf spring assembly, which is used in heavy duty trucks, by reducing the number of leaves together with weight and cost reductions. At the first step of the project, the stiffness of the leaf spring is calculated with in-house software based on mathematical calculations using the thickness profile of the leaves. Then the results of these calculations are compared with non-linear elastic leaf spring calculations which are conducted with Abaqus. This elastic leaf spring finite element (FE) model is transferred into multi body simulation (MBS) model, which is established with Simpack, in order to determine the forces acting on the leaf spring. Using the results of the MBS calculations, which are time histories of the internal forces and moments on the leaf spring, the FE simulations are performed with Abaqus. Stress results are not always enough to conclude that the leaf spring will preserve its integrity until the desired lifetime although they may give a rough estimation. Concurrently, without fatigue life analysis it is not correct to make a final decision. Therefore, as the final step of the project, a multi-channel fatigue life calculation process is performed to assess the durability characteristics of the new design, showing that the weight and cost reduction is achievable without worsening the fatigue life of the leaf spring.

Keywords: Damage, Dynamics, Elasticity, Fatigue Life Analysis, Finite Element Analysis, Multi-Body Dynamics, Optimization, Suspension

Coupled Analysis of Fracture Mechanics and Piezoelectricity in Active Layers in the Abaqus Code Operated Through the Isight Tool

Eugenio Brusa (Politecnico Di Torino - Dipartimento Di Ingegneria Meccanica e Aerospaziale)
Mehdi Mohammadzadeh Sari (Politecnico Di Torino)
Cosima Fiaschi, Mauro Parodi (EXEMPLAR)

A key issue of design of piezoceramic layers is preventing some damage of material like fracture, fatigue and creep. Nevertheless, prediction of the crack propagation through a piezoelectric layer while its active functions are exploited is rather difficult. Analytical methods were already proposed, but they fail when geometries are more complex than those of rectangular patches or structural behavior becomes nonlinear because of the geometrical effects, like in case of large vibration amplitudes, typical of some energy harvester. Developing a reliable numerical tool to predict the fracture mechanics of actively coupled piezostructures is a current demand of industry to face the problem of predicting the risk of damage of material with a sufficient confidence to assure the safety factors required by the technical standards. Until now very seldom it was required to investigate simultaneously the fracture mechanics in presence of either the direct or reverse piezoelectric effect. Therefore providing a coupled numerical solution which might take into account both the piezoelectric phenomenon and the crack propagation looks quite a new need for the developers of software. The Abaqus code proposes two different numerical tools for analyzing the fracture mechanics and the piezoelectricity, although they are not yet directly connected to perform a coupled solution. This paper describes the research activity performed to investigate the crack propagation in coupled piezoelectric structures by resorting to those tools. Fracture mechanics inside the smart material is described by calculating the so-called Stress Intensity Factor (SIF), the J-integral as well as the crack propagation by including the local effects induced by crack geometry upon the material being due to the electromechanical coupling between electric field and mechanical strain. The Isight program was then used to connect the unit dealing with piezoelectricity and that of fracture mechanics inside the Abaqus code and to perform the required analysis. Results demonstrate that a strong coupling effect is present in propagation and some peculiar behaviour is exhibited, being somehow helpful to reduce the impact of crack upon the piezoceramic and increasing its life.

Keywords: Composites, Coupled Analysis, Crack Propagation, Dynamics, Fracture, Piezoelectric Structures, Smart Materials
**XFEM Simulation of Radial-Median Crack Evolution in Knoop Indentation of Brittle Materials**

Felix Rickhey (Sogang University)
Jin Haeng Lee (Korea Atomic Energy Research Institute)
Hyunggil Lee (Sogang University)

In sharp indentation of brittle materials, a complex three-dimensional stress field evolves in the material beneath the indenter. During unloading, the plastic zone, which has emerged during the loading cycle around the indenter tip, inhibits the neighboring elastic matrix to return to its original shape. The resulting high tensile stresses at the indenter diagonals may cause crack initiation or extension. For the simulation of crack evolution in a material subject to a stress field which changes with position, time and crack propagation, the extended finite element method (XFEM) seems to be an ideal tool. Using XFEM, we analyze the formation of cracks occurring during Knoop indentation of brittle materials. Issues associated with XFEM application in Abaqus/Standard are discussed for the problem at hand, and suggestions are made with regard to the artificially attributed viscosity parameter, the damage-initiating stress threshold and meshing. Applying the maximum principal stress criterion for damage initiation, we find that cracking is confined to the median plane through the major indenter diagonal. It is observed that by contrast with Vickers indentation the high aspect ratio of the Knoop indenter and its blunt nature cause a flattening of the plastic zone (which can no longer be regarded as hemispherical) and the crack to initiate not at the surface but beneath the plastic zone inside the material. The final crack has nevertheless the same half penny-shape, which is characteristic of Vickers and Berkovich indentation.

**Keywords**: Knoop Indentation; Brittle Materials; Fracture Toughness; XFEM

**Evaluation of the Fatigue Strength of Threaded Connections using Local Concepts**

Dominic Olveda (Technische Universität Darmstadt)
Ralf Waterkotte (Schaeffler Technologies GmbH & Co. KG)
Matthias Gechsner (TU Darmstadt)

VDI (Association of German Engineers) guideline 2230 [1] provides a basis on which the fatigue strength of bolt-nut joints can be estimated. However, it is not suitable for use in reference to general threaded joints, as the VDI guideline is based on the nominal stress concept and it is often not possible to define a nominal surface area for such general joint types. Furthermore, regulations that do not explicitly deal with the design of threaded joints – such as the FKM guideline “Analytical Strength Assessment of Components” [2] – also deliver unsatisfactory estimates of the fatigue strength of threaded joints. This paper deals with the design of a steel/aluminum tapped thread joint. The concept developed by Schneider [3] is used as the basis for assessment up to the occurrence of technical cracks. Crack propagation, up to component failure, is described with linear elastic fracture mechanics and using the finite element method (FEM). For the purpose of validating the concept, the calculated lives are compared with experimental tests.

**Keywords**: Durability & Fatigue

**Strip Shape Prediction using a Simplified Static Model of a 4-High Cold Rolling Process**

Mario Alberto Bello-Gomez, Martha P. Guerrero-Mata, Luis A. Leduc-Lezamaa and Tania P. Berber-Solano (UANL)
Fernando Gonzalez (Ternium Mexico)

The cold rolling process is one of the most important in the metal industry. The strips produced by this process are used in different applications. The quality and shape control of these products are very important factors during the production process. It is well known that the strip shape is influenced by: the rolling speed, strip width, forces, rolls deflection, etc. This work considers modelling a 4 high cold rolling process and obtains the strip profile, with a static model using an arc contact with springs representing the whole model. This idealized model was developed to reduce the computational time of the dynamic model being used. An arc is defined with the optimal dimensions to reproduce the process, taken into account concentrations of the stresses, forces and displacements at the work roll/strip contact; the arc used in the static model was meshed in accordance with the mesh of the work roll in the dynamic model. The arc in this model is helped by springs placed at the arc inner surface, the stiffness of the springs was calculated with previous measurements of the work roll deflection values. The results show that the arc static model reproduces correctly the roll profile and strip shape with an important difference in the computational time.

**Keywords**: Durability & Fatigue

**Validation of Flow Simulation on Abaqus/CEL™**

Edgar Ramirez (Universidad Nacional Autónoma de México (UNAM))
Alan Saucedo, Osvaldo Ruiz, Victor Jacobo and Armando Ortiz (Engineering School, UNAM)

The main paper topic is to show the Eulerian-Lagrangian coupled analysis (CEL) validation applied to Newtonian fluid dynamics simulations described by an equation of state (EOS) suggested by Mie-Grüneisen. The model is driven only by body forces through cavity filling. This implementation allows to obtain free surfaces profile, splash droplet formation, velocity field, instantaneous pressure at the contact points between solid and fluid, as well as in solid parts; stresses and strains, all at the same time. The free surface validation obtained in the fluid is made through comparisons with radiographs obtained experimentally. The filling is achieved through solid-fluid interaction (FSI) with “General contact (explicit)” (GC) this is able to track the material distribution during the free surfaces formation and applies non-slip boundary condition as interaction between fluid and solid in order to determine the effect of this approach on the material flow. As a result the complete standardized cavity filling is achieved satisfactorily using CEL analysis.

**Keywords**: Cavities Filling, Material Flow Analysis, CEL, Fluid-Solid Interaction (FSI).
On the Numerical Implementation of a 3D Fractional Viscoelastic Constitutive Model
Gioacchino Alotta (University of Palermo - University of Oxford)
Olga Barrera, Alan C. F. Cocks (University of Oxford)
Mario Di Paola (University of Palermo)

The aim of this paper is the implementation of a 3D fractional viscoelastic constitutive law in a user material subroutine (UMAT) in the finite element software Abaqus. Essential to the implementation of the model is access to the strain history at each Gauss point of each element in a constructive manner. Details of the UMAT and comparison with some analytical results are presented in order to show that the fractional viscoelastic constitutive law has been successfully implemented.

Keywords: Fractional viscoelasticity, Constitutive model, Abaqus/Standard, Creep, Relaxation

Fractional Viscoelasticity, Constitutive Model, Abaqus/Standard, Creep, Relaxation Modeling of Hydrogen Embrittlement Mechanism in Metals
Olga Barrera, Alan C. F. Cocks and Edmund Tarleton (University of Oxford)

It is well known that high strength steels are tremendously affected by hydrogen. The aim here is to provide a modelling of the HELP (Hydrogen Enhanced Local Plasticity) mechanism fully coupled with hydrogen transport equation. The hydrogen diffusion equation is implemented in a UMATHT subroutine by considering similarities between heat and mass diffusion equations. A coupled temperature-displacement procedure has been adopted to allow the coupling between hydrogen diffusion and the mechanical behaviour of the material.

Keywords: Hydrogen embrittlement, Constitutive models, hydrogen diffusion equation, UMATHT

LIFE SCIENCES

Developing a Constitutive Model for a Highly Deformable Latex Rubber with Implementations in Lagrangian Frameworks
Christopher Basciano, Marcus Rademacher, Patrick Downie and Anita Bestelmeyer (BD)
Brian Croop (DatapointLabs)
Jorgen Bergstrom (Veryst Engineering LLC)

To improve product design and enhance the realism of BD’s advanced finite element models, the development of a constitutive model capable of predicting mechanical behavior at high deformations was initiated for a commercially available latex rubber. A set of baseline experiments was conducted to determine the isothermal mechanical response(s) of the latex rubber. The experiments were designed to provide fundamental insight into any viscoelastic, hyperelastic, and Mullins effects exhibited by the material. A constitutive model was constructed for specific load cases from the aforementioned experimental data. Attention was given to the model’s primary use case, numerical stability, and its ability to perform at high deformation ranges. A second set of experiments was then conducted, aimed at characterizing the failure of the rubber and providing a set of experimental data to validate the material’s constitutive model. The first round of experiments revealed that the material’s stress vs. strain curves held limited dependence on the applied loading rate, while the failure stress and strain had a moderate dependence. Additionally, cyclic loading and unloading of the material was shown to have a prominent effect on the failure stress and strain. Based on the model’s intended use case and baseline mechanical response, a hyperelastic constitutive model was calibrated to the monotonic, uniaxial tension data. The model was then implemented in Lagrangian and Eulerian geometries and compared against the second set of experimental data. The constitutive model exhibits good agreement to the calibration data when strain is less than 5.0, but begins to significantly underpredict the stress values at strains above 6.0. Validation studies revealed good agreement between the experimental tear test data and different implementations of the constitutive model with different boundary conditions. Work is ongoing to extend the implementations of the constitutive model to an Eulerian framework and refine the constitutive model calibration.

Keywords: Constitutive Models, Hyperelasticity, Hyperelastic Material, Tear Test, Rubber

Developing a FE Head Model for Impact Simulation in Abaqus
Philippe Young (Simpleware)
Siddiq Qidwai, Alan Leung, Amit Bagchi and Nithyan and Kota (Naval Research Laboratory)

A computational model of the human head has been developed by Simpleware and the U.S. Naval Research Laboratory (NRL) to simulate head impact. The head model was generated using high-resolution scan data, segmentation and image-based meshing techniques, and has been used in Abaqus for a range of simulations. The NRL/Simpleware head model addresses several key limitations of existing techniques for head impact modelling, most notably mesh adaptability for different applications. This paper will focus on the development of the modelling techniques used to create the NRL/Simpleware head model, and its validation in Abaqus/Explicit. The model has a high level of geometric fidelity, and is suitable for explicit and implicit finite volume-based analyses.

Keywords: Abaqus, Abaqus/Explicit, Biofidelic, Blast Impact, Blunt Impact, CAD, FEA, Finite Element Method, Head Model Mesh Generation, Image Segmentation, Simpleware, Military, MRI Scan, Traumatic Brain Injury, US Naval Research Laboratory, and Visualization.
Drop Test of a Hearing Aid
Morten Birkmose (Soendergaard GN Resound A/S)

In the development of hearing aids many factors are important to make a good hearing aid. It can be the sound quality, design and easiness of use. But it is also important that the device can withstand the environment that it is used in, both regarding moisture, sweat and to be dropped on a hard surface. One of the most sensitive parts in the hearing aid regarding dropping the device on a hard surface is the receiver (name for the loudspeaker in the hearing aid). The development of hearing aids demands shorter time to market and higher rate of success in achieving all the development requirements for the device. Therefore it is important to be able to predict and improve designs in development projects. In this case a FE model of the hearing aid is built to predict the g force of a receiver inside the hearing aid, dropped from 1 m height. The simulated g force is compared to real drop test. The simulation is in 2 steps. First some parts needs to be prestressed in Abaqus/Standard. Secondly the deformed parts are imported in the drop test model build in Abaqus/Explicit. In this model it is only the sensitive parts that are deforming, all other parts are made rigid. The receiver has normally a soft suspension surrounding it, which will absorb the impact. Different designs of the soft suspension are simulated, to give the best impact protection as possible.

Keywords: Impact Simulation, Finite Element, Hearing Aid, Hearing Instrument, Industrial Application, Microphones, Receiver, Loudspeaker, Ear, Hearing, Life Science, Medical, Dynamics & Shock

Modeling of a Flatfoot Using Abaqus for Improving Surgical Treatment
Zhongkui Wang and Shinichi Hirai (Ritsumeikan University)

This work presents a finite element (FE) model of flatfoot for studying flatfoot pathology and improving surgical plan for individual patient. Despite being a common foot deformity, the pathology of flatfoot has not been fully understood. Regarding optimal surgery, medical doctors tend to have different opinions for an individual patient. Studies using cadaver foot have difficulties to cope with individual differences of foot geometry and therefore have limited contributions for improving surgical planning. One option for solving this problem is to use computer-based model. Due to the essence of continuum mechanics, FE method has been used to model human foot for various applications. In this work, software package Abaqus is used to model and simulate flatfoot. The geometries of bones and soft tissue of a flatfoot were generated from CT images and imported into Abaqus as separated parts, which were then assembled and meshed with linear tetrahedra. Ligaments, planta fascia, tendons were modeled with truss elements and the approximate positions of origin and insertion were referred from an anatomy book and under the instruction of medical doctors. The contact interactions between different bones and between soft tissue and ground were modeled using penalty method with friction. A total of 17 bones, 62 ligaments, 9 tendons, 6 plantar plantar fascia segments, and one soft tissue part have been modeled. Weight-bearing simulation was performed and the stress distribution was compared with experimental measures to validate the model. Validated model will be used to study flatfoot pathology and investigate the optimal surgery for individual patient.

Keywords: Flatfoot Deformity, Surgical Simulation, Biomechanics, Elasticity, Hyperelasticity

TRANSPORTATION & MOBILITY

Efficient Evaluation of Material Cards for Non-reinforced and Reinforced Thermoplastics
Peter Reithofer (4a)

The demands for shorter product development times and the consequent increased use of simulation programs have led to a greater need for quickly acquiring material data. The testing system 4a impetus™ was developed for this purpose by 4a engineering GmbH. Granta Design Ltd. helps to manage information on materials (metals, plastics, composites and more). Their software products (CES Selector™, GRANTA MI™) are a helpful extension in the material card generation process, enabling full traceability and management of the materials information involved, especially at the start and end of the process. The system 4a impetus™ includes the complete material card generation process, typically consisting of: Performing many test methods (bending, clamped bending, pressure tests, puncture tests, component tests) • Managing the tests and simulations in a database, using various solvers (Abaqus, LS-Dyna, Pam-Crash) for simulating the tests, and finally, linking to optimization algorithms to identify the material parameters by reverse engineering. Mainly the focus of the testing is the bending load cases, which has the advantages of simple testing, near-reality loading, and that test specimens can be taken from parts. By using the testing device 4a impetus™ strain rates up to 200 1/s can be reached, which enables generation of material cards for crashworthiness simulations. Setting up more complex material cards (general yield surface, failure) is possible through additional tests (e.g. clamped bending, puncture test). So 4a impetus™ enables very easy, quick, and cost efficient creation of validated material cards. This will be illustrated in several examples.

Keywords: Material Modeling

IDIADA’s Virtual Proving Ground (VPG) for Durability Analysis
Jordi Arbiol, Xavier Armengol, Jose Antonio Muñoz and Inés Lama (APPLUS IDIADA)

It is usual that the applied loads on structural elements are in fluctuating situations. Sometimes the component can break under its ultimate or its yield strength. This break situation is due to the fluctuating loads that have been applied for long periods. For this purpose this breaking type is called “fatigue breaking”. Fatigue breaking is a consequence of a crack growth. When no prototypes are available, it is necessary to calculate the life prediction as soon as possible. Simulation through the VPG is being increasingly used for life prediction. The present work shows a methodology that uses a Finite Element Model (FEM) that runs on a proving ground. In this case, no MBS (multibody software) model is used to obtain the loads on the
System Modeling and Simulation at Dana Holding
Frank Popielas (Dana Holding Corp)

System Modeling and Simulation (SMS) is the next logical step within Dana's engineering community to achieve a fully integrated, front-end driven, optimized engineering ecosystem. We believe this evolution will enable Dana to maintain continued leadership in the industry for delivering product innovation, quality and customer responsiveness. In order to achieve this vision in an efficient and coordinated way, we at Dana identified the basic steps we have to take, determine which engineering areas are critical participants, and, decide on which technology pillars we have to focus our attention. At the very beginning of this journey we understood that this means a step change for a new way of thinking as well as a cultural change within our engineering community, company and extended enterprise.

In this paper we explain what SMS means to Dana, how we approach this topic, and what technologies we are investigating in order to achieve this new level of engineering. Since the process is very complex, we decided to start with a business value assessment (BVA). This engineering assessment was undertaken to understand the current status of our engineering organizations and existing technology, and then identify the gaps to where we want to be with our business. This helped to formulate a foundational strategy and long-term roadmap. With an existing strong focus on the Abaqus/CAE environment this also meant we needed, as previously mentioned, enhance the mindset and our global culture to make the next step in our systems evolution. This only can be achieved through a dedicated effort to educate and engage the people throughout the

Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach
Vinod Kumar, Vinaig Somashekkar and Srivatha Jagalur (Autoliv India Pvt. Ltd.)

Electronic devices used in automotive safety has a trend to reduce size in packaging and increase performance of the product, both of which contribute to exponential increase in power consumption of the system consequently increasing the heat generation. Designers have limited choice on PCB board components but have flexibility in designing the enclosures over the PCBs. This calls the need for an optimal enclosure design to maximize the heat dissipation and to reduce the temperature well below the allowable operational limit. Majority of heat generated by these devices get dissipated by conduction and natural convection modes. Heat generated by PCB chipsets due to power consumption is transferred to the cover and housing via heat sink and thermal pads respectively by convection and conduction. The surface film coefficients estimated from the bench tests were used to simulate convection heat transfer from housing and cover to ambient using Abaqus. To arrive at an optimal housing and cover design, a DoE approach is used with geometrical, material and positioning as parameters. Efficient cover and housing design ensuring better functionality with maximum heat dissipation capability in terms of conduction and convection is achieved with minimum number of physical tests by DoE approach of thermal analysis using Abaqus. Keywords: Thermal management, Steady State Heat Transfer, Automotive safety, DoE

Keywords: Thermal Management, Steady State Heat Transfer, Automotive safety – A DoE approach

Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach
Vinod Kumar, Vinaig Somashekkar and Srivatha Jagalur (Autoliv India Pvt. Ltd.)

Development of a Separator Plate for Valve Bodies in a Vehicle Automatic Transmission
Harald Rebien, Tobias Tauscher and Rohit Ramkumar (Dana Holding Corp)

Development of a Separator Plate for Valve Bodies in a Vehicle Automatic Transmission. Automatic transmission in light vehicles are complex hydraulic systems. To change gears, oil in hydraulic channels need to switch between correct valves. These hydraulic channels are ordered mainly on two valve body flanges. The separator plate seals all channels from both flanges and at some locations the separator plate needs to provide a fluid connection between two channels. Beside this, the separator plate needs to manage low bolt forces, reduce distortion of the valve body flanges and has to be durable under all conditions. Problem: Nowadays, the transmissions need to be economically optimized system. To meet CO2 goals, the transmission also has to be more efficient. One possibility is to raise the operating pressure in the hydraulic fluids. But this leads to a higher leakage rate and therefore larger pressure pumps are required. To overcome this dilemma it was necessary to bring technical innovations to the classical separator plate. Lower leakage rates at higher pressure demand a more efficient sealing solution. If this solution works efficiently it could provide opportunity to reduce the number of required bolts. This helps in making the design of hydraulic channels more flexible. Results: With the help of simulation tools provided by Simulia (Abaqus & Isight) it was possible to create a finite element model which replicated the real world conditions to predict the sealing performance, flange and valve seat distortions at different operating conditions & also optimize the sealing beads for manufacturability. Simulation tools helped us to develop a new high performance gasket which met all the requirements. Comparisons with traditional separator plates showed that it is possible to eliminate several bolts, which leads to a better hydraulic channel design at lower cost. Conclusion: Using upfront Abaqus/CAE tools helped us to create a simulation driven design. New innovative sealing technology could be developed and applied to individual customer demands. With the new innovative separator plates DRNA could convince customers to apply the new technology on their products. Additionally this new innovative separator plates are nominated for the PACE-Award for 2014. Keywords: Automatic transmission, Powertrain, Drivetrain, Automotive, Bending, Bolt load, CAE, Deformation, Design optimization, FEA, Forming, gasket, Sealing pressure, Transportation

Development of a Separator Plate for Valve Bodies in a Vehicle Automatic Transmission
Harald Rebien, Tobias Tauscher and Rohit Ramkumar (Dana Holding Corp)

Development of a Separator Plate for Valve Bodies in a Vehicle Automatic Transmission. Automatic transmission in light vehicles are complex hydraulic systems. To change gears, oil in hydraulic channels need to switch between correct valves. These hydraulic channels are ordered mainly on two valve body flanges. The separator plate seals all channels from both flanges and at some locations the separator plate needs to provide a fluid connection between two channels. Beside this, the separator plate needs to manage low bolt forces, reduce distortion of the valve body flanges and has to be durable under all conditions. Problem: Nowadays, the transmissions need to be economically optimized system. To meet CO2 goals, the transmission also has to be more efficient. One possibility is to raise the operating pressure in the hydraulic fluids. But this leads to a higher leakage rate and therefore larger pressure pumps are required. To overcome this dilemma it was necessary to bring technical innovations to the classical separator plate. Lower leakage rates at higher pressure demand a more efficient sealing solution. If this solution works efficiently it could provide opportunity to reduce the number of required bolts. This helps in making the design of hydraulic channels more flexible. Results: With the help of simulation tools provided by Simulia (Abaqus & Isight) it was possible to create a finite element model which replicated the real world conditions to predict the sealing performance, flange and valve seat distortions at different operating conditions & also optimize the sealing beads for manufacturability. Simulation tools helped us to develop a new high performance gasket which met all the requirements. Comparisons with traditional separator plates showed that it is possible to eliminate several bolts, which leads to a better hydraulic channel design at lower cost. Conclusion: Using upfront Abaqus/CAE tools helped us to create a simulation driven design. New innovative sealing technology could be developed and applied to individual customer demands. With the new innovative separator plates DRNA could convince customers to apply the new technology on their products. Additionally this new innovative separator plates are nominated for the PACE-Award for 2014. Keywords: Automatic transmission, Powertrain, Drivetrain, Automotive, Bending, Bolt load, CAE, Deformation, Design optimization, FEA, Forming, gasket, Sealing pressure, Transportation

Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach
Vinod Kumar, Vinaig Somashekkar and Srivatha Jagalur (Autoliv India Pvt. Ltd.)

Electronic devices used in automotive safety has a trend to reduce size in packaging and increase performance of the product, both of which contribute to exponential increase in power consumption of the system consequently increasing the heat generation. Designers have limited choice on PCB board components but have flexibility in designing the enclosures over the PCBs. This calls the need for an optimal enclosure design to maximize the heat dissipation and to reduce the temperature well below the allowable operational limit. Majority of heat generated by these devices get dissipated by conduction and natural convection modes. Heat generated by PCB chipsets due to power consumption is transferred to the cover and housing via heat sink and thermal pads respectively by convection and conduction. The surface film coefficients estimated from the bench tests were used to simulate convection heat transfer from housing and cover to ambient using Abaqus. To arrive at an optimal housing and cover design, a DoE approach is used with geometrical, material and positioning as parameters. Efficient cover and housing design ensuring better functionality with maximum heat dissipation capability in terms of conduction and convection is achieved with minimum number of physical tests by DoE approach of thermal analysis using Abaqus. Keywords: Thermal management, Steady State Heat Transfer, Automotive safety, DoE

Keywords: Thermal Management of Electronic Devices used in Automotive Safety – A DoE approach
Vinod Kumar, Vinaig Somashekkar and Srivatha Jagalur (Autoliv India Pvt. Ltd.)
process. Therefore, as a very important topic we will also discuss the meaning of simulation, Abaqus/CAE in general, system engineering (SE), model-based system engineering (MBSE) in the context of SMS for this new environment.

Keywords: Multi-body Dynamics and Systems Engineering

**Modeling the Draping of NCF Composite Preforms**
Muhammad-Ali Siddiqui and Colmar Wocke (Dow Europe GmbH)

Non Crimp Fabrics (NCF) are widely used textile reinforcements in fibre reinforced composite materials structures. The most important prediction from a draping analysis featuring preforms from these materials, is their resulting fibre orientations, the orientations influence several other properties including fabric permeability and final component strength. The analysis is complicated by the influence of localized stitching patterns and their constraint on relative sliding of individual layers within a single ply during the draping process. The modeling procedure involves a process of material characterization valid for fabric preform materials and has been carried out using the Abaqus /Explicit finite element analysis code. The work is concerned with the development of such a predictive model using a rebar-approach in Abaqus. Biaxial and triaxial NCF are modeled using multiple shells. The anisotropic properties of the preform are naturally represented by the model. The details and capabilities of the model are discussed and qualitative comparisons are made with real fibre angles taken from prepreg NCF reinforcement on hemispherical dome and three dimensional test shape forming experiments. The model has been used to compare predictions to the force vs. deflection results from single dome forming tests as well as the comparative ability to predict the location and severity of wrinkles. The current model demonstrates a high level of functionality allowing accurate prediction of yarn and stitch pattern movement. The adopted modeling technique using separate shells works reasonably well, although it brings with it an increased requirement on computer resources.

Keywords: Abaqus, Preforms, Composite, Anisotropic

**New Constitutive Model for Woven Thermoplastic Composite Materials**
Pieter Volgers, Helga Kuhlmann and Zhenyu Zhang (DuPont Engineering Technologies)

This paper describes the behavior of the DuPont thermoplastic composite sheet offering, under the name Vizilon™ and matches this to composite models as implemented in Abaqus. Subsequently, to obtain a more accurate description of the actual behavior of these materials, DuPont has developed a new material model by combining both plastic deformation of the thermoplastic matrix material with a more classical composite damage model for the description of the non-linear permanent deformation of the woven continuous fiber reinforcement and subsequent failure. The (current) results obtained with this model, implemented in Abaqus, are presented, demonstrating the potential of this model for thermoplastic composites.

Keywords: Composites, Constitutive Model, Crashworthiness, Damage, Fabrics, Failure, Plasticity, Polymer

**Durability Assessment of Heavy Duty Diesel Engine Cylinder Head**
Ahmet Hamdi Güzel (Ford Otosan)

The recent heavy duty truck market towards higher power output, lower fuel consumption and reduced emissions. This trend leads challenges in heavy duty diesel engine design due to increasing thermal loads for hot components. One of the most important design feature for heavy duty diesel engines is cylinder head fatigue durability and gasket sealing performance. Cylinder head is subjected to low cycle thermo-mechanical fatigue (TMF) and high-cycle fatigue (HCF) which are driven by combustion process. Gasket sealing performance also has to be considered for avoiding any performance loss resulting from gas leakage. Fatigue life prediction robustness of cylinder heads has reached a high level of confidence past few years; there is still work to increase the reliability of these analyses by improving modeling methodologies and material models. Current study will focus on heavy duty engine cylinder head made of compacted graphite iron development methodology using coupled 3D combustion simulation, conjugate heat transfer and both high cycle fatigue (HCF) and thermo-mechanical fatigue (TMF) analysis methodologies. Also gasket sealing performance is calculated with a proper methodology and it is verified by durability tests. Further strategies such as integration of material property variation and residual stresses into durability analyses. Also, component level rig tests to improve both Abaqus/CAE studies and Design Verification (DV) efficiency are mentioned within the context of this work.

Keywords: Cylinder Head, High Cycle Fatigue, Thermo-Mechanical Fatigue, Engine Gasket Failure

**Prediction of Compressive Characteristic of Hat Section High-strength Steel Frame Taking Bauschinger Effect into Account**
Kenji Takada (Honda R&D)

In recent years, in order to improve both rate of body weight reduction and the safety performance for crash of a vehicle, the application of high-strength steel sheets is increasing. Generally, the Bauschinger effect is strong in high-strength steel sheet. Conventional, material properties used in crash analysis treated as isotropic hardening material. However, the analysis results, average loading and buckling wavelength, deviate between analysis and tests in some case of axial collapse problem of hat section frame that composed of high-strength steel sheet. When an axial collapse occurs in a hat cross section frame, a buckling will occur together with a frame ridge rolling. Bauschinger effect will occur in the steel sheet at both surface and back surface, when the rolling is progress. In this study, in order to perform the axial collapse analysis considering the Bauschinger effect, firstly, a high-strength steel sheet tension-compression test is performed, then, in order to calculate the parameters of the Bauschinger effect, a correlation tool is developed to perform correlation process by using both Isight and Abaqus. It is confirmed that analysis results fit test results in good precision in both Chaboche and Modified ANK models, when the parameters generated by the tool are used.

Keywords: Buckling & Collapse Material Modeling
**The Effect of Rubber in Strength Analysis for Insulator Assembly**
Hoo-Gwang Lee, Ji-Hoon Kim and Sung-Jin Yoon (Hyundai Motor Company)

The insulator bracket assembly in the suspension system supports shock absorber, spring and bump stopper. Previously, the rubber core for shock absorber is enclosed by steel plates and main upward large load is applied to steel plate part directly. However it is simple and robust, it could make some noise. Recently, the rubber core and steel plates are piled up with small overlap and enclosed by rubber at a time to solve this problem. Because only steel plate supports spring and bump stopper load in the previous system, the simulation of permanent deformation by large upward loading from the road wheel can be performed only for steel parts. But, there is small volume of rubber which links rubber core and steel plate in the new insulator system to reduce noise. If the deformation of rubber during large upward loading could be ignored, the simulation would be simple. But the comparison between some test and simulation results without rubber shows quite large difference in stiffness and permanent deformation. The simulation result with finely defined rubber model using Abaqus gives similar force - deformation relation and applied to bracket design improvement. The possibility of fatigue life assessment for insulator rubber using FE-SAFE/RUBBER module would be checked up also.

**Keywords**: Assembly Deformation, Hyperelasticity, Rubber, Strut insulator, Suspension

**Study on Body Rigidity, Strength and Fatigue Performance for Carbon Fiber Reinforced Plastics Vehicles**
Oh Se-Hee and Son Yu-Na (Hyundai Motor Company)

Weight reduction of car is now main issue in order to increase vehicle fuel efficiency. Among various light weight materials, CFRP (Carbon Fiber Reinforced Plastics) is at the head because of superior stiffness and strength to weight ratio. A durability performance development process of CFRP vehicle was studied, and applied to Intrado (HED9 concept car) in order to check compatibility of the process. CFRP material was identified as a solution for satisfying weight lightening and performance of vehicle concurrently by comparing with general steel structure vehicle.

**Keywords**: Composite, Vehicle, Stiffness, Strength, Fatigue

**Parametric Study of Wear of Variable Displacement Vane Pumps Using Finite Element Analysis**
Ju Hwan Lee, Seoung Joon Kim, Tae Gyun Shin, Jeong Soo Lim and Dong Il (Hyundai Motor Company)

Variable valve actuation has become a very popular feature in today’s engines. To better optimize the engine lubrication system, a variable displacement vane pumps has been adapted to the engine and durability of vane pumps is important factor. This study deals with wear of outer ring in the vane pumps. A multibody model has been developed in order to achieve detailed knowledge in the dynamic loads. Also this study considers outer ring movement because of oil pressure, and numerical model is verified with experimental results. Design guidelines leading up to good pump reliability have been obtained as a result of this activity.

**Keywords**: Durability & Fatigue

**Simulation of Pipe Bending Process with Abaqus**
Satoshi Ishikawa (IDAJ Co., Ltd.)
Yoshihiro Ishikawa (SANGO Co., Ltd.)

Exhaust manifolds are generally simple cast iron or stainless steel units which collect engine exhaust from multiple cylinders and deliver it to the exhaust pipe. The exhaust manifolds of steel pipe are formed with multiple bending operations. The pipe bending process is exceedingly complicated and the forming conditions such as controls of pressure are significant. This paper presents several Finite elements modeling techniques for simulating and optimizing pipe bending process using Abaqus/Explicit. Designing and tuning pipe-metal forming tools are quite complicated and time consuming tasks. These tasks must take into consideration a number of potential issues, such as the rate of forming, pipe shape smoothness, reduction ratio of metal sheet thickness, and pipe metal wrinkling. To shorten the design cycle and reduce the number of forming tool prototypes for the multi-process bending pipe, simulations with Abaqus served as virtual test grounds to provide valuable insight into the pipe’s complex forming processes. Because of large deformation and contact interactions, Abaqus nonlinear capabilities were well suited for these tasks. This paper demonstrates Abaqus forming applications that helped resolve matters arising from realistic industrial forming design and production processes. Furthermore this paper argues the difference of three element types which are shell, continuous shell and solid. The analysis results were compared with the experimental thickness and the performance and accuracy were examined.

**Keywords**: Plastic forming, Element formulation, Friction, Explicit

**Development of the Automated Weight Optimization Process for Automotive Wheel Bearing**
Jungho An, Inha Lee, Seungpyo Lee (ILJIN Global Corporation)
Youngmin Cho and Sangkyu Park (Dassault Systèmes SIMULIA)

Automotive wheel bearing is an essential component of the vehicle. It transmits engine power into wheels and supports vehicle weight. In recent years, the demand for weight reduction in automotive parts in order to improve fuel efficiency has world widely increased. Despite of it, automakers are requiring that wheel bearing performance in the areas of stiffness, strength, and fatigue life be maintained or improved. In this study, weight optimization process for automotive wheel bearing was developed and evaluated it onto the real design cases. Design variables were the bearing geometry and bearing specification. The cost function was bearing weight and constraint conditions were bearing fatigue life and stiffness. Abaqus for CATIA V5 was used to build up the geometry and Abaqus was used to carry out the stiffness analysis. To construct the optimization
formulation, the commercial optimization software, Isight was used. As a result, the optimal model satisfied all of constraint conditions and reduces the weight by comparing with that of the initial geometry. In conclusion, by using the developed weight optimization process, the test and development costs of an automotive wheel bearing can be reduced.

**Keywords.** Design Optimization, Durability & Fatigue, Process Automation

**Automated Analysis in Abaqus for CATIA V5**

Stephen Wallis (Jaguar Land Rover)

The traditional process for assessing the strength, stiffness and modal performance of a chassis component such as a suspension link, and iterating the design to meet targets in the optimum way, is time consuming. Some design engineers experimented with using integrated analysis in Abaqus for CATIA V5 to assess and iterate the design more efficiently. Success was limited due to the complexity of setup and poor correlation with the results obtained by Abaqus/CAE specialists using the standardised process. This paper outlines how a combination of Abaqus for CATIA V5 analysis and scripting was used to solve these problems for a number of component types. The success of the tools released to date and the advantages of using them will be discussed. Guidelines for identifying processes which are suitable for automation will be presented.

**Keywords.** Process Automation, CAD Integrated Analysis

**Simulation of Fully Coupled Thermo-Mechanical Effects in a Disc Brake Rotor**

Ganesh Karunakar, Julian Oscroft and Richard Taulbut (Jaguar Land Rover)

The frictional heat generated during braking of a vehicle can cause numerous negative effects on the brake system, such as brake fade, premature wear, thermal cracks causing the brake to have inherent noise and vibration issues under braking, called rumble. An opportunity therefore exists to eliminate brake rumble from the vehicle, thus providing a quality improvement opportunity. In this paper, a fully-coupled, temperature-displacement Analysis using Abaqus/Standard was carried out to take into the account both thermal and structural effects on the brake rotor. The effect on thermal performance of rotational speed (corresponding to V_max braking conditions) and rotor temperature, due to the rotating heat source, was been evaluated. Disc thickness variations are shown to arise from the temperature profile across the disc cheek surface. The scenario simulated considers the thermal effect of braking a vehicle which was prone to a rumble noise. The simulation measured the thermal and geometric changes in the disc, demonstrating the disc thickness variation and thermal banding of the rotor, and the location of these on the disc surface due to the effect of the thermo-mechanical loading. Using this simulation technique, design changes to the pads to eliminate the rumble by moving the hottest band of the disc to a more central location were assessed, and verified on a vehicle. Future developments will include further investigation of the sensitivity of the rumble noise to pad profile and compressibility, and to the internal vane pattern of the disc.

**Keywords.** Noise, Vibration & Acoustics

**Simulation of Combined Forming and Injection Molding Processes**

Timo Mueller (Karlsruhe Institute of Technology (KIT))

Injection molding of fiber reinforced thermoplastic materials is a proven and efficient process to produce lightweight parts for automotive applications. By adding continuous fiber reinforced composites or sheet metal, the possible range of applications can be greatly increased while retaining the advantages of the injection molding process. The reinforcements are formed by the closing motion of the mold and the fluid pressure of the injection molding material. Specialized software is used to simulate injection molding, but to date there is no solution to simulate the combination of forming and injection molding. Abaqus/Explicit is often used for sheet metal and composites forming simulation, but not for injection molding simulation. It is possible to simulate combined forming and injection molding processes using Abaqus Coupled Eulerian Langrangian (CEL) simulation and a combination of Abaqus material models and user subroutines like VUEOS.

**Keywords.** Injection Molding, Compression Molding, In-Mold Forming, Overmolding, Metal Forming, Preimpregnated Sheet Forming, Coupled Eulerian-Lagrangian, Abaqus CEL, Folgar-Tucker, Fiber Orientation, Tait Equation, pvT-model, Abaqus VUEOS

**Application of Predictive Engineering Tool (Abaqus) to Determine Optimized Automotive Rubber Door Harness Grommet Design**

Praveen Mishra and Dagananda Gowda (Mercedes Benz India R & D Pvt Ltd)

A cable grommet is a tube ring through which an electric cable passes. They are usually made of rubber or metal. The cable grommet is used to protect, improve friction or seal cables passing through it, from a possible mechanical or chemical attack and for aesthetic look. It becomes very important to determine the optimum length of the grommet used for particular application. Bigger length will have results into folding of grommet and lesser length will yield into stretching of rubber grommet. Finite element simulation technique presented in this paper can be used to determine the optimum geometry of the grommet. Component used for this study is an automotive door harness rubber grommet. Using Abaqus/CAE hyper elastic material models are evaluated by creating response curve using selected energy potentials. Based on the stability for that particular application hyper elastic model is finalised. Finite element simulation using Abaqus/Explicit helps the designer to optimize the length or material of grommet for that particular application. This gives engineering direction to designer to do necessary changes in the beginning of the design phase freeze which leads into save of time and money.

**Keywords.** Grommet, Hyper elasticity, Optimization, Abaqus CAE, Abaqus /Explicit.
Structural Analysis of an Automotive Forming Tool for Large Presses within Dynamic FEM Simulation using Abaqus/Standard

Krzysztof Swidergal and Marcus Wagner (OTH Regensburg)
Christian Lubeseder and Ingo von Wurmb (BMW Group)
Steffen Marburg (Universität der Bundeswehr München)

In the deep drawing tools for forming car body parts, heavy blankholders are used to prevent buckling and wrinkling of the blank. During each press cycle, those large masses need to be lifted, raising thereby the structural dynamic load on the deep drawing tool and on the press. Therefore a detailed knowledge about the blankholder’s lift-off event is essential for an accurate and robust design of forming tools. In this paper, a dynamic finite element method (FEM) simulation of blankholder’s lift-off in a selected automotive deep drawing tool is presented enabling identification of regions of critical stresses. The FEM model is created/developed within the Abaqus/CAE environment and solved with the Abaqus/Standard. Each dynamic analysis is preceded by a static analysis where the gravity load is applied and the lifting bolts are prestressed. A special emphasis is put on modelling the elastomer dampers, which are installed between lifting bolts and blankholder to avoid hard impacts during the lift-off event. Those dampers are modelled using a hyperelastic material with hysteresis. In addition, an experimental validation of a blankholder’s vibration under operating loading was carried out. The simulation results are in good agreement with the measurements.

Keywords: Blankholder, Deep Drawing Tool, Elastomer Damper, Vibration, Mechanical Press

Strength Assessment of Injection Molded Short-fiber-reinforced Plastic Components

Wolfgang Korte, Marcus Stojek, Sascha Pazour (PART Engineering)

The use of short-fiber-reinforced plastics takes place in many different industries. In particular, in the automotive industry short-fiber-reinforced plastics are increasingly being used as the preferred engineering plastic. This is due to the excellent mechanical and thermal properties of these materials compared non-reinforced grades. The use of these materials enables the manufacturer of automotive components to a significant weight reduction compared to metallic materials. Since components of short-fiber-reinforced plastics are stressed highly both mechanically and thermally an intelligent component design is required in order to fully exploit the potential of these materials. Hence, the design of such components must be based on a reliable strength assessment. For this purpose models for the description of the anisotropic and elasto-plastic failure behavior of short-fiber-reinforced plastics are required. In contrast to the widespread use of short-fiber-reinforced plastics, methods for a reliable strength assessment based on FE analyses for components made of these materials have not been sufficiently developed yet. This paper presents an approach for the strength assessment of plastic components based on FE analyses. In the scope of this, appropriate failure limits and failure criteria for these materials are presented.

Keywords: Anisotropy, Failure Criterion, Injection Molding, Polymer, Plastics, Short-Fiber-Reinforced Composites, Tsai-Hill, elasto-plastic

Simulation Driven Design for Manufacture of a Pressed Composite Automotive Tailgate

Andy Ngai and Mark Arnold (Penso Consulting)

A current key automotive industry initiative is delivery of mass savings through the use of lightweight materials, to reduce vehicle CO2 emissions. Penso were commissioned by Jaguar Land Rover to design and manufacture a continuous fibre reinforced composite tailgate prototype for the Jaguar F-Type. The new carbon fibre design had to meet all existing strength and stiffness targets, whilst maintaining existing A-surface geometry and all hardware. The three main parts of the tailgate considered for light weighting were the tailgate inner panel, tailgate outer panel and spoiler mechanism carrier. An optimised laminate design was developed by the engineering team using a non-linear static FE analysis performed with Abaqus/Standard. The Tsai-Wu failure index and a user defined inter-laminar failure criterion were used as design metrics in post processing. The design was then evaluated with Penso’s composite manufacturing team to develop preliminary ply shapes and overlap joint locations suitable for manual draping and Penso’s pressed composite technology. These ply boundaries and material directions were updated in the FE model for further refinement of ply joint locations, prediction of finalised mass/performance and generation of layup manuals. The A and B-surfaces of the parts were then updated within CAD and used to create tool geometry for pressed composite. The final design of parts gave a combined mass save of over 65% whilst having comparable structural performance to the production design.

Keywords: Carbon Fibre, Composites, Design Optimization, Failure, Finite Element Analysis, Laminate, Manufacture

Brake System Model Reduction for Squeal Noise Study

Samuel Nacivet and Olivier Fazio (PSA Peugeot Citroën)
Jean-Jacques Sinou (Ecole Centrale de Lyon)

Although the problem of friction-induced vibration has been the subject of many investigations over recent decades, it is still responsible for a large number of nuisances in the field of automotive. Using Abaqus for CATIA V5 (RAC) and Abaqus, this study presents a numerical process based on modal reduction to generate a superelement for the whole brake. The size of the model is minimized using a specific non-linear modeling at the frictional interface. Then, the superelement is used in matlab to check is ability to predict the stability analysis of large finite element models that correspond to real automotive braking systems subjected to friction-induced vibrations. The effect on squeal instabilities (i.e. the frequencies and the associated unstable modes) of the number of contact nodes at the frictional interface is investigated in order to reach a high-quality estimation with the smallest number of DOF possible. To finish, what could be improved in Abaqus for CATIA V5 and Abaqus to make the superelement more representative and easier to create is discussed.

Keywords: Vibration, Friction, Squeal, Reduction, Super-Element.
Application of Coupled Eulerian Lagrangian Approach in Finite Element Simulation of Friction Stir Welding

N. K. Sanjeev and B. P. Ravikiran (Robert Bosch Engineering and Business Solutions Limited)

Finite Element Simulation of Friction Stir Welding (FSW) is a problem involving large deformations and is often difficult to solve using the classical Finite Element Method (FEM). Large mesh distortions and contact problems can occur due to the large deformations such that a convergent solution cannot be achieved. Since in Abaqus, a Coupled Eulerian Lagrangian (CEL) approach has been developed to overcome the difficulties with regard to FEM and large deformation analyses. In this article, this method is investigated regarding its capabilities in simulating FSW process. FSW is a mechanical process whereby solid-state welding is performed using heat generated from the friction of a rotating tool and plastic deformation of weld material. FSW is a modification of the traditional Friction Welding, which was invented at the Welding Institute in Cambridge, England in 1991. Since then, it is gaining significance in many joining applications, by overcoming the limitations of other fusion welding processes. For successful incorporation of its potential during industrial applications, mechanism of joining needs to be properly comprehended. Various experimenting and study techniques are used in widening of its applicability in joining process. FE Simulation is one such technique which would help in getting an insight of the process phenomena during the process and the overall result could be used to observe the effect of process parameters on weld quality. Here, an attempt is made to develop a FE model using Abaqus/Explicit with help of CEL formulation, Johnson-Cook material law and Coulomb's Law of friction. The model has been found to be capable of obtaining certain insight on right processing conditions prior to performing welding process, by predicting the effect of process parameters on outputs like temperature, force on tool and formation of defects. Finally the FE Simulation results are used to obtain the optimized process parameters.

Keywords: Friction Stir Welding, Finite Element Simulation, Coupled Eulerian Lagrangian, Abaqus

Innovative Anisotropic Material Modeling Approach for Fiber Reinforced Thermoplastics

Recep Yaldiz (SABIC)

Gertjan Kloosterman (Dassault Systèmes B.V. 3DS/SIMULIA)

Current industrial state of the art for predictive engineering of fiber reinforced thermoplastic materials utilizes nonlinear isotropic material modeling and/or anisotropic linear modeling. Either approach gives inaccurate predictions, since the effect of the orientation and length of the fibers is not taken into account in the former, whereas the latter omits the plastic behavior typically observed in thermoplastics. An innovative approach is developed in the last years which takes into consideration the anisotropic nonlinear behavior of the material. This approach which is based on micromechanical homogenization theories that considers both correct processing and fiber orientation, can significantly improve the accuracy of mechanical predictions, but is unfortunately very costly in terms of analysis time. The authors follow the innovative approach of combining the processing conditions of the fiber reinforced materials with obtained fiber length and orientation in the end applications, but couple this with a novel material model which incorporates anisotropic non-linear elasticity, anisotropic hardening plasticity, and multilayer anisotropic non-linear visco-elastic behavior which can support anisotropy ratios that would cause a numerical breakdown when the Hill yield criterion would be employed. Since thermoplastics have complex fracturing behavior, the material model also incorporates a tailor made damage initiation/damage evolution process. Influence of the processing conditions and fiber orientation on mechanical properties will be elaborated, as well as how the model was calibrated to the large amount of test data available within SABIC.

Keywords: Thermoplastic Materials, Anisotropy.

Topology Optimization of a Motorcycle Swing Arm Under Service Loads using Abaqus and TOSCA

Roberto Saponelli (Sacmi/Protesa)
Massimo Damasio (Exsemplar)

In research and development environment and concept design, people involved in new projects often need to design a completely new shape for the structure target of the analysis. The loading conditions and constraints are usually known but the designer hardly knows how to create the geometry of the structure that meets the requirements of the project and that can be manufactured at the same time respecting the target costs. The weight is known to be one of the main factors that impacts on the performance and on the costs, and for this reason its reduction often becomes the main objective to achieve. Often geometries are realized based on experience or on similarity to previous projects, but this approach can be long and expansive, without the certainty of being able to reach the optimum. The use of the Abaqus/CRE (Computer Rided Engineering) is strategic in this environment, in order to verify and virtually simulate different types of geometries avoiding the cost and time of prototyping and testing. A FEM model is the starting point for the optimization process and provides, where appropriate, the subsequent integration of the results with different physical disciplines, such as CFD or Heat Transfer. In this paper is presented the topology optimization of a motorcycle swing arm, starting from a design space which takes into account the overall dimensions, and trying to minimize the mass while maximizing stiffness. The use of TOSCA, that drives the FEM simulation of the swing arm performed with Abaqus, allowed to obtain a shape that can be manufactured in compliance with the requirements of the project.

Keywords: Topology optimization, Nonlinear Analysis, Motorcycle, Swing arm
Consideration of Manufacturing Pre-history Influence of Fatigue Life of Sheet Metal Formed Automotive Parts

Dmitry Buzlaev (TESIS)
Viktor Kalaev and Dmitry Zheleznyakov (GAZ UEC)
Anders Winkler (Dassault Systèmes)

This presentation considers the effects of sheet metal forming processes on the fatigue performance of LCV wheel discs, manufactured from microalloyed steel. Traditionally such parts have to pass standard cyclic load test, but road tests in real life condition sometimes show fatigue cracks in highly stressed areas in predominant bending loading mode. To estimate fatigue life and safety margins standard fatigue tests should be accomplished by numerical simulations based on FEA results. Unfortunately, in spite of long history of fatigue analysis codes, based on FEA results, the industry still heavily relies on tests because of often disagreements of simulation and test results. In most cases stress analyst does not take into account manufacturing pre-history of the part, its thinning, plastic strains and residual stresses accumulated in a part as a result of sheet metal forming process used for its manufacturing. This study tries to fill the gap between sheet metal forming simulation tools, like Stampack, SIMULIA Abaqus as stress analysis tool and SIMULIA fe-safe as a fatigue analysis tool. It should significant difference between the original simplified model of wheel centre disk, which used virgin material and did not take into account real disk geometry and residual stresses and hardening, and the model which used results from sheet metal forming simulation as a starting point.

Keywords: Bolt Loading, Wheel Disk, Fatigue Life, Deep Drawing, Metal Forming

Development of a Multifunctional Truck Cross Member Made of Continuous Fiber Reinforced Plastics

Herwig Kirchberger (TEUFELBERGER Composite Ges.m.b.H.)
Stefan Fischmeister and Helmut Dannbauer (Engineering Center Steyr GmbH & Co KG)

Lightweight design of trucks enables to increase the payload and represents a way to reduce the fuel consumption per transported weight. Many engineers around the world are focusing on this topic. One idea to handle this issue is to combine several vehicle parts or functions in an integrated module. This contribution shows the development of a cross member made of a combination of continuous fiber reinforced plastic and steel. Beside the function as an important structure of the truck main frame, the part represents an air reservoir at the same time. Beside the management of design issues, the assessment regarding stiffness and strength was the major focus of the project. By means of FE simulations, done with Abaqus, the structure was optimized regarding manufacturing possibilities in combination with the other essential properties like stiffness and strength. Many activities were set to find an appropriate combination of the plastic area and the steel brackets, which enable the fitting within the truck main frame structure. Due to the fact that the function integrated structure is much stiffer (resulting from the bigger size and cylindrical shape) than the original cross member, the specific decoupling of the truck main frame and the new developed module was challenging. After solving the main design problems the development team focused on the optimized layout of the plies with respect to stiffness and durability. The necessary analysis will be presented in detail. Last but not least the lecture will compare the results of the simulated loadcases with tests done in our facility.

Keywords: Composites, Stiffness & Strength, Optimization, Commercial Vehicles

Improvement Studies After Misuse Analysis of the Engine Hood and Correlation with Physical Tests

Tamer Aydiner and Halil Bilal (Tofaş Türk Otomobil Fabrikası A.Ş.)

Nowadays, there are highly necessary to investigate possibility of various problems caused by drivers. In this study, we focused that to avoid any damage on lock and vehicle body after misuse of bonnet. When the bonnet is released at full open position, it closes with high velocity thus occur very high forces on body and lock zones. Under this conditions, it is deserved to avoid crashing another components at neighborhood and occurring any damage affecting bonnet function. Accordingly, it was provided a correlation by doing physical test and CAE analysis on existing commercial vehicle to provide accommodating physical tests and FE model to be used in newly developed light commercial vehicle project. FE model compatible with physical tests was adapted to new project and related analysis with Abaqus/Explicit has been done therefore it was resolved some negativities which were seen after the analysis. Subsequently, there were seen a highly correlation between Abaqus/CAE model and physical tests.

Keywords: Engine hood, Misuse, Abaqus/Explicit, Explicit Dynamic Analysis, Topography Optimization, Correlation
INDEX

4a  48

ABB Turbo Systems AG  44
Abedrabbo, N.  44
Aernnova Engineering Division S.A.  31
Airbus Defence and Space  31
Airbus Group  31
Airbus Operations Ltd.  31
Airbus Operations SAS  32
Al-Khalil, M.  31
Alotta, G.  47
ALPLA Werke Alwin Lehner  38
Alvares de Castro e Sousa  42
An, J.  51
Ansaldo Energia  39
APPLUS IDIADA  48
Arbiol, J.  48
Arndt, S.  39
ATA Engineering, Inc.  32
Autoliv India Pvt. Ltd.  49
Aydiner, T.  55
Baker Hughes  39, 40
Bäker, M.  35
Bakir, M.  45
Barrera, O.  47
Basciano, C.  47
BD  47
Belles Meseguer, A.  34
Bello-Gomez, M.  46
Birkmose, M.  48
Brusa, E.  45
BTU Cottbus–Senftenberg  32
Buzlaev, D.  55
CEA/DAM/DIF  32
C-FER Technologies  40
Charnaux, C.  43
Chiminelli, A.  34
Coskun, V.  34
Cousin, L.  32
Crespo, M. J.  37
Dana Holding Corp  49
Dasari, G.  41
Dekker, A.  39
Dong, Y.  37
Dow Chemical  38
Dow Europe GmbH  50
DTU Wind Energy  41
DuPont Engineering Technologies  50
Essa, Y.  31
ESTECH Corporation  36
ExxonMobil Upstream Research Company  41, 42
F
Fedulov, B.  35
Fihri Fassi, H.  36
Fokker Landing Gear BV  33
Ford Otosan  50
FSTS  36
G
Garzon, J.  41
Gerlach, T.  42
Goller, B.  33
Greene, Tweed & Co.  33
Guzel, A.  50
H
Heimbs, S.  31
Hitachi, Ltd.  45
Honda R&D  50
HPC Advisory Council  44
Hyundai Motor Company  51
I
IDAJ Co., Ltd.  51
ILJIN Global Corporation  51
INTALES GmbH  33
Ishikawa, S.  51
ITAINNOVA  34
J
Jaguar Land Rover  52
J P Kenny  42
K
Kajima Corporation  37
Karlsruhe Institute of Technology  52
Karunakar, G.  52
Kilic, H.  33
Kipouros, T.  55
Kobayashi, T.  38
Korte, W.  53
Kostov, N.  42
Kujawski, D.  36
Kumar, V.  49
L
Lee, H.  51
Lee, J. H.  51
Leibniz University Hanover  42
Liu, S.  42
Lui, P.  44
M
Maas, J.  39
Mechanical Design & Analysis Corporation  38
Mercedes Benz India R & D Pvt Ltd  52
Mercedes-Benz Türk A.Ş.  45
Mishra, P.  52
Momii, H.  36
Moscow State University  34
Mueller, T.  52
N
Nacivet, S.  53
Ngai, R.  53
Nikaido, Y.  37
Ping, N.  41
N. K., S.  54
O
Olveda, D.  46
OTH Regentsburg  53
P
PART Engineering  37, 53
Pavlov, M.  35
Pazour, S.  37
Penso Consulting  53
Pinna, D.  39
Politecnico Di Torino  45
Polipelas, F.  49
Prime Aerostructures  34
Princípio  37
PSA Peugeot Citroën  53
R
Ramirez, E.  46
Rebien, H.  49
Reithofer, P.  48
Revels, N.  32
Rickhey, F.  46
Ritsumeikan University  48
Rivero Arevalo, L.  31
RMV Consultoria  42
Robert Bosch Engineering  54
Roketsan Missiles Inc.  34
Roma Technology  43
Rozin, N.  35
S
SABIC  54
Sacmi/Protesa  54
Saponelli, R.  54
Se-Hee, O.  51
Shanghai Turbine Works Company Ltd  43
Siddiqui, M.  50
Simpleware  47
Singapore-MIT Alliance for Research and Technology  37
Skolkovo Institute of Science and Technology  35
Smurfit Kappa  39
Soendergaard GN Resound A/S  48
Sogang University  46
Sosa, E.  38
Stellinger, E.  32
Subsea 7  43
Swiderski, K.  53
T
Takada, K.  50
Tan, S.  43
Technische Universität Braunschweig, Institut für Werkstoffe  35
Technische Universität Darmstadt  46
Terasaki, T.  45
TESIS  55
TEUFELBERGER Composite Ges.m.b.H.  55
Teufel, P.  44
Tofaș Türk Otomobil Fabrikasi A.Ş.  55
Torres, J.  43
Trek Bicycle Company  39
TsGRI  35
U
UANL  46
Universidad Nacional Autónoma de México  46
University of Cambridge  36
University of Oxford  47
University of Palermo  47
V
Valtiner, O.  38
van der Veen, S.  32
van der Zee, W.  40
Vershinin, V.  34
Volgers, P.  50
W
Wallis, S.  52
Wang, Z.  48
Weatherford  44
Western Michigan University  36
West Virginia University  38
Wilson, W.  33
Wocke, C.  38
X
Xie, J.  40
Y
Yaldiz, R.  54
Young, P.  47
Z
Zharenov, I.  35
INTEGRATED SOLUTIONS ABSTRACTS

A Study of the Flow Induced Vibration of a JUMPER Due to Multi-Phase Internal Flow with an Objective of Understanding Life-Cycle Fatigue
Dr. Alan Mueller (CD-adapco)

A study was conducted to study flow induced vibration due to an internal multiphase flow. The approach uses the Abaqus co-simulation feature in STAR-CCM+ to fully couple the fluid dynamics of the internal VOF flow to the Jumper structural mechanics. This study focuses on the internal flow and assumes the Jumper external flow is quiescent. Several different simulations were conducted. a. The simulation of the internal VOF flow in a rigid Jumper to study the major fluid dynamics modes. b. An eigenmode analysis to examine the modes of vibration in a quiescent internal and external fluid. c. A two-way coupled simulation to understand the response frequency and damping in the structure due to the internal VOF flow. d. A one-way coupled simulation where the fluid loads from a. are used to study the FIV in the structure. The consequences of various assumptions regarding “added mass and added damping” applied in the one-way coupled solution may then be evaluated by comparison to the fully coupled solution particularly in terms of fatigue.

Adding Electromagnetics to your Multiphysics Simulations
Dr. Irina Munteanu (CST)

Computer Simulation Technology AG (CST), with headquarters in Darmstadt, Germany, was founded in 1992 and has quickly become one of the world’s most renowned providers of software in the domain of 3D simulation of high-frequency electromagnetic fields. In Europe, CST is the largest software provider with focus on the 3D electromagnetic simulation. The main software product of CST is CST STUDIO SUITE. It includes 3D simulation engines in time and frequency-domain, specialized solutions for Printed Circuit Boards (PCB) and cable harness electromagnetic simulations, as well as an electric circuit simulator. A large variety of applications can be modeled and simulated using CST STUDIO SUITE, such as mobile phones, electronic devices, electric motors, eddy current-based devices used in induction heating, or the influence of electromagnetic fields on the human body. However, in today’s world, the design of a device needs to cover different engineering areas – e.g. not only electromagnetic, but also thermal, structural or acoustic. The strong interaction of these apparently disjoint domains often requires a coupled simulation in order to reach the required accuracy. The collaboration between CST and SIMULIA aims at providing simulation specialists with enhanced co-simulation capabilities for such types of applications. The talk will present an introduction to the electromagnetic simulation world, its capabilities, challenges and areas of application, with a special focus on multiphysics applications.

Complete Laminated Composite Modeling and Simulation Process Using the ANSA and µETA Software Tools for the SIMULIA Abaqus Solver
Dr. Ioannis Nerantzis (BETA CAE Systems SA)

This presentation demonstrates the process of modeling, solving, and evaluating the results of laminated composite structures using ANSA, µETA and the Abaqus solver. The composites modeling workflow often requires the use of a multitude of software tools, each aiming at diversifying fields concerning design, manufacturing and CAE. The common problem that arises, when laminated composite products are developed, is that various file formats and information, stored in complicated data structures, need to be communicated across all software tools. As a result, the required workflow for a simple modification of an existing model usually consists of many steps. This increases modeling time, process complexity, and the risk of making mistakes. The interoperability that ANSA and µETA provides by supporting various file formats and modeling tools, allows for significant reduction of processing time and overall workflow complexity. A laminated composite model can be created or imported in ANSA and handled with the Laminate tool. Laminate data can be created, imported, and modified using sophisticated tools that extend to various concepts. Digital mockup, draping simulation, mapping of laminate data and merging of zones are some of the composites oriented tools that ANSA offers. When the procedure is coupled with the Abaqus solver a robust set of results can be achieved. Results can be calculated at multiple section points through the thickness of each layer providing a detailed overview of the structure’s behavior. The CompositePost toolbar in µETA consolidates all needed tools to post process laminated composite results. Failure indices and reserve factors according to standardized methods can be calculated directly by µETA. This makes possible the re-evaluation of the results for different material limits without the need to solve the model again thus reducing time, effort, and disk space.

Dr. Andrey Aksenov and T. Luniewsky (Capvidia)

FlowVision is a universal CFD code for solving wide range of industrial, medical and life science applications. Based on the fluid volume definition, FV offers fully automatic mesh generation with dynamic grid adaptation. The integration with Abaqus/SIMULIA through Direct Coupling Interface (DCI) and Co-Simulation Engine (CSE) provides solution to the most complex Fluid-structure–interaction simulations. Automatic matching of the CFD and FEA and use of the Abaqus FEA mesh as CFD boundary simplifies the set-up of the project and provides controlled accuracy in data exchange between structure and fluid domains. Through support of the CSE engine the implicit coupling between Abaqus and FlowVision is realized, which significantly extends co-simulations capability to support solution for the most complex problems such as e.g. human heart beat simulation (Living Heart Project). The presentation will address above mentioned topics and illustrate practical use of Abaqus/FV co-simulation to solve problems from aerospace, nuclear industry and life science.
boots with different design features (material, fillet radius, wall thickness, and preloading in compression or tension) have been
provides these functions, containing the lubricant and keeping out debris. The boot can fail from fatigue, leading to issues
Constant velocity joints on front-wheel drive vehicles require lubrication and protection from contamination. A rubber CV boot
Fatigue Analysis of an Elastomeric CV Joint Boot Cover with fe-safe/Rubber

Huge and complex results data, and now? EnSight, Postprocessor for Scientific and Engineering Simulation Data
Mr. Arnold Holler (CEI)
Looking at the evolution of numerical simulation in the industry we currently are in an exciting phase. Model size grew
dramatically over the last few years, fortunately hardware performance did as well. Today, methods and algorithms allow to
run complex simulations or even multi-disciplinary analyses. Users have high expectations regarding the engineering value of
numerical analysis. Simulation results were only available in column of figures in the beginning of numerical simulation.
Nowadays, this would not be enough to satisfy the demand of postprocessing complex results of huge simulation models.
That’s why in the process chain of numerical simulation good postprocessing tools are equally important than solver
technologies. Cray Research recognized this importance already quite early and began to develop a postprocessor for scientific
and engineering simulation data. In 1994 CEI was founded to focus on further development of this high-end postprocessor,
EnSight. Out of the headquarters in Apex (NC, USA) CEI meanwhile acts globally with offices in China, Germany, India and
Japan. In this presentation features of EnSight are outlined (e.g. parallel processing and rendering, reading multiple analysis
cases from different disciplines, support of advanced 3d display technologies, …) which enable to analyze, visualize and
communicate simulation results of complex multi-disciplinary analysis cases, such as ABAQUS implicit, explicit or CFD, and
meet postprocessing-expectations of modern engineers.

FTire, the Virtual Tire in MBS and FE Environments: Physics Beats Magic
Mr. Gerald Hofmann (cosin)
FTire (Flexible Ring Tire Model) belongs to the class of strictly mechanics-based tire models, suitable for use in general vehicle
dynamics simulations. FTire development started in 1998, using certain ideas and numerical concepts of the ‘coarse-mesh’ FE
model DNS-Tire (Dynamic Nonlinear Spatial Tire Model), as well as the nonlinear ‘rigid-ring’ model BRIT. FTire’s complexity is
below that of detailed FE models, but far above classical ‘point contact’ models. Consequent use of mechanically consistent,
highly non-linear structure and friction models allows ‘safe’ extrapolation into operating conditions not covered by respective
laboratory experiments. FTire returns plausible dynamic tire forces even at multiple high-frequency excitation, caused by road
height profile and deformation, friction variation, suspension vibrations, drive and brake torque, tire non-uniformity and
imbalance, temperature and pressure variation, and misuse events. This is not achieved by overly compromising computing
time. Depending on activation of subsystems and on timely and spatial resolution, FTire simulation only takes about 1 to 50
times real-time. Due to FTire’s multi-core support, all tires of a vehicle can be simulated in parallel at the same computation
speed. FTire constitutes a full tire simulation environment. More than just a single model, it provides a scalable tire model kit,
ranging from parallelized, real-time capable versions for hardware-in-the-loop application, up to high-resolution realizations,
connected to explicit FE solvers. Upon demand, FTire provides a tread pattern, a tread temperature distribution, and a tread
wear model, as well as visco-elastic rim and road models. Assisting tools are available for editing the model data file, for
parameterization and data fit, for static, steady-state, and modal analysis, for visualization, for linearization, for DOE studies,
for model export, and more. Parameterization may be based on laboratory measurements, on tire design data, on similarity
considerations, or on combinations of these. Using numerically robust co-simulation, FTire is made available as tire model
plug-in for most of the relevant commercial simulation environments, covering MBS, FE, specialized vehicle dynamics, and
system simulation approaches. It has been used together with nearly all rubber-tired types of vehicles, including passenger
cars, light and heavy trucks, motorcycles, scooters, aircrafts, all-terrain vehicles, and more. FTire applications comprise primary
and secondary ride, handling on flat and uneven road surfaces, tire forces influenced by suspension control systems, NVH,
mobility, tire-imperfection induced suspension and steering vibrations, misuse, and road load prediction for durability.

Fatigue Analysis of an Elastomeric CV Joint Boot Cover with fe-safe/Rubber
Dr. William Mars (Endurica)
Constant velocity joints on front-wheel drive vehicles require lubrication and protection from contamination. A rubber CV boot
provides these functions, containing the lubricant and keeping out debris. The boot can fail from fatigue, leading to issues
with the joints. Using ABAQUS to model the boot in operation and the fe-safe/Rubber fatigue solver to calculate fatigue life, CV
boots with different design features (material, fillet radius, wall thickness, and preloading in compression or tension) have been
analyzed. The operating cycle is taken to be one rotation of the boot while it is bent at a specified turning angle. Of the cases analyzed, the design giving maximum fatigue life is made out of polychloroprene 50 HRF, with a fillet radius of 1mm, a wall thickness of 2mm, and a compressive pre-displacement of 5mm. The fe-safe/Rubber fatigue solver has enabled the CV boot design space to be thoroughly explored for fatigue performance, using numerical simulation of possible changes in material, geometry, and loading. Since qualifying fatigue performance is often the most costly phase of the development process, fe-safe/Rubber represents a significant opportunity to save iteration costs and avoid late-program risks.

**MpCCI – Multiphysics Interfaces**  
Mr. Klaus Wolf (Fraunhofer Institute SCAI)

Fraunhofer SCAI is an independent provider of vendor-neutral interfaces for bi-directional co-simulation as well as uni-directional mapping. Fraunhofer SCAI collaborates with worldwide leading software vendors: ANSYS Inc., CD adapco Group, Cedrat SA, Creo, CEI, Dassault Systèmes Simulia Corp., Infolytica, JSOL, MentorGraphics, MSC, NUMECA Intl., SIMPACK AG, and ThermoAnalytics Inc. The MpCCI interface platform is used in various application areas: Abaqus can be coupled with ANSYS/Fluent or OpenFOAM to model fluid-structure-interactions and aero-elasticity problems. MpCCI offers iterative transient FSI and adaptive under-relaxation capabilities. Abaqus in combination with FINE/Turbo or ANSYS/Fluent can be used to calculate thermal stresses in turbo-machinery systems. MpCCI allows non-matching periodic models in FEA and CFD. The FSI Mapper provides schemes to map pressure excitations (calculated with FINE/Turbo’s harmonic methods) to Abaqus models for flow induced vibration analysis of turbine blades. The co-simulation between Abaqus and MSC Adams or SIMPACK allows for the integration of non-linear component behavior in vehicle and system dynamics applications. Combining e.g. SIMPACK and 3rd party CFD-codes like Fluent, OpenFOAM or STAR-CCM can be used to model cross-wind effects on driving cars. Manufacturing simulation results from LS-Dyna, PRM-Stamp or Autodesk Autoform can be transferred as local model properties (local thickness, stresses, material properties) to Abaqus input decks. A prototype solution to transfer properties from CFRP manufacturing processes (draping, molding) has been tested successfully. The quality thermal stress models for PCB and IC components can be improved by incorporating mapped thermal results from a preceding cooling simulation in FloTHERM.

**Managing Large Abaqus Simulation Results for Efficient File Transfer and Storage**  
Mr. Christoph Kaulich (GNS)

Reducing the Ave. Element size in up-to-date FEA models in the automotive industry leads to very large models that often exceed 25 million elements. Such models in the end lead to very large simulation result files. And the increasing reliance on performing computations in far-away countries that have cheaper green energy resources means that a lot of data has to be transferred across networks and stored. To keep the simulation workflow smooth, and as responsive as with a local cluster, the transfer times have to be short and the file sizes have to be reduced. GNS mbH software tools such as Animator4, Animator Database, and FEMZIP-R4DB are proven solutions for reducing the ODB file sizes generated by Abaqus simulations.

**Materials Gateway 3.0 for Abaqus/CAE – Enterprise-wide Materials Knowledge for Abaqus Users**  
Sebastian Schwaegele (Granta Design)

CAE engineers need quick and reliable access to approved CAE materials models – and the implications of incorrect or inappropriate materials data on the validity of simulation results can be profound. But obtaining this information can be challenging and time-consuming for CAE engineers – especially for more complex non-linear and high strain-rate analyses. Their managers, meanwhile, require valid, traceable simulation results, and consistency in CAE approach across the company. For over 12 years, leading engineering organizations worldwide have been working together in a pre-competitive industry and government collaboration (see http://www.mdmc.net/), defining and evolving best practice in materials data management. The resulting software supports managed repositories of validated and approved data, together with the tools to provide enterprise access to end-users in Design and CAE, in the specific formats that they need. Long-time SIMULIA partner Granta Design is the software contractor to the initiative. This presentation introduces Granta’s Materials Gateway 3.0 for Abaqus/CAE, a powerful plug-in-application enabling Abaqus users to access their company’s managed materials knowledge – as well as Granta’s comprehensive library of authoritative reference data on metals, plastics, ceramics, composites, and natural materials. The presentation will report on approaches taken to challenges such as defining the optimum schema for representing CAE materials model information, providing update notification to CAE users when source data is updated; providing version control of the transforms used to derive materials models from database attributes – in the context of accessing this functionality and traceability without needing to leave the familiar Abaqus/CAE environment.

**J MAG: Simulation Software for Electromechanical Design**  
Thiebaud Pfister (Powersys)

J MAG is software for electromechanical design and has an excellent track record in industries since 1983. Electromagnetic field plays important roles with other physics in many machines and equipment and is becoming more actively used in those designs adapting the trend of electricifications. In many cases our customers have demanded that these analyses must go beyond a single domain analysis and include structural and thermal performance. This is where Co-Simulation with Abaqus/J MAG or Star-CCM+/J MAG are necessary to provide the highest fidelity to actual machine performance. J MAG is enhancing its functionality through a partnership with SIMULIA.
How SIMULIA Products Accelerate Composite Design Process

Mr. David Da Silva (KEONYS)

At the heart of the digital company, KEONYS has been a preferred PLM (Product Lifecycle Management) and 3D solutions partner to customers of all sizes and in all industries, to develop and manufacture better products and innovative services. In the framework of its participation in the SIMULIA Community Conference, KEONYS is pleased to present you a simulation offer about Composites Design. The added-value of KEONYS’ offer lies in an integrated approach of Composite Design based on simulation. This offer covers all of the Composite Design Cycle from Preliminary Design to Detailed Design:

- The preliminary approach allows users to make Composites Technological choices (laminate, sandwich, etc.) based on Simulation results (Linear Analysis and Modal Analysis).
- The next step is to simulate a more detailed model taking into account the deviation of the ply orientation due to the manufacturing process. This step is still integrated into the design tools (CATIA).
- The optimization tools, which can be used during any step of the process, allow the user to improve Composite Design (ply number, ply orientation, ply definition, etc.).

This presentation will cover the entire methodology and advantages of a fully integrated process as well as highlight the different user profiles who work on this project.

Using Abaqus for Fatigue Analysis of Components with Large Displacements (e.g. rubber)

Mr. Rael Werkhausen (Magna Engineering Center Steyr GmbH & Co KG)

The Engineering Center Steyr developed FEMFAT from an internal procedure to handle and assess stress results for fatigue and safety factor analysis to a commercial software, which is worldwide accepted and used in automotive business same as other mechanical engineering services. It supports multiple interfaces but Abaqus interface is one of the best approved ones as 25 consulting engineers use it in their daily work right beside the development and support team. Maybe this is also the reason why the difficulties in analysis procedure for very long time, nearly stochastic loading and large displacements were subject of investigations. By an extra tool called ELASTOLOADS the 3 dimensional path of a rather large displaced in combination with inner pressure or torsion is discretized and made easy for accessment. How? My presentation will show.

Faster Innovation - Accelerating SIMULIA Abaqus simulations with NVIDIA GPUs

Mr. Baskar Rajagopalan (NVIDIA)

GPUs have been accelerating graphics for decades but it is now common to see use of GPUs as CPU computational accelerators for all major server manufacturers. 2-3x gains in wall clock time for Computational Structural Mechanics are regularly seen when adding GPUs to workstations and servers. Attend this session and learn the techniques for improved acceleration that result in reduced energy consumption in Workstations & Clusters/Servers. You will also learn how to improve productivity by accessing simulation data anytime, anywhere with virtualized desktops powered by NVIDIA GRID.

S-Life - Integrated Static and Fatigue Strength Assessment with Abaqus According to German FKM Guideline

Dr. Wolfgang Korte (PART Engineering GmbH)

The software S-Life is an integrated solution for Abaqus users in order to conduct an automatic static and fatigue strength assessment according to German FKM guideline. The FKM guideline is, at least in German-speaking countries, the de-facto-standard regarding strength assessment of machine parts made of steel, cast iron and aluminum. In contrast to competitive fatigue solvers which implement a variety of fatigue assessment approaches, S-Life focuses solely on the FKM guideline. Hence, the user interface is straight forward and easy-to-use even for the occasional user who does not need to have a profound understanding of fatigue assessment theories. Basically a linear-elastic FEM analysis is needed in order to conduct the assessment. S-Life reads the local stresses directly from the Abaqus odb-file. Subsequently S-Life processes these stresses according to the FKM code and provides as result the static and cyclic utilization ratios of the component as contour plots. Additionally for each node a numerical report of all variables according to the FKM computation scheme can be requested. The complete procedure is done solely in post-processing. In the case of multiple cyclic loads applied on the component, S-Life computes automatically the most critical load case combination. In the included material database strength properties of more than 1600 material grades are provided. User materials can be created as well. PART Engineering GmbH is an engineering services and software supplier. As an engineering firm PART supplies on- and off-site support in terms of FE simulation services as well as software tools in order to improve the efficiency and accuracy of FE analysis of its customers.

Innovative Solutions for Complex FEA Problems

Mr. Scott Shaw (SGI)

SGI is focused on helping SIMULIA customers solve their most demanding challenges by delivering high performance computing, server, storage, data management solutions. SGI develops, markets and sells a broad line of low-cost, mid-range and high-end scale-out and scale-up servers and data storage solutions, as well as differentiating software solutions on SGI platforms. In the HPC industry SGI is recognized as a trusted leader in technical computing and has over 25 year partnership with SIMULIA.
Technologies will show theoretical background, examples and demonstration of their integrated solutions for understanding loads and linear dynamics results are challenges in most engineering problems. Dr. Tim Hunter (Wolf Star Technologies)

Load Measurement and Linear Dynamics with Wolf Star Technologies

In the second part of our talk some examples of application of our products for the solution of industrial tasks in the field of thermomechanical fatigue will be presented.

Understanding and interpreting loads and linear dynamics results are challenges in most engineering problems. Dr. Nikolay Osipov (Transvalor)

Propagation Simulations

The first part of the presentation shows some of the capabilities of SC/Tetra which enable challenging FSI simulations while the second part shows some examples including one validation case and a study on simulation of Micro Air Vehicle (MAV).

More Innovation Using 2-way Direct Coupling Using SC/Tetra and Abaqus for FSI Simulations

Mr. Yuya Ando (Software Cradle)

SC/Tetra, developed by Software Cradle Co., Ltd., is a standalone general purpose CFD software based on Finite Volume Method using unstructured solver. It can be coupled with Abaqus through Co-Simulation Engine to solve Fluid Structure Interaction (FSI) problems. SC/Tetra is capable of handling moving bodies using Arbitrary Lagrangian Eulerian (ALE) approach. It also has a capability called overset mesh which makes it easy to handle complicated motions. Combining these capabilities makes it possible to handle motions in which one object touches another object or wall where fluid volume becomes zero. These capabilities enable SC/Tetra to handle challenging FSI simulations including large deformation and contacting motions easier.

Advanced Automotive Thermal Simulation Process

Antti Jussila (ThermoAnalytics)

One of the primary focuses in the automotive industry is the total vehicle integration of all major sub-systems and the evaluation of the thermal load experienced within the vehicle under-hood and under-body environments. Advanced simulation processes have been created to simulate real vehicle operating conditions. Typical challenge is the development of a simulation process that fits the need of product development requirements. The challenges here are for example efficiency in computational methods, ability to represent real transient vehicle operating conditions, and turnaround time of simulation that fits, again, the need in product development. A common vision in the virtual vehicle development is the ability to shorten the vehicle development time. This in turn calls for advanced method development both from the computational fluid dynamics and thermal modeling. This presentation discusses the commonly used advanced aero-thermalsimulation methods and their accuracy and turnaround time. Real vehicle simulation examples will be discussed as well as new advanced simulation methodologies that aim to shorten the virtual vehicle development time while introducing more transient details into the simulation methodology.

Z-set to Abaqus: Efficient Tools for Advanced Material Modeling, Damage Analysis and Crack Propagation Simulations

Dr. Nikolaj Osipov (Transvalor)

Transvalor S.A. is a software and engineering services company, specializing in development and integration of numerical simulation codes. Transvalor transfers research results, mainly software packages, from leading French research institutions (Mines ParisTech and ONERA) to industry. Since 1996 Transvalor S.A. commercializes Z-mat for Abaqus - a library of constitutive models for plasticity and viscoplasticity with robust integration methods and advanced coefficient identification procedures. Our presentation will consist of two parts. In the first part of the talk we will go through the presentation of the Z-set products interfaced with Abaqus: * Z-mat - a library for advanced material modeling and parameter calibration * Z-post - a module for FE post-processing and creep-fatigue lifetime estimation * Z-cracks - a software for 3D fracture mechanics simulation In the second part of our talk some examples of application of our products for the solution of industrial tasks in the field of thermomechanical fatigue will be presented.

Load Measurement and Linear Dynamics with Wolf Star Technologies

Understanding and interpreting loads and linear dynamics results are challenges in most engineering problems. Mr. Yuya Ando (Software Cradle)

Load Measurement and Linear Dynamics with Wolf Star Technologies

Dr. Tim Hunter (Wolf Star Technologies)

Wolf Star Technologies will show theoretical background, examples and demonstration of their integrated solutions for understanding loads and interpreting linear dynamic solutions. True-Load™ is the tool provided by Wolf Star Technologies that turns...
components into their own load transducers. True-LDE™ is the post processing solution for *MODAL DYNAMIC, *STEADY
STATE DYNAMICS and *RANDOM RESPONSE solves which makes post processing interactive and natural while reducing
solution time and storage requirements by up to 90%. All of the Wolf Star Technologies solutions are Abaqus/CAE plug-ins
and have direct interface to fe-safe® which makes fatigue analysis a natural part of the design evaluation cycle.

Application of Abaqus to Analysis of 3D Cracks and Fatigue Crack Growth Prediction
Mr. Chris Timbrell (Zentech)
Zencrack provides a state-of-the-art fracture mechanics capability for modelling and analysing 3-dimensional cracks, predicting
their behaviour and growth under linear, non-linear, thermo-mechanical fatigue and time-dependent loading. Zencrack
uses Zentech’s proprietary “crack-block” methodology to quickly and easily generate 3D finite element models of cracked
components from user-supplied uncracked (.inp) meshes. Zencrack generates a new input file of the cracked component,
runs Abaqus, extracts results from the Abaqus output and processes the results to provide energy release rates and stress
intensity factor distributions. Interaction with the uncracked Abaqus mesh, development of the Zencrack input data and
execution of an analysis can all be carried out from within the Zencrack GUI. Post-processing of analysis results in the GUI
allows fast visualisation of crack surface development and generation of a variety of xy plots related to the crack behaviour.
For static or transient loading without crack growth, Zencrack can evaluate and report SIFs, j-integrals and T-stresses along
one or more crack fronts. Both linear and non-linear analyses can be undertaken using a range of the available capabilities in
Abaqus/Standard. For crack growth analysis, Zencrack predicts the magnitude and direction of 3D crack growth along each
crack front and automatically advances each crack through the F.E. model. Several options are available for crack growth data
definition including a user subroutine capability for complex proprietary data. A flexible “load system” approach is provided
for defining load histories which may include complex thermo-mechanical and non-linear loading effects with fatigue and/or
time dependent crack growth.
MAP KEY
- Registration
- Presenter Check-in
- Media Room–Potsdam IV

DS OFFERINGS
- Academic Posters–Postdam Foyer
- Special Exhibit–Potsdam Foyer
- Usability Testing–Schöneberg I
- Make an Appointment/ Customer Support–Schöneberg II
- Partner Demos–Schöneberg III
- Forum Room–Bellevue

INTERCONTINENTAL HOTEL

Schöneberg

Charlottenburg

Köpenick

Tiergarten

Tegel

Bellevue

Potsdam IV

Potsdam I/III (Keynotes and General Lectures)

Potsdam II (Concurrent Lunches)

Exhibit Hall and 3DS Playground

Entrance

WC

WC

WC

WC

WC

WC

WC

WC
Our 3DEXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.