YOUR ESSENTIAL GUIDE TO

SCIENCE
IN THE AGE OF EXPERIENCE™
MAY 15-18, 2017 | CHICAGO, IL

McCORMICK PLACE, CHICAGO
TRAINING DAY: MAY 15
ADDITIVE MANUFACTURING SYMPOSIUM: MAY 15
3DS.COM/EVENTS/SCIENCE-IN-THE-AGE-OF-EXPERIENCE

PLAN YOUR TIME EFFECTIVELY USING OUR
COMPREHENSIVE GUIDE TO SIMULIA ACTIVITIES!
JOIN US FOR SCIENCE IN THE AGE OF EXPERIENCE

Science in the Age of Experience is the next generation in a long line of annual SIMULIA customer conferences extending back nearly three decades allowing SIMULIA users from across the globe to join together to exchange valuable industry knowledge and experiences.

The McCormick Place Convention Center will host SIMULIA, BIOVIA and GEOVIA customers May 15-18, 2017. Last year’s conference was a huge success, and we are planning even more for 2017. Join us in Chicago for the scientific event of the year.

Science in the Age of Experience will contain an expanded thought-leadership program that focuses on the expanding role of science, modeling and simulation to power innovation, solve important problems in products, nature and life, and provide even more value to our customers and to society.

However, attendees will still see the traditional SIMULIA Community Conference features that they love and more! Don’t miss out on your chance to:

• Hear from keynote speakers, including:
  - Juho Könö, Wärtsilä Corporation, Manager, Digital Design Platform
  - Dr. Yonggang Huang, Walter P. Murphy Professor of Mechanical Engineering, Civil and Environmental Engineering, and Materials Science and Engineering - Northwestern University
  - Bernie Riemer, Development Team Lead, SNS Instrument & Source Division, Oak Ridge National Laboratory
• Choose from more than 70 customer presentations in your favorite industries, including Transportation & Mobility, Aerospace & Defense, Industrial Equipment, High Tech, Life Sciences, Consumer Packaged Goods and more
• Choose from 9 Technology Updates
• Attend our expanded Training Day on Monday with a choice of six courses to choose from
• All-new Additive Manufacturing Symposium and Hackathon featuring expert speakers and panelists from NASA, Pennsylvania State University and NIST
• Visit the hands-on demonstrations in the expanded 3DEXPERIENCE® Playground, as well as see what our partner solutions have to offer in the exhibit hall
• Meet R&D experts from SIMULIA, BIOVIA and GEOVIA

We understand it is increasingly difficult to justify time out of the office to attend conferences, trade shows and seminars. Previously, events such as these were the only way to find out about a new product or advance in a specific subject area. However, today, with so much information available via the web, this justification is rarely valid especially in cases requiring international travel. With this in mind, we have assembled this user eBook to convey some of the wider benefits you will receive by attending Science in the Age of Experience.

We hope this eBook will help to prepare you for this year’s exciting event, and we look forward to seeing you in Chicago!
<table>
<thead>
<tr>
<th>Time</th>
<th>Monday, May 15</th>
<th>Tuesday, May 16</th>
<th>Wednesday, May 17</th>
<th>Thursday, May 18</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Morning</strong></td>
<td>Training &amp; Additive Manufacturing Symposium</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Break</strong></td>
<td>Break</td>
<td></td>
<td>Breakouts</td>
<td>Break</td>
</tr>
<tr>
<td><strong>Late Morning</strong></td>
<td>Training &amp; Additive Manufacturing Symposium</td>
<td>Plenary</td>
<td>Breakouts</td>
<td>Breakouts</td>
</tr>
<tr>
<td><strong>Lunch</strong></td>
<td>Lunch</td>
<td></td>
<td>Lunch</td>
<td>Lunch</td>
</tr>
<tr>
<td><strong>Afternoon</strong></td>
<td>Training &amp; Additive Manufacturing Symposium</td>
<td>Breakouts</td>
<td>Plenary</td>
<td>Breakouts</td>
</tr>
<tr>
<td><strong>Break</strong></td>
<td>Break</td>
<td></td>
<td>Break</td>
<td>Adjourn</td>
</tr>
<tr>
<td><strong>Late Afternoon</strong></td>
<td>Training &amp; Additive Manufacturing Symposium</td>
<td>Plenary</td>
<td></td>
<td>Breakouts</td>
</tr>
</tbody>
</table>
The course covers the following topics: using detailed modeling of the microscopic behavior to determine the behavior of composite materials, defining anisotropic elasticity with Holowan models for combining the fiber-matrix response and defining composite layups using Abaqus/CAE, achieving the correct material orientation of the layers of composite shells and solid elements, as well as recent enhancements in the technical capabilities focused on multiscale material modeling and damage.

Although the course content is a subset of the standard Analysis of Composite Materials with Abaqus course, attendees will receive the full set of lecture notes from the standard course. This course material has additionally been updated with new material on multiscale modeling and damage based on new features in the Abaqus product suite. Attendees will be asked to download and install the Abaqus Student Edition on their personal laptops prior to the course in order to fully participate in the hands-on exercises.

**Hands-on with Acoustics and Linear Dynamics with Abaqus: Effective Solution of Structural Vibration Problems**

Linear dynamics and acoustics simulation has become standard practice in many industries to increase user comfort, increase product efficiency, decrease fatigue, and improve the overall experience with engineering products by consumers. This course introduces the user to the algorithms and methods used to study linear dynamic problems with Abaqus/Standard. Additional attention will be devoted to acoustics simulation and noise. In this course, attendees will learn how to model problem cases about a certain frequency, maximize the convergence rate during eigenvalue extraction, determine whether the number of extracted eigenmodes is sufficient to represent the structure’s response adequately, perform transient, steady-state, response spectrum and random response analyses using the eigensolutions, use multiple base motions, and apply damping in linear problems for structures and solids, including internal and external acoustics. Hands-on exercises will reinforce the fundamental concepts presented in the lectures.

Although the course content is a subset of the standard Linear Dynamics with Abaqus course, attendees will receive the full set of lecture notes from the standard course. This course material has additionally been updated with new material on acoustics modeling and simulation based on new functionality in the SIMULIA product suite. Attendees will be asked to download and install the Abaqus Student Edition on their personal laptops prior to the course in order to fully participate in the hands-on exercise.

**ENHANCE YOUR KNOWLEDGE WITH TRAINING DAY**

On Monday, May 15, we present the SIMULIA Training Day. This day was historically known as the Advanced Seminar day. We have revamped and expanded the day to include more technical courses and seminars.

**TECHNOLOGY SEMINARS**

**Computational Fluid Dynamics (CFD) Solutions inside 3DEXPERIENCE**

This course provides an introduction to the Computational Fluid Dynamics (CFD) based roles available in the 3DEXPERIENCE platform. Topics such as geometry preparation, meshing, preprocessing, result visualization, and CFD-CFD integration will be addressed during this seminar. The goal of this course is to educate attendees about the key aspects of the user interface, workflow, and core functionalities of CFD applications offered by 3DEXPERIENCE.

**Topological Optimization using Tosca for Abaqus**

When used in combination with finite element analyses, topology optimization techniques can significantly reduce the number of design iterations required during a product development process. This leads to huge savings in tooling costs and vastly reduces the time to market for the product. The objective of this seminar is to introduce users to the topology optimization capabilities available in “Tosca for Abaqus.” An easy to use interface native to Abaqus/CAE is available for the setup, execution, monitoring and post-processing of topology optimization problems. In combination with Abaqus analysis products, Tosca for Abaqus offers an unparalleled structural optimization capability for linear and highly nonlinear problems. Attendees will leave the course with an understanding of how to apply topology optimization techniques to a structural finite element analysis including setting up design volumes, defining simulation objectives and constraints to consider manufacturing and structural requirements. They will be able to post-process and interpret the results of the simulation to make design recommendations or design alternatives.

**HANDS-ON TRAINING COURSES**

These two Hands-on Training Courses offer attendees a unique opportunity to gain hands-on training while attending Science in the Age of Experience. Attendees will do workshop problems on their own Windows laptops overseen by experienced instructors from SIMULIA. Attendees will receive instructions before the event to pre-load the Abaqus Student Edition on their laptops before coming to the conference. This edition will be theirs to keep after the conference.

**Hands-on with Composites Simulation: Advanced Composite Materials Analysis—Damage and Multiscale**

Composite materials are used in many industries and across a range of design applications because of their high stiffness-to-weight ratios and the ability to tailor their response as needed. This course aims to introduce users to the capabilities in Abaqus enabling effective modeling of composite materials with special emphasis on the topic of multiscale material modeling and damage.
Speakers at the symposium include: engineers and manufacturing engineers/managers, software and hardware vendors, product designers/executives, research leaders, independent consultants, people from different sectors of technology including industry leaders as well as presentations about the latest trends in hardware development. The afternoon session will include panel discussions and technical presentations. We will also be hosting an Additive Manufacturing Hackathon session that will start in the afternoon.

Come prepared to experience the future of design and manufacturing—all driven by science! Visit the Science in the Age of Experience website for the latest agenda and information about the symposium and hackathon.

Goals and objectives of the Symposium:
• Learn about Additive Manufacturing from design to manufacturing
• Engage with software and hardware developers
• Obtain hands-on experience through the Hackathon and network with the Additive Manufacturing community
• Consult with industry experts to address challenges
• Learn about Additive Manufacturing trends in new materials, new processes and new applications

Who should attend the Symposium:
• Software developers
• Hardware vendors
• Research professionals
• Corporate executives and business owners
• Product designers and engineers
• Manufacturing engineers and managers

Who should attend the Hackathon:
• Students
• Research professionals
• Product designers and engineers
• Manufacturing engineers

What is a Hackathon?
Inspired by the well-known tech software/hardware industry hackathons, our Additive Manufacturing Hackathon is an extended creative team-oriented exercise in learning, creating, and doing. Attendees will be divided into teams and given a challenge problem. They will work together throughout the Hackathon, guided by roaming experts on simulation and 3D printing to produce a finished part that will be printed on-site.

Hackathons are fun-filled, inspiring and exhausting races-to-the-finish with a final presentation that will show off the work, creativity, and output from the team. You will be pushed to your limits, but will enjoy every moment of it! If you haven’t done one before, check it out!
GENERAL LECTURES

LEARN ABOUT THE LATEST TECHNOLOGY FROM SIMULIA

Our SIMULIA experts will present a series of General Lectures throughout the conference. These lectures are meant to equip and empower you to innovate.

Delivering Sustainable Innovation with the 3DEXPERIENCE Platform

This lecture explores the emerging reliance on simulation to deliver sustainable innovation and illustrates how the 3DEXPERIENCE platform provides a foundation to meet the increasing demand on simulation to drive the discovery and power innovation in tomorrow’s world. The expanded simulation scope on the 3DEXPERIENCE platform will make possible unparalleled multiphysics-multiscale modeling and simulation for all users.

The Growing Portfolio of SIMULIA’s Multiphysics-Multiscale Technology & Solutions

This presentation highlights applications of our traditional products—including Abaqus, fe-safe, Tosca, Insight and Simpack—to demonstrate the value of SIMULIA’s broad range of simulation technologies. The structures domain continues to be a core focus for SIMULIA, but the availability of multibody simulation, optimization and design space exploration provides users with significant additional simulation capability. Utilizing these combined technologies allows simulation teams to provide enhanced contributions to the product development process. The lecture will also include examples of how the well-established SIMULIA technologies can be used through the 3DEXPERIENCE platform, allowing simulation teams to continue to use the advanced technologies and workflows they have learned while benefiting from the integration, collaboration and management capabilities of 3DEXPERIENCE.

These acquisitions complement existing technology for structures, optimization, plastics, durability & fatigue, and multibody simulation. The strengths and values of these new technologies will be reviewed and placed in context with the future of simulation to enable discovery and power innovation in tomorrow’s world.

The Expanding Scope of Simulation with Established Products

This lecture will explore the emerging reliance on simulation to deliver sustainable innovation and illustrates how the 3DEXPERIENCE platform provides a foundation to meet the increasing demand on simulation to drive the discovery and power innovation in tomorrow’s world. The expanded simulation scope on the 3DEXPERIENCE platform will make possible unparalleled multiphysics-multiscale modeling and simulation for all users.

SIMULIA KEYNOTE SPEAKERS

Science in the Age of Experience—Keynote Speakers

Dr. Yonggang Huang, Walter P. Murphy Professor of Mechanical Engineering, Civil and Environmental Engineering, and Materials Science and Engineering, Northwestern University

Yonggang Huang is the Walter P. Murphy Professor of Mechanical Engineering, Civil and Environmental Engineering, and Materials Science and Engineering at Northwestern University. He is interested in mechanics of stretchable and flexible electronics, and mechanically guided deterministic 3D assembly. He is a member of the US National Academy of Engineering. His recent research awards include the Larson Rvard (2003), Melville Medal (2006), Richards Award (2010), Drucker Prize (2013), and Nadai Medal (2016) from the American Society of Mechanical Engineers (ASME), Young Investigator Medal (2006) and Prager Medal (2017) from the Society of Engineering Sciences (SES), International Journal of Plasticity Medal (2007), Guggenheim Fellowship (2008), and ISI Highly Cited Researcher in Engineering (2009) and ISI Highly Cited Researcher in Materials Science (since 2014). He is the Editor of Journal of Applied Mechanics, a member of the Executive Committee of the RSME Applied Mechanics Division (Chair, 2019-2020), and was the President of SES (2014).

The Spallation Neutron Source—a Tool for Materials Discovery

 Bernie Riemer, Development Team Lead, SNS Instrument & Source Division, Oak Ridge National Laboratory

Bernie Riemer had the good fortune to land an engineering job at the Oak Ridge National Laboratory back in the 80s. Supporting the advancement of science has provided a relentless series of challenges, from fusion energy to neutron sources. Having told a supervisor “I like a challenge”, he got one with the Spallation Neutron Source mercury target. “Twenty tons a minute of mercury flowing through a vessel, being hit with a megawatt of pulsed proton beam … ‘sounds like fun!’” The thermal, stress and dynamics issues – combined with mercury caviation – make for some interesting design and analysis.

SCIENCE IN THE AGE OF EXPERIENCE—KEYNOTE SPEAKERS

Science in the Age of Experience opens on Tuesday, May 16, with plenary material, including speakers such as Scott Berkey and Bruce Engelmann from SIMULIA.
CUSTOMER USER PRESENTATIONS

• Integrating the 3DEXPERIENCE Platform Into Aircraft Analysis Workflows: a Users Perspective, Arthur Dubois - Joby Aviation, LLC
• Topology Optimization of Missile and Aviation Components for AM Fabrication, Devin Hagduke - Materials Sciences Corp.
• Simulation of Semi-Crystalline Composites in the Extrusion Deposition Additive Manufacturing Process, Bastian Brixen - Purdue University
• Simulation of Polymeric Composites Additive Manufacturing using Abaqus, Anthony Favaloro - Purdue University
• A Revolutionary Framework to Enable High-Fidelity Multiscale Modeling, Wen bin Yu - Purdue University
• Modeling Dynamic Behavior of a Safety and Arming Mechanism, Kenan Gurses - Roketsan Missile Inc.
• Tackling the Challenges of Deployable Thin-Film Space Structures with Abaqus, Chiaching Lee - Stinger Ghaffarian Technologies, Inc.
• The Role of Finite Element Analysis in the Development of a 60mm Sensor Mortar Projectile, Pasquale Carlucci - U.S. Army - ARDEC
• Implementation CAD and EF Composite Parts for Aerospace, Minh-Quan THAI - University of Science and Technology of Hanoi
• Numerical Analysis of Ballistic Impact Damage in Layered Isotropic Plates, Ivica Smojar - University of Zagreb, FMEVA

Architecture, Engineering, & Construction
• Finite Element Analysis of Structural Silicone of Warped Insulated Glass Units, Kedar Malusare - Stutzki Engineering
• Sensitivity of Predicted Temperature in a Fillet Weld T-joint to Parameters Used in Welding Simulation with Prescribed Temperature Approach, KIEN NGUYEN - University of Kansas

Consumer Goods & Retail
• Design and Optimization of a Prototype of a Olive Press for Home Use, Hicham Fihri Fassi - University Hassan I

Consumer Packaged Goods & Retail
• Insight Automatic Process in the Virtual Drop Test Simulation for Innovative Packaging Design, Emanuele Sorrentino - Electrolox Italia SPR
• Failure is Good: Perforation Patterns and Stretchy Paper Towels, Nathaniel Hollingsworth - Kimberly Clark Corporation
• Predicting Non-vanishing Web Compression Performance from Fiber Properties, Ted Tower - Kimberly Clark Corporation
• Squeezeable containers: Improving the Consumer experience, Sumit Mukherjee - Plastic Technologies, Inc.

Energy, Processes & Utilities and Natural Resources
• FER Modeling for Thermal Well Casing Connection Evaluation Protocol (TWCCEP), Jueren Xie - C-FER Technologies Inc.
• Using Abaqus to Model Permanent Set in Rubber: Assessment and Sensitivity Study, Allan Zhong - Halliburton
• Improved Thermal Stress Prediction in Quenched Cylindrical Bodies Through a Dynamic Convection Coefficient Library, Patrick Garrity - Naval Nuclear Laboratory-Knolls Atomic Power Lab
• Metal Big Area Additive Manufacturing via Direct Energy Deposition: Process Validation, Srijan Sinovonic - Oak Ridge National Laboratory
• Fatigue Crack Propagation Comparison of a Hydropower Main Inlet Valve using 3D Crack Meshes in a Full Model Versus a Sub-model, Greg Thorwald - Quest Integrity Group, LLC
• Multi-Stage, Multi-Wellbore Hydraulic Fracturing Simulation in Naturally Fractured Reservoirs Using Cohesive Zone Model, Mahdi Haddad - The University of Texas at Austin
• Validation of XFEM-based Simulation Capabilities for Fluid-driven Fractures in Permeable Media, Sandeep Srivastava - University of California, Davis

High-Tech
• Defect Detection and Localization by Lamb Wave Simulation, Ge Li - Beijing Sygna Technologies Co, Ltd
• Finite Element Simulation of the Multi Jet Fusion Process using Abaqus, Daniel Fadil - HP Inc
• Application of Artificial Damping Method to Practical Instability Problems, TAKAYA KOBAYASHI - Mechanical Design & Analysis Corporation
• Making Efficient and Effective Use of Simulation Results for Critical Design Decisions, Charlie Wood - Motorola Mobility
• Challenges with Creep in Under Hood Rotative Parts, Alex Arzounian - Psytech, Inc.

Industrial Equipment
• High Rise Elevators’ Challenges and Solutions in Ride Comfort Simulations, Jan Hernelind - ST Engineering AB
• Exploring the Complex Welding Engineering Design Space using Computational Weld Mechanics, Mahyar Asadi - Applus Canada
• Front-End Loader Linkage Durability Study Using Load Input from True Load, David Slowinski - CHN Industrial
• A Design Modification to Back-Extrusion Process for Through-Thickness Strain Uniformity, Abhishek Bhattacharyya - H.C. Starck
• Development of the Subroutine Library ‘UMM0p’ for Anisotropic Yield Functions, Kai Oide - Mechanical Design & Analysis Corporation
• Mechanical Testing of FDM Parts for Process Simulation, Siddharth Dev - Strategic Technologies Inc.
• Predicting the Properties of Additively Manufactured Parts, Tyler London - TWI Ltd

Life Sciences
• Biomechanics of Craniofacial Fractures: A Simulation Study, SHOBHA E.S. - Dayananda Sagar College of Dental Sciences
• Example Nonparametric Optimization Cases for Additive Manufacturing Using Tosca and Abaqus, Juan Betts - Front End Analytics LLC
• Three Dimensional Modeling and Finite Element Analysis of the Human Spine and Axial Skeletal System Under Dynamic Loads, Roberto Camminio - Illinois Institute of Technology
• Personalized Medical Devices: Contact Lens, Robert Stupplebeen - Optimal Device
• Solution from Lattice Sizing Optimization to Additive Manufacturing, Luyao Cai - Purdue University
• Parametric Study to Evaluate the Effect of Strut Geometry on PLLA Coronary Stent Recoil, Ross Blair - Queen’s University Belfast

It is clear that the sharing of user experiences is the centerpiece of Science in the Age of Experience. These presentations provide first-hand knowledge of deploying SIMULIA solutions and developing workflows for real-world realistic simulation. These presentations usually contain additional detail such as videos and animations not available in the published papers. More importantly, the live presentations enrich the papers by providing unique user-specific viewpoints.

The conference offers a very full agenda of more than 70 user presentations in parallel sessions. Attending the conference also provides you with the opportunity to ask questions at the end of each presentation to clarify specific points, and to meet the presenters in the networking sessions for more detailed discussions of ideas. The titles of the customer presentations being presented during the conference are listed below. The full abstracts can be found on page 23.

Aerospace & Defense
• A Computational Framework for the Analysis of an Aero-Thermochemical-Elastic Erodent Nozzle, Eric Blades - HTR Engineering, Inc.
• Bird Strike and Novel Design of Fan Blades, Syed Noman Husainie - Aventec Inc.
• Lessons Learned in Part Design from Topology Optimization through Qualification, Haley McKee - Honeywell PMAT
• Leak-before-break Leakage and Heat Transfer Simulation of Aircraft Bleed Air System Ducts, Jian Ye - Eaton Aerospace LLC
• A build Distortion Prediction and its Validation in Powder Bed Fusion Additive Manufacturing Processes, Lang Yuan - GE Global Research
• Numerical Analysis of Ballistic Impact Damage in Layered Isotropic Plates, Ivica Smojar - University of Zagreb, FMEVA

Architecture, Engineering, & Construction
• Finite Element Analysis of Structural Silicone of Warped Insulated Glass Units, Kedar Malusare - Stutzki Engineering
• Sensitivity of Predicted Temperature in a Fillet Weld T-joint to Parameters Used in Welding Simulation with Prescribed Temperature Approach, KIEN NGUYEN - University of Kansas

Consumer Goods & Retail
• Design and Optimization of a Prototype of a Olive Press for Home Use, Hicham Fihri Fassi - University Hassan I

Consumer Packaged Goods & Retail
• Insight Automatic Process in the Virtual Drop Test Simulation for Innovative Packaging Design, Emanuele Sorrentino - Electrolox Italia SPR
• Failure is Good: Perforation Patterns and Stretchy Paper Towels, Nathaniel Hollingsworth - Kimberly Clark Corporation
• Predicting Non-vanishing Web Compression Performance from Fiber Properties, Ted Tower - Kimberly Clark Corporation
• Squeezeable containers: Improving the Consumer experience, Sumit Mukherjee - Plastic Technologies, Inc.

Energy, Processes & Utilities and Natural Resources
• FER Modeling for Thermal Well Casing Connection Evaluation Protocol (TWCCEP), Jueren Xie - C-FER Technologies Inc.
• Using Abaqus to Model Permanent Set in Rubber: Assessment and Sensitivity Study, Allan Zhong - Halliburton
• Improved Thermal Stress Prediction in Quenched Cylindrical Bodies Through a Dynamic Convection Coefficient Library, Patrick Garrity - Naval Nuclear Laboratory-Knolls Atomic Power Lab
• Metal Big Area Additive Manufacturing via Direct Energy Deposition: Process Validation, Srijan Sinovonic - Oak Ridge National Laboratory
• Fatigue Crack Propagation Comparison of a Hydropower Main Inlet Valve using 3D Crack Meshes in a Full Model Versus a Sub-model, Greg Thorwald - Quest Integrity Group, LLC
• Multi-Stage, Multi-Wellbore Hydraulic Fracturing Simulation in Naturally Fractured Reservoirs Using Cohesive Zone Model, Mahdi Haddad - The University of Texas at Austin
• Validation of XFEM-based Simulation Capabilities for Fluid-driven Fractures in Permeable Media, Sandeep Srivastava - University of California, Davis

High-Tech
• Defect Detection and Localization by Lamb Wave Simulation, Ge Li - Beijing Sygna Technologies Co, Ltd
• Finite Element Simulation of the Multi Jet Fusion Process using Abaqus, Daniel Fadil - HP Inc
• Application of Artificial Damping Method to Practical Instability Problems, TAKAYA KOBAYASHI - Mechanical Design & Analysis Corporation
• Making Efficient and Effective Use of Simulation Results for Critical Design Decisions, Charlie Wood - Motorola Mobility
• Challenges with Creep in Under Hood Rotative Parts, Alex Arzounian - Psytech, Inc.

Industrial Equipment
• High Rise Elevators’ Challenges and Solutions in Ride Comfort Simulations, Jan Hernelind - ST Engineering AB
• Exploring the Complex Welding Engineering Design Space using Computational Weld Mechanics, Mahyar Asadi - Applus Canada
• Front-End Loader Linkage Durability Study Using Load Input from True Load, David Slowinski - CHN Industrial
• A Design Modification to Back-Extrusion Process for Through-Thickness Strain Uniformity, Abhishek Bhattacharyya - H.C. Starck
• Development of the Subroutine Library ‘UMM0p’ for Anisotropic Yield Functions, Kai Oide - Mechanical Design & Analysis Corporation
• Mechanical Testing of FDM Parts for Process Simulation, Siddharth Dev - Strategic Technologies Inc.
• Predicting the Properties of Additively Manufactured Parts, Tyler London - TWI Ltd

Life Sciences
• Biomechanics of Craniofacial Fractures: A Simulation Study, SHOBHA E.S. - Dayananda Sagar College of Dental Sciences
• Example Nonparametric Optimization Cases for Additive Manufacturing Using Tosca and Abaqus, Juan Betts - Front End Analytics LLC
• Three Dimensional Modeling and Finite Element Analysis of the Human Spine and Axial Skeletal System Under Dynamic Loads, Roberto Camminio - Illinois Institute of Technology
• Personalized Medical Devices: Contact Lens, Robert Stupplebeen - Optimal Device
• Solution from Lattice Sizing Optimization to Additive Manufacturing, Luyao Cai - Purdue University
• Parametric Study to Evaluate the Effect of Strut Geometry on PLLA Coronary Stent Recoil, Ross Blair - Queen’s University Belfast
CUSTOMER USER PRESENTATIONS

- A Concurrent Patient-specific Musculoskeletal and Finite Element Modeling Framework for Predicting in Vivo Kinematics and Contact Mechanics of Total Knee Replacement, SHU Liming - University of Tokyo

Transportation & Mobility

- Cylinder Head Water Jacket Fatigue Optimization Workflow, Zachary Yates - Caelynx
- Designing Cylinder Head Gaskets for New Generation Powertrains, Rohit Ramkumar - Dana Incorporated
- State of the Art Hytrel® Material Modeling Development for the Design of Jounce Bumper, Helga Kuhlmann - DuPont
- Analysis of DuPont Engineering Polymers - Challenges and Solutions, Pieter Volgens - DuPont
- Fatigue Life Prediction Techniques for Polymers and Polymer Matrix Composites, Amir K. Shojaei - DuPont
- Bi-directionally Coupled Multi-Physics Analysis on Fuel Tank, Nagarjun Jawahar - Hero MotoCorp Ltd.
- Dynamic Simulation Methodology of Half-toroidal CVT Variator System with Feedback Control, Toshihiro Saito - Honda R&D Co., Ltd.
- Improvement of Piston Skirt Scuffing Problem using 3D Piston Motion Simulation, Sandwoo Cha - Hyundai Motor Company
- Comparison of ALE, CEL and SPH for Self-piercing riveting, Satoshi Ishikawa - IDAJ Co., Ltd.

FlowVision & Abaqus 2-Way Strongly Coupled FSI Simulation of Automobile Tire Aquaplaning, Min-Feng Sung - Kenda Rubber Ind. Co. Ltd
- Welding Spot Simulate in Compression Analysis of New Energy Vehicle Battery Package, Luyang He - Pan Risa Technical Automotive Center
- A New Fatigue Analysis Method for Bracket of Engine Exhaust System Based on Abaqus Scripting Interface, Degjuan Xu - Pan Risa Technical Automotive Center
- Modelling Rubber Bushings using the Parallel Rheological Framework, Javier Rodriguez - Principia
- Stress Analysis Under Random Loading, James Guo - Robert Bosch LLC
- Optimization of Conformal Cooling Part by Simulia Additive Manufacturing Solution, Wei Zhang - Shanghai Behr Thermal System Co., Ltd
- Reliability Based Design Optimization of Sub-systems / Component using Monte Carlo Simulation of Insight, Saxryn Bonalia - Tata Consultancy Services
- Development of the High-performance Bending Model Using Abaqus, Satoshi Ito - Toyota Motor Corporation
- Vehicle Front-end Shape Optimization for Pedestrian Injury Risk Reduction, Atul Gupta - Waymo (Google self-driving car)
- Vibro-Acoustics (new technology from wave6)
- Abaqus/CAE Updates, including Powered By the 3DEXPERIENCE Platform
- Large Model Solution on the 3DEXPERIENCE Platform

TECHNOLOGY UPDATES

Science in the Age of Experience will present 9 Technology Update presentations in parallel. The material will be repeated several times during the conference allowing attendees to attend the topics that matter to you most. All of the presentation material will be available to all attendees on the SIMULIA Learning Community after the event.

- Abaqus Solvers (/Standard & /Explicit), Contact, Performance, Mechanics, and Materials
- Major Advances in CFD (Fluids and Plastics)
- Multibody Simulation Enhancements
- Structures and Simulation on the 3DEXPERIENCE Platform
- Process Automation and Design Exploration on the 3DEXPERIENCE Platform
- Multiscale Systems Simulation using Multiscale-Multiphysics Co-Simulation on the 3DEXPERIENCE Platform
- Vibro-Acoustics (new technology from wave6)
- Abaqus/CAE Updates, including Powered By the 3DEXPERIENCE Platform
- Large Model Solution on the 3DEXPERIENCE Platform

ELECTROMAGNETICS FORUM

The Electromagnetics Forum is a special parallel track supplementing the technical program on Thursday, May 18. During this Forum, SIMULIA will introduce its industry leading technology on electromagnetics that was added to its portfolio through the acquisition of CST in 2016. The Forum starts with a broad overview of the electromagnetics technology highlighting the application areas it serves.

The overview is followed by industry focused presentations demonstrating the value electromagnetic simulation brings in addressing industry specific challenges in Aerospace & Defense, Transportation & Mobility, and High Tech industries. The Electromagnetics Forum is appropriate for anyone interested in learning more about electromagnetic simulation and how this simulation fits into their usage of other SIMULIA products, including coupled structural-thermal-electromagnetics analyses.
USABILITY TESTING
Help us develop and participate in Usability Studies at Science in the Age of Experience with trained usability researchers and complete typical tasks in various parts of our software. Help us understand how easy or difficult 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 do this?
Participating in a usability study lets you, the customer, give feedback before the release of software. It makes you a partner in our development effort.

Interested in helping?
Sessions run throughout the conference. Contact the Usability Research Team in the Exhibit Hall to schedule a session during or after the conference.

LIFE SCIENCES SPECIAL INTEREST GROUP: MONDAY, MAY 15
The Life Sciences Special Interest Group (SIG) meeting is an open forum to learn about recent advances in modeling and simulation that are driving transformation in healthcare and in every industry where human experience-based design concepts are essential. The forum is an opportunity to exchange ideas and explore challenges and solutions with others from around the world who are working on modeling and simulating the human body at different length and time scales, using different physics and chemistry, and for different purposes. Further, it will be a unique opportunity to interact directly with key SIMULIA and BIOVIA technology and business leaders who share the goal to accelerate the use of modeling and simulation to inspire personalized solutions, ranging from personalized medical treatments to personalized protective gear, wearable devices, and everyday items such as shoes and clothes.

NEED CUSTOMER SUPPORT?
Customer Support and Application Engineers will be available for one-on-one help and advice with your current simulation models at the Support Desk. Get more specific information about your current simulation tasks or ask about best-practices and get the advice of our experienced support team for how to start a future activity. You are welcome to bring in your models and files on a memory stick, if needed, for help on model-specific issues. No appointment necessary!
The Customer Support Desk will be open during breaks, lunch and the technical paper sessions.

ACADEMIC POSTERS
We are pleased to once again present the Academic Poster Showcase. We receive posters from around the globe covering a variety of research and teaching endeavors. They represent a cross section of the many important industries we support with our SIMULIA products. Be sure to visit our display located in the exhibit hall to view the impressive work these professors, students and researchers have shared and while you’re there, VOTE for your favorite!

Saint Louis University, Sheila M. Buswell, 2016.

LIFE SCIENCES SPECIAL INTEREST GROUP, CUSTOMER SUPPORT
This Special Interest Group meeting will be invaluable to researchers and engineers from the medical device, pharmaceutical, high tech and consumer products industries who are trying to put people at the center of design and engineering, using virtual human modeling and simulation.
EXPLORE THE SIMULIA ECO-SYSTEM

Join our Alliances Partner Software Demonstrations

A selection of SIMULIA partners will provide a 20 minute demonstration of their software, showing how their software integrates with SIMULIA software to create a complete solution. Join us in the Partner Theater in the Exhibit Hall.

**BETA CAE**

When reality is ahead of simulation, 3D geometry often comes in the form of a tomography. RETOMO will reliably turn tomography voxels to virtual prototypes. With its streamed image processing technology it can process large datasets and deliver effortlessly large FE-Models, ready for pre-processing.

**Capvidia**

2-Way Strongly Coupled FSI between Abaqus & FlowVision
- Automatic Re-Meshing (Not Limited to Mesh Deformation)
- Coupling; Natural Data Exchange between CFD and FE Meshes (No Intermediate Mesh like MPccI)
- Multi-Physics Manager; (Re)Starting Co-Simulation & Real-Time Results Monitoring
- Completely Independent (Implicit/Explicit) Integration Steps
- Data Exchange at User-Defined Number of Time Steps

**Endurica**

This demo shows how multiple operating scenarios can be combined with fe-safe/Rubber’s multi-block analysis feature to compute fatigue life for an elastomeric component. The demo will also show how you can visualize results from the critical plane search to trace, diagnose and mitigate damage sources.

**Granta Design**

Materials data is essential for simulation. GRANTA MI ensures that analysts get the right materials data, in the right format, error-free. Your company can manage the full lifecycle for simulation data and all related datasets. Guarantee full traceability, increase confidence in results and re-use of data, and enable a more efficient workflow between CAD and CRE. This demonstration will show how this data can then be accessed directly within Abaqus/CAE via the embedded GRANTA MI:Materials Gateway app. Material models can be found through text and/or property searches. These can then be imported into your CRE models.

**Intel**

Simplify HPC stack integration, validation and maintenance. Join us for an interactive presentation and video demonstration of the installation of Intel® HPC Orchestrator on a four-node Intel® Xeon Phi™, Omni-path Architecture cluster. This session will also provide an overview and highlight the components that make up Intel® HPC Orchestrator. We will highlight and discuss the benefits and ease of installation of Intel® HPC Orchestrator for HPC clusters ranging from four to tens of thousands of nodes.

**Synopsys**

This demo is aimed at those interested in creating FE/CFD models from 3D image data (MRI, CT, microCT, Microscopy,...). We will demonstrate the ease of generating high fidelity meshes in Simplware and exporting directly to Abaqus. Workflows presented will include applications from Life Sciences, Materials and NDE & Reverse Engineering.

**ThermoAnalytics**

ThermoAnalytics will demonstrate our TRATherm transient 3D thermal software simulating the transport of consumer goods requiring a temperature controlled environment. The scenario will have a commercial hybrid electric vehicle operating under a common delivery drive-cycle. The electrical and thermal effects from the vehicle’s battery charge and discharge will be modeled during the transient drive-cycle. The fast transient thermal modeling features will be highlighted, including transient environmental loads from natural weather (summer and winter) and advanced features that enable faster and more efficient engineering decisions.

**WolfStar Technologies**

Dr. Tim Hunter, creator of True-Load, will demonstrate using True-Load to reconstruct operating loads on a Baja car suspension. This technology can be applied to a variety of applications from Aerospace to Medical Technologies. The reconstructed loads produce correlated FEA strain response and can also be used for fatigue analysis.
nTopology makes 3D design software that exploits the benefits of industrial additive manufacturing. Our first product, Element, allows for highly complex lattice structures to be designed, simulated, and tuned for demanding applications.

nTopology’s first product, Element, is a 3D design suite made specifically for additive manufacturing. Element provides the feedback engineers need to account for the design limitations and opportunities of additive manufacturing. With Element, parts that before would take weeks to design can be imagined and sent to manufacturing in a matter of hours.

ThermoAnalytics provides Total Thermal Solutions for complex heat problems and engineering analysis. Our software is the leading thermal analysis program for rapid design and optimization of heat transfer problems, including transient brakes, underhood models, exhaust and underbody simulation. HVAC, cabin, battery packs for HEV/EV, electronics and other thermal sensitive components. ThermoAnalytics’ rapid transient thermal analysis and coupling to FER and CFD software is key to an efficient design optimization process. Our software is commonly used in the automotive, aerospace, motorcycles, heavy vehicles and the railway industry. ThermoAnalytics also offers advanced consulting services with our engineering teams that specializes in thermal, CFD, infrared simulation and testing.

Learn More

Visit ThermoAnalytics.com to learn more about their Total Thermal Solutions for complex heat problems and engineering analysis.

View Brochure

Wolf Star Technologies provides first to market solutions for FEA: True-Load™ turns any component into an N-DOF load transducer to generate strain correlated loading; True-LDE™ provides intuitive post-processing for dynamic FER solutions. All Wolf Star Products automate the creation of complex durability analysis via direct interface to fe-safe.

Use password TL2017 to Watch Webinar.

ThermoAnalytics


Application Areas: Fluid Dynamics, Human Modeling, Solids & Structures

TotalCAE sells and manages turnkey High Performance Computing cluster solutions for on-premise and the public cloud that reduce turnaround time and increase productivity for engineers. TotalCAE makes it simple to adopt both private and public cloud computing solutions for all your engineering applications as the one stop shop for engineering IT solutions.

View Brochure

ThermoAnalytics


Application Areas: Fluid Dynamics, Multiphysics, Solids & Structures

Use password TL2017 to Watch Webinar.

TotalCAE


Application Areas: Compute Management

ThermoAnalytics

Industries Served: Transportation & Mobility, Aerospace & Defense, Marine & Offshore, Industrial Equipment, High Tech, Consumer Packaged Retail Goods, Life Sciences, Natural Resources

Application Areas: Fluid Dynamics, Human Modeling, Solids & Structures

Visit ThermoAnalytics.com to learn more about their Total Thermal Solutions for complex heat problems and engineering analysis.

View Brochure

Wolf Star Technologies

Industries Served: Transportation & Mobility, Energy Process & Utilities, Natural Resources, Marine & Offshore, Industrial Equipment

Application Areas: Durability, Noise & Vibration, System Modeling

Visit WolfStar.com to learn more about their True-Load™ and True-LDE™ solutions for industrial additive manufacturing.

View Brochure

nTopology

Industries Served: Transportation & Mobility, Aerospace & Defense, Marine & Offshore, Industrial Equipment, High Tech, Consumer Packaged Retail Goods, Life Sciences, Natural Resources

Application Areas: Fluid Dynamics, Multiphysics, Solids & Structures

View White Paper

Endurica


Application Areas: Durability, Materials, Noise & Vibration

Endurica brings fatigue life prediction capabilities to the rubber industry. We offer computer simulation software for engineers to address durability issues at the design stage by using your FEM. In addition to fatigue life simulation software, we also provide testing equipment, materials characterization, consulting services, training and FE modeling.

View Video
CONFERENCE REGISTRATION

We understand that it is increasingly difficult to justify time out of the office to attend conferences, trade shows and seminars. 
Need to justify attendance to your manager? Download this letter!

Conference Registration Fees
Conference Pass: $119
Training Day: $50
Additive Manufacturing & Hackathon: $50
Day Pass: $595
Academic: $695*

*Please note that verification of your student status or accreditation as a full-time instructor at an accredited academic institution is required at the time of registration.

VENUER AND ACCOMMODATIONS

Adjacent to the Convention Center is the conference Hotel, Hyatt Regency McCormick Place with sweeping views of Lake Michigan, comfortable, spacious rooms and close to many of the City’s exciting attractions such as Soldier Field, the Museum Campus, the Art Institute of Chicago, Skydeck Ledge and shopping on the Magnificent Mile.

Rooms are limited. Book your room online today to ensure you receive the discounted rate.

Airport Information
The nearest major airport is Chicago Midway International Airport. This airport has international and domestic flights from Chicago and is 12 miles from the center of McCormick Place, IL.

Another major airport is Chicago O’Hare International Airport, which has international and domestic flights from Chicago and is 23 miles from McCormick Place, IL.

About Chicago
Located only minutes from downtown Chicago, McCormick Place is North America’s premier convention facility, attracting close to 3 million visitors annually. Chicago, on Lake Michigan in Illinois, is among the largest cities in the U.S. Famed for its bold architecture, it has a skyline punctuated by skyscrapers such as the iconic John Hancock Center, 1,451-ft. Willis Tower (formerly the Sears Tower) and the neo-Gothic Tribune Tower. The city is also renowned for its museums, including the Art Institute of Chicago with its noted Impressionist and Post-Impressionist works.

AEROSPACE & DEFENSE


A multiphysics simulation capability has been developed that incorporates fundamental interactions between aerodynamics, structural response from aerodynamic heating, and heat transfer. The multiphysics framework couples CFD (1D, 2D, and 3D, implicit char ablator solver), Loci/CHEM (a computational fluid dynamics solver for high speed chemically reacting flows), and ABAQUS/Standard to create a fully coupled aerothermochemical charring ablative solver. The solvers are tightly coupled using the Co-Simulation Engine to resolve the effects of the ablating pyrolysis and char mass loss process upon the flow field, the changes in surface geometry due to recession upon the flow field, and thermal-structural analysis of the body from the induced aerodynamic heating from the flow field. The multiphysics simulation capability was successfully demonstrated on a solid rocket motor graphite nozzle erosion application. Comparisons were made with available experimental data that measured the throat erosion during the motor firing. A series of parametric studies were conducted using the coupled simulation capability to determine the sensitivity of the nozzle erosion to different parameters. The parameter studies included the shape of the nozzle throat (flat versus rounded), material properties, the effect of the inclusion/exclusion of heat conduction and the mechanical thermal expansion. Overall, the predicted results match the experiment very well and the predictions were able to bind the data within acceptable limits.

Bird Strike and Novel Design of Fan Blades, Syed Noman Husaini - Aventec Inc.

The paper will include research performed at the University of Toronto for a Master of Engineering project. The intent of this project was to find an appropriate bird substitute model and evaluate how engine blade properties can be tweaked to better handle bird strikes. Several bird geometries and material models were evaluated using the Lagrangian and smoothed particle hydrodynamic (SPH) modeling approaches. The SPH method was found to be more efficient for modeling a bird. All simulations were performed using ABAQUS. The material model for the calibrated bird model was best represented using a tabulated equation of state with 10% porous gelatin. The results for the hemispherical-ended cylinder with a length-to-diameter ratio of 1.6 correlated well with experimental results. Bird impact simulations on various blade geometries showed that increasing the blade twist for a rotating blade would improve its impact performance. It’s recommended that leading edges of an engine blade are kept solid Regions away from the leading edge may adopt a sandwich configuration to minimize weight.

Leak-before-break Leakage and Heat Transfer Simulation of Aircraft Bleed Air System Ducts, Jian Ye - Eaton Aerospace LLC

For the safe design of aircraft bleed air system ducts and assessment of ducts with minor cracks found in the quality inspection, it is required to understand and predict the crack behavior and leak air leakage. A leak before break is desired so a leak can be detected and a repair made before a structural failure. Measuring the leaked air temperature is a widely used technique to detect the leakage flow. The ground maintenance team uses thermal imaging and temperature probes, while the on-board system uses temperature sensor wires to detect the hot air leakage. In this study, the Extended Finite Element Method (XFEM) is used to evaluate if an initial crack will grow into unstable failure. CFD analysis is used to calculate the leak-before-break crack leakage rate and heat transfer (including conduction, convection and radiation) in the case of a stable through-crack, utilizing the crack dimensions obtained from crack propagation simulation. The prediction of the leakage flow and temperature field enables a better knowledge of the air leakage and heat loss through the crack and provides the guideline for the duct leakage detection system.

As-build Distortion Prediction and its Validation in Powder Bed Fusion Additive Manufacturing Processes, Lang Yuan - GE Global Research

During powder bed fusion additive manufacturing (RM) processes, distortion is one of the most common manufacturing issues that occur during the fabrication process. It needs to be pre-compensated or post-processed after build. To achieve the desired geometric tolerance, finite element modeling has been applied to provide insights into the major factors that lead to distortion and the residual stresses which may lead to cracking. This enables engineers to pre-compensate part geometry and optimize part design to significantly reduce design-to-built iterations and save cost. A streamlined workflow that associates subdomains of a thermal-structural finite element model consistent with different portions of build path data and appropriately integrated heat input energy was carried out on a canonical geometry. Validation studies will
The Kansas City National Security Campus (KCNSC) is a government sponsored, engineering and manufacturing facility located in Kansas City, MO. Simulation analysts at the KCNSC has been utilizing Tosca for topological optimization since December 2014. Topology optimization is a powerful method in the field of manufacturing, but currently lacks any sort of guiding workflow. This paper introduces a first iteration workflow for using topology optimization as a design method within a digital manufacturing framework. Unlike traditionally designed and machined parts, topology optimized and additively manufactured parts lack definitive standards for processing technologies mature, new design workflow processes, similar to those described in this paper, will become more common. The presented work with focus on the modeling of material crystallization is believed to be an essential step towards the manufacturing of components generated using multi-physics topology optimization techniques. MSC will present a complete solidification simulation of 3D printed carbon fiber tooling and parts.

Simulation of Semi-Crystalline Composites in the Extrusion Deposition Additive Manufacturing Process, Bastian Brenken - Purdue University

A UMATHT user subroutine was developed in Abaqus to combine a non-isothermal dual crystallization kinetics model with a statistical melting model in order to describe the simultaneous solidification/re-melting behavior of 3D printed parts during the Extrusion Deposition (ED) process. This subroutine is designed to describe the crystallization behavior is significant and strongly dependent on the utilized polymer. As an outlook, the interaction with a second UMAT user subroutine that will be employed to predict residual stresses and deformations is explained.

Recent developments in additive manufacturing have been made in ED. In order to improve stiffness and minimize warpage of printed parts, polymers with fiber reinforcement have gained special interest. Tools and molds are a key technology in this field. In order to print carbon-fiber composite tooling, a high thermal stability of the printed tool is essential to maintain shape throughout multiple thermal cycles. Consequently, high temperature thermoplastics with high fiber contents should be employed. Semi-crystalline polymers like polyphenylene sulfide (PPS) or polyparaphenylene ether ketone (PEEK) are potential candidates. Their partially crystalline structure adds significant thermal stability, but also additional shrinkage during cool down. Therefore, a crystallization simulation needs to be included in an overall solidification analysis to predict the residual stress and deformation state of printed tools.

The presented work focuses on the modeling of material crystallization is believed to be an essential step towards a complete solidification simulation of 3D printed carbon fiber tooling and parts.

Simulation of Polymeric Composites Additive Manufacturing using Abaqus, Anthony Favaloro - Purdue University

Additive manufacturing, specifically the extrusion deposition process, involves the progressive addition of material at elevated temperatures following a prescribed machine path at prescribed speed. In order to properly simulate the thermal history and resulting mechanical deformations and stresses, this transient addition of material must be captured. Utilizing the newly available features of Abaqus 2017, specifically element activation and event series, progressive element activation is performed by implanting the user subroutine UEPActivationVol. In this work, the system at Purdue University for extrusion deposition of highly filled, high temperature thermoplastics is modeled through a user subroutine suite which coordinates element activation according to machine instructions, assigns appropriate local coordinate systems for using anisotropic material properties, and sets relevant initial state variables for user material models. Simple structures are modeled and printed to calibrate convection and radiation properties.

A case study of printing an air inlet duct tool is presented showing the full part simulations using a voxel mesh predicting both in process deformations and stresses as well as deformations and stresses following a spring back analysis representing part removal from a build plate.

The impact velocities are in the range from 75 to 550 m/s, whereas the mass of the impactor is approximately 0.2 kg. The thickness of the stationary target plates has been varied from 0.2 to 20 mm. The simulations have been performed in Abaqus/Explicit as to employ the broad range of available material models and due to the robustness of the simulation in highly nonlinear transient problems, where complex contact conditions are present. The Johnson-Cook strain rate dependent plasticity model has been employed to realistically simulate the impact behavior of the target plate. Additionally, a Continuum Damage Mechanics model has been employed as to model the damage processes in the material. According to this model, damage effects are triggered based on the value of equivalent plastic strain in the material. Plate perforation has been simulated using the element deletion criterion based on the damage parameter.

The material parameters of the constitutive model have been determined from the literature and by validation against the experimental ballistic curves available in the literature. The numerical simulations have correctly replicated the available experimental results, thereby confirming the validity of the numerical procedure and the employed material constitutive, degradation and failure models.

Lessons Learned in Part Design from Topology Optimization through Qualification, Haley McKee - Honeywell FM&T

The work presents a numerical procedure employed for the simulation of high-velocity impact phenomena at Weldox FMENA. The simulations have been performed in Abaqus/Explicit as to employ the broad range of available material models and due to the robustness of the simulation in highly nonlinear transient problems, where complex contact conditions are present. The Johnson-Cook strain rate dependent plasticity model has been employed to realistically simulate the impact behavior of the target plate. Additionally, a Continuum Damage Mechanics model has been employed as to model the damage processes in the material. According to this model, damage effects are triggered based on the value of equivalent plastic strain in the material. Plate perforation has been simulated using the element deletion criterion based on the damage parameter.

The material parameters of the constitutive model have been determined from the literature and by validation against the experimental ballistic curves available in the literature. The numerical simulations have correctly replicated the available experimental results, thereby confirming the validity of the numerical procedure and the employed material constitutive, degradation and failure models.

Integrating the 3DEXPERIENCE Platform into Aircraft Analysis Workflows: A User’s Perspective, Arthur Dubois - Joby Aviation, LLC

Aircraft design is an inherently multidisciplinary process; it involves cooperation across many engineering specialties. Such cooperation hinges on efficient exchange of information between different teams and tools. One key variable for information flow is consistency across engineering tools, be they for Computer Aided Design, Analysis or Project Management. Dassault Systemes 3DEXPERIENCE platform is aimed at providing an all integrated environment where most engineering tools coexist under the same roof. Joby Aviation is seeking to take full advantage of the 3DEXPERIENCE platform and relax the boundaries between design and analysis. This paper outlines some of the benefits and challenges, both technical and organizational, associated with migrating from a segregated design environment where design tools are separate from analysis tools to an integrated one. The work introduces several technical use cases showcasing some current capabilities of the 3DEXPERIENCE platform. Firstly, the design and analysis of a metal bracket will be presented to highlight the ability to quickly iterate on simple concepts. Secondly, a control surface mechanism will be discussed to highlight simulation capabilities on complex assemblies. Finally, the design of a flight critical linkage arm will be considered to showcase the use of the Generative Design Explorer toolbox for topology optimization.

Topology Optimization of Missile and Aviation Components for AM Fabrication, Devlin Hayduke - Materials Sciences Corp.

Components within missile systems are vulnerable to performance degradation as a result of heat and vibrations generated by neighboring components. Conventional methods to alleviate this degradation include installing passive vibration-damping materials, adding material to shift resonance frequencies and adding heat sinks to remove unwanted heat. All of these approaches add parasitic weight to the system. Topology optimization methods are well established analysis tools that are used to determine an optimal material distribution of a design space, subject to a performance constraint, for a given set of loads and boundary conditions, making them ideal for tailoring the thermal and frequency response of missile components and associated structures without adding parasitic weight. Generally, topology optimization results in complex geometries that have been difficult to realize with conventional manufacturing methods. However, given the recent advances in additive manufacturing (AM) technologies, the full potential of topology optimization as a design tool can be realized. Materials Sciences Corporation (MSC) and the U.S. Army Aviation & Missile Research Development & Engineering Center (AMRDEC) have focused on linking Abaqus and Tosca with in-house codes to develop a design, fabrication and verification process that enables additive manufacturing of components generated using multi-physics topology optimization techniques. MSC will present several examples of optimized designs for missile and aviation applications that were developed using the multi-physics topology optimization capabilities, i.e., statically determined stress, frequency, temperature, etc., within Tosca.

Simulation of Polymeric Composites Additive Manufacturing using Abaqus, Anthony Favaloro - Purdue University

Additive manufacturing, specifically the extrusion deposition process, involves the progressive addition of material at elevated temperatures following a prescribed machine path at prescribed speed. In order to properly simulate the thermal history and resulting mechanical deformations and stresses, this transient addition of material must be captured. Utilizing the newly available features of Abaqus 2017, specifically element activation and event series, progressive element activation is performed by implanting the user subroutine UEPActivationVol. In this work, the system at Purdue University for extrusion deposition of highly filled, high temperature thermoplastics is modeled through a user subroutine suite which coordinates element activation according to machine instructions, assigns appropriate local coordinate systems for using anisotropic material properties, and sets relevant initial state variables for user material models. Simple structures are modeled and printed to calibrate convection and radiation properties. A case study of printing an air inlet duct tool is presented showing the full part simulations using a voxel mesh predicting both in process deformations and stresses as well as deformations and stresses following a spring back analysis representing part removal from a build plate.

The work presents a numerical procedure employed for the simulation of high-velocity impact phenomena at Weldox FMENA. The simulations have been performed in Abaqus/Explicit as to employ the broad range of available material models and due to the robustness of the simulation in highly nonlinear transient problems, where complex contact conditions are present. The Johnson-Cook strain rate dependent plasticity model has been employed to realistically simulate the impact behavior of the target plate. Additionally, a Continuum Damage Mechanics model has been employed as to model the damage processes in the material. According to this model, damage effects are triggered based on the value of equivalent plastic strain in the material. Plate perforation has been simulated using the element deletion criterion based on the damage parameter.

The material parameters of the constitutive model have been determined from the literature and by validation against the experimental ballistic curves available in the literature. The numerical simulations have correctly replicated the available experimental results, thereby confirming the validity of the numerical procedure and the employed material constitutive, degradation and failure models.

Integrating the 3DEXPERIENCE Platform into Aircraft Analysis Workflows: A User’s Perspective, Arthur Dubois - Joby Aviation, LLC

Aircraft design is an inherently multidisciplinary process; it involves cooperation across many engineering specialties. Such cooperation hinges on efficient exchange of information between different teams and tools. One key variable for information flow is consistency across engineering tools, be they for Computer Aided Design, Analysis or Project Management. Dassault Systemes 3DEXPERIENCE platform is aimed at providing an all integrated environment where most engineering tools coexist under the same roof. Joby Aviation is seeking to take full advantage of the 3DEXPERIENCE platform and relax the boundaries between design and analysis. This paper outlines some of the benefits and challenges, both technical and organizational, associated with migrating from a segregated design environment where design tools are separate from analysis tools to an integrated one. The work introduces several technical use cases showcasing some current capabilities of the 3DEXPERIENCE platform. Firstly, the design and analysis of a metal bracket will be presented to highlight the ability to quickly iterate on simple concepts. Secondly, a control surface mechanism will be discussed to highlight simulation capabilities on complex assemblies. Finally, the design of a flight critical linkage arm will be considered to showcase the use of the Generative Design Explorer toolbox for topology optimization.
The objective of this study was to investigate the use of composites as an alternative in the aircraft wing and thus help in reduce the weight of the plane. In general, composite material is used in the aviation industry to purpose as an aircraft wing. The wing structure is design in CATIR V6, and the skin of wing is applied with CATIR composite design. Then the model is exported to Abaqus to calculate, simulate and optimize.

The Role of Finite Element Analysis in the Development of a 60mm Sensor Mortar Projectile, Pasquale Carlucci U.S. Army - ARDEC

Reconnaissance projectiles are being developed by the U.S. military with increasing frequency to facilitate a variety of emerging needs. One of those needs includes large range deployment of these projectiles, while maintaining the survivability of electronic sensors and other components. To help fill this need, a completely new remotely deployable 60mm sensor mortar projectile (SmortarNet) was designed and developed by ARDEC. Through the use of finite element analysis (FEA), the design cycle was drastically reduced to the point where first article prototypings successfully passed live firing testing at Yuma Proving Ground. FEA was used to ensure that the projectile would successfully survive live fire launch and blow-by overpressure conditions before any testing was conducted. This paper will give an overview of the design, and the key role that FEA played in developing a successful design at a reduced development cost and short project schedule.


Powder bed fusion is an additive manufacturing process that produces parts in a layer-by-layer fashion using a digital CAD file. This method provides the ability to produce complex geometries that are either impossible or inefficient to create through traditional manufacturing processes. In spite of its potential benefits, however, powder bed fusion technology faces considerable challenges preventing its widespread adoption into industry. In particular, rapid melting and solidification caused by the moving heat source (either electron beam or laser) occurs on very small time scales (~1 millisecond), which can induce large distortions and residual stresses. In this study, experiments are performed and then simulated using the general purpose finite element code Abaqus in order to predict distortions and residual stresses in a laser powder bed fusion process. Separate experiments (termed “thermal” and “mechanical”) are conducted to validate the thermal and mechanical problems. For the thermal experiments, temperatures are measured with thermocouples in a 316 stainless steel powder bed in order to characterize the thermophysical properties of the bulk material. For the mechanical experiments, thin arches are built using the same material. By measuring the spring-back of the arches (using digital image correlation) after their removal from the build plate, residual stresses are inferred. For the simulations, a sequential thermal-mechanical coupling is adopted, in which temperatures are calculated first and then used as inputs for the stress analysis. After modifications to model parameters, the simulation successfully predicts the observed distortions.

ARCHITECTURE, ENGINEERING, & CONSTRUCTION

Finite Element Analysis of Structural Silicone of Warped Insulated Glass Units, Kedar Malasure - Stinger Gaffarian Technologies, Inc.

New sophisticated modeling and analysis tools are allowing both architects and engineers to push the limits of their ingenuity to design and build unique structures. This abstract pertains to out-of-plane warped glass of the McKinney and Olive building located in Dallas, TX. The insulated glass units are held together using aluminum mullions through structural silicone. The warping of glass units puts the silicone in a state of tensile stress, which if fails will cause the glass units to fall off the structure. A solid rhino model was obtained from the client and was first analyzed to determine maximum amount of warping using an in-house grasshopper script. The warping information was then used to model the warped glass units (129 in number) in Abaqus using shell elements. The maximum tensile stress near the warped corner was then analyzed for all the thin glass units. The entire repetitive procedure of model building and analysis was accomplished through python scripting in Abaqus. The results from the shell modeling were used to determine the worst-case glass units which were then built and analyzed in Abaqus using solid elements. The solid models were then used to accurately determine the tensile stress in the structural silicone and were determined to be below the allowable stress limits.

Sensitivity of Predicted Temperature in a Fillet Weld T-Joint to Parameters Used in Welding Simulation with Prescribed Temperature Approach, Kien Nguyen - University of Kansas

Abaqus can be used to simulate welding processes, but the procedure can be time consuming due to a large number of steps necessary to generate weld beads and the associated thermal loads and convective fluid interactions. Recent developments of the Abaqus Welding Interface (RWI) address these challenges, as the RWI utility automatically creates all of those steps. While the RWI procedure is quite straightforward, its accuracy can be expected to be highly dependent on the magnitude of several parameters defined in RWI: torch temperature, temperature ramping option,
and deposited weld “chunk” length. To use the RWI capabilities, these parameters need to be calibrated to achieve a proper thermal solution for each welding simulation. However, there is no available guidance on the calibration procedure, and the effects of these parameters are not fully understood. This paper presents a sensitivity study of temperature field parameters using a case study of a T-joint fillet weld. Ten models were created in which the welding process was simulated using the RWI utility with varying torch temperatures, ramping options, and deposited weld chunk. The obtained results were compared with available data from published work for the same welded detail. For the weld studied, it was observed that the best option for the weld chunk size was 10mm, while the torch temperature should be in the range of 1400-1500°C, which can be complemented by adjusting the ramping options to obtain an improved result. A discussion regarding a general procedure using the RWI to calibrate a welding simulation, and the merits of the prescribed temperature approach used in RWI is presented.

CONSUMER GOODS & RETAIL

Design and Optimization of a Prototype of an Olive Press for Home Use, Hicham Fihri Fassi - University Hassan 1

The domestic oil press is considered as one of the most important machines in the field of olive farming. It facilitates the process that is for the means of the small farmers. A machine replaces the old pressing process at different levels. On the one hand, it is a perfect equipment for the treatment of olives immediately after picking in order to extract a high quality extra virgin olive oil and on the other hand, it can press other dried fruits and extract their oils. The specificity of the domestic oil press is that it combines the technique of the old process, especially the grinding and material. The originality in this machine is the possibility of adding a part that will allow us to separate the oil from the water, and another that allows to treat the residues of olives and used after. This machine is designed and simulated by CARR and Rabaus.

CONSUMER PACKAGED GOODS & RETAIL


This paper illustrates a SIMULIA SEE (Fipiel) process involving Rabaus/Explicit in order to perform an automatic complex sequence of virtual drop test on the packaging of gas hobs. The virtual drop process is intended to be used by the designers without specific knowledge of the explicit dynamic simulations and it involves automatic mesh of the packaging CAD geometries. The Insight virtual drop process enables the design of innovative packaging; it supports the introduction of new materials in the packaging technologies reducing the time to market within the usual try & fix of the physical testing. The process has been designed flexible enough to address the meaningfull library of the gas hobs product line shapes and material. Some numerical versus physical test validation will be shown.

Failure is good: Perforation Patterns and Stretchy Paper Towels, Nathaniel Hollingsworth - Kimberly Clark Corporation

Engineering analyses are often targeted to help design parts and systems that provide a margin of safety against failure. Most sophisticated models that include plasticity, damage, and failure help the analyst assess the part or system’s ability to continue to serve its intended function safely in a partially-damaged state. However, at Kimberly Clark we like failure—specifically when consumers are tearing a paper towel off the roll (known as “dispensing”) in the parlor of consumer packaged goods). By utilizing Abaqus, the essential role of the mechanical behavior (including failure) of the towel and the perforation pattern has in dispensing has become clearer.

In 2014, Kimberly-Clark launched a new, stretchable paper towel, VIVAR® Vantage® (denoted Vantage). Vantage towels are unique in that they can stretch about four times further than other paper towels before tearing. To explore dispensing performance, the towels were modeled with a neo-Hookean hyperelastic model combined with a plasticity model, damage initiation criteria, and a damage evolution rule. New mechanical tests were developed to calibrate the material model. Perforation patterns were parameterized to investigate the interplay of the mechanical properties and the perforation pattern. Both the material calibration process and the perforation pattern analysis were accelerated through programmatically generating models through the Abaqus Python API. Developing the method for simulating VIVAR Vantage dispensing performance provided additional value for the corporation after other paper towel and bath tissue product teams integrated modeling into their workflow.

Predicting Non-woven Web Compression Performance from Fiber Properties, Ted Tower - Kimberly Clark Corporation

Non-woven webs are a critical component to many of KC’s disposable absorbent articles. For example, some webs are responsible for fluid intake and distribution, and therefore must remain sufficiently open even under compressive loading. Material development can be accelerated by predicting functional web performance from experimental fiber data. A finite element model was developed to evaluate compression resistance of a non-woven web as a function of fiber properties. We will also outline examples of using an open-source computer graphics package, Blender, to more effectively communicate simulation results to the broader organization. With careful application, simple rendering approaches can help disseminate and market the use of simulation.

Squeezable containers: Improving the Consumer Experience, Sumit Mukherjee - Plastic Technologies, Inc.

Consumers are attracted to visually-appealing packages on the store shelf, but also expect a highly-functional container. Brand owners also need to make sure performance attributes such as top load and dispensability are delivered at the lightest possible weight. Products such as ketchup, mayonnaise, salad dressing, dish and hand soap, are just some product types that rely on their packaging to meet these key performance characteristics. It’s important to understand the various aspects of container processing, as well as material properties, package design, product characteristics and thickness distribution. Each of these factors can affect the ease of product dispensing from the package. The objective is to not show creasing and enable package recovery close to its original shape.

This study examined existing commercial packages and analyzed the design features that can be modified to improve the container squeeze ability. These geometric features involve structural components like ribs in the panel area and also the vertical and horizontal profile of the container design. The results of this study define the relationship between the container aspect ratio, label panel area, thickness distribution and cross section geometry. These relationships then define the optimum combination to maximize squeeze performance without compromising the other required specifications. The results show an effective way to arrive at a solution involving simulating the performance under realistic predictions of material distribution.

ENERGY, PROCESSES & UTILITIES & NATURAL RESOURCES

FER Modeling for Thermal Well Casing Connection Evaluation Protocol (TWCEP), Jueren Xie - C-FER Technologies Inc.

Thermal recovery methods, such as Cyclic Steam Stimulation (CSS), Steam Drive (SD), and Steam Assisted Gravity Drainage (SAGD), have been widely used in production of heavy oil and bitumen. The cyclic thermal loading with high peak temperatures may cause the casing and connection to deform plastically and therefore the casing connections are considered as one of the most critical components in thermal wells. Engineering evaluation programs, such as physical testing and numerical analysis, are often used to ensure adequate structural integrity and sealability of the connections over the full service life of a well. The recently developed Thermal Well Casing Connection Evaluation Protocol (TWCEP) specified connection evaluation conditions and requirements for thermal well applications. This paper starts with a review of the thermal well casing connection evaluation requirements, and presents Finite Element Analysis (FER) modeling considerations to assist TWCEP programs. The paper covers a few key topics for FER modeling, including the material modeling, determination of biased test population, impact of material properties, impact of make-up torques, and the determination of stiff-length for the testing program. Analysis examples are presented to demonstrate the use of the proposed approaches and methods.
Direct Energy Deposition (DED) or Laser Metal Deposition (LMD) is a group of additive manufacturing (AM) processes where material is injected into a focused beam of a high power laser or arc weld. The high intensity heat source heats the underlying material and creates a melted pool, where material is progressively added into the finite element simulation as the laser traverses the conventional AM processes. We have developed a simulation framework and models to support metal big area additive manufacturing via direct energy deposition: process validation, Srdjan Halliburton.

The transient temperature distribution of a quenched solid is driven by the convection coefficient at the wetted interface and the temperature difference between the component and the quenching fluid. During finite element analysis, accurate application of the convection coefficient is especially difficult due to the temporal evolution of the temperature during the quenching process. For example, excursions through multiple regimes such as nucleate boiling, nucleate pool boiling for laminar thermal boundary layers. Additionally, a constant-valued heat transfer coefficient is often used in lieu of the dynamic coefficient that accurately represents the convective boundary. Here, we present the application of an automated convection library to a quenched cylindrical body. The library is an add-on package to Abaqus and runs within the standard Abaqus environment. The library contains numerous heat transfer models that are defined functionally between convective regimes as determined by dimensionless fluid numbers (Reynolds, Rayleigh, etc.). Moreover, temperature and pressure dependent properties are used and post-processing visualization of all convection parameters is available in the Abaqus Visualization module. Our results are compared to a quenching analysis using a constant heat transfer coefficient. From the comparison, we show an improved prediction of residual stresses and a refined prediction of internal stress reversal that is known to lead to possible crack induced component failure.


Direct Energy Deposition (DED) or Laser Metal Deposition (LMD) is a group of additive manufacturing (AM) processes where material is injected into a focused beam of a high power laser or arc weld. The high intensity heat source heats the underlying material and creates a melted pool, where material is progressively added into the finite element simulation as the laser traverses the conventional AM processes. We have developed a simulation framework and models to support metal big area additive manufacturing via direct energy deposition: process validation, Srdjan Halliburton.

Improved Thermal Stress Prediction in Quenched Cylindrical Bodies through a Dynamic Convection Coefficient Library, Patrick Garrity - Naval Nuclear Laboratory-Knolls Atomic Power Lab.

The transient temperature distribution of a quenched solid is driven by the convection coefficient at the wetted interface and the temperature difference between the component and the quenching fluid. During finite element analysis, accurate application of the convection coefficient is especially difficult due to the temporal evolution of the temperature during the quenching process. For example, excursions through multiple regimes such as nucleate boiling, nucleate pool boiling for laminar thermal boundary layers. Additionally, a constant-valued heat transfer coefficient is often used in lieu of the dynamic coefficient that accurately represents the convective boundary. Here, we present the application of an automated convection library to a quenched cylindrical body. The library is an add-on package to Abaqus and runs within the standard Abaqus environment. The library contains numerous heat transfer models that are defined functionally between convective regimes as determined by dimensionless fluid numbers (Reynolds, Rayleigh, etc.). Moreover, temperature and pressure dependent properties are used and post-processing visualization of all convection parameters is available in the Abaqus Visualization module. Our results are compared to a quenching analysis using a constant heat transfer coefficient. From the comparison, we show an improved prediction of residual stresses and a refined prediction of internal stress reversal that is known to lead to possible crack induced component failure.

The first approach inserts the crack mesh into the full model and connects it with tied contact, solving the full model including the crack. The second approach uses a sub-model containing the crack, where the boundary displacements are obtained from the uncoupled full model results. The two methods are introduced and compared, first using a cylinder made of steel with an axial surface crack, which allows comparison to known finite element solutions. Then, a main inlet valve model shows a circumferential surface crack, which grows due to cyclic loading from the valve opening and closing. The sub-model results may differ from the full model as the crack size increases, but it runs more quickly and still provides a useful engineering solution. Model setup effort and analysis run times are compared to help consider when to use each approach.

Engineers using Abaqus benefit from having several approaches to model fatigue crack propagation. Computing accurate stress intensity values improves the crack growth rate calculations compared to more conservative crack models. This can help increase the expected life of the component, and could justify fewer inspections during the service life before repair or replacement is needed.

Multi-Stage, Multi-Wellbore Hydraulic Fracturing Simulation in Naturally Fractured Reservoirs Using Cohesive Zone, Mahdi Haddad - University of Texas at Austin.

Model Microseismic surveys have demonstrated the abundance of natural fractures where shear slippage occurs due to hydraulic fracturing. These natural fractures and their intersection with hydraulic fractures significantly complicate the optimization of hydraulic fracturing strategies especially in shaly reservoirs with multiple simultaneous or sequential stimulation stages. The clusters' hydraulic connection within a stage may substantially influence the hydraulic fracture pattern and the propagation rate. A case study of a well that was fracture stimulated using a high-performance frac job is presented. We simulate the fracture growth with a CCM-based model, where the fracture propagation pattern is controlled using a constant fracture propagation rate, and variable perforating efficiencies in planar fractures. Using a validated cohesive zone model, we simulate hydraulic fractures including the fracture mesh to compute the crack-front J-integral and stress intensity values, used to compute the crack growth rates.

The first approach inserts the crack mesh into the full model and connects it with tied contact, solving the full model including the crack. The second approach uses a sub-model containing the crack, where the boundary displacements are obtained from the uncoupled full model results. The two methods are introduced and compared, first using a cylinder made of steel with an axial surface crack, which allows comparison to known finite element solutions. Then, a main inlet valve model shows a circumferential surface crack, which grows due to cyclic loading from the valve opening and closing. The sub-model’s results may differ from the full model as the crack size increases, but it runs more quickly and still provides a useful engineering solution. Model setup effort and analysis run times are compared to help consider when to use each approach.

Engineers using Abaqus benefit from having several approaches to model fatigue crack propagation. Computing accurate stress intensity values improves the crack growth rate calculations compared to more conservative crack models. This can help increase the expected life of the component, and could justify fewer inspections during the service life before repair or replacement is needed.

Validation of XFEM-based Simulation Capabilities for Fluid-driven Fractures in Permeable Media, Sandeep Kumar - Upstream Research Company.

The extended Finite Element Method (XFEM) refers to a simulation technique where a fracture is modeled via splitting of special enriched finite elements during the course of a simulation. XFEM, unlike Cohesive Zone Method (CZM), allows simulating nucleation and growth of a fracture along an arbitrary, solution-dependent path without remeshing the material in the bulk. In this work, we have used XFEM, coupled with pore-pressure degrees of freedom, to simulate fluid-driven (hydraulic) fractures in a permeable medium under a variety of different scenarios. Specifically, through these simulations, we have investigated (a) the creation of fracture-tortuosity due to a misalignment between initial perforation and maximum in-situ stress direction, (b) the impact of in-situ stress anisotropy on the near-wellbore fracture geometry, (c) the deflection of fracture path due to a spatial pore-pressure gradient, (d) the refracturing of an already existing hydraulic fracture, and (e) the merging of a hydraulic fracture with a pre-existing fracture in the medium. For each of these cases, the predicted fracture geometry is compared with the experimental results available in the literature. The simulated and the measured fracture geometries are found to be in good agreement.
This paper provides an explanation for the artificial damping method, taking the snap-through buckling of a two-bar truss as an example. The analytical results for the problem using the arc-length method, the static artificial damping method, and the arc-length method with artificial damping are compared to experiment. Additionally, a nonlinear viscoelastic UMAT is used to predict large deformation behavior and do more with the data we collect.

Challenges with Creep in Under Hood Routomatic Parts, Alex Rzoumanidis - Psylotech, Inc.
This work leverages a frequency domain dynamic mechanical analysis to infer time domain creep at any loading history and any temperature history. Under hood polymer parts experience creep failure. Creep experiments are time consuming and expensive. Moreover, effects on creep are accentuated when polymer parts are exposed to accelerated environment test matrix, further driving up costs. An approach to creep simulation in Abaqus is presented, where frequency domain experimental data is shifted using time-temperature superposition to create the material’s time dependent response. Frequency domain master curves and their associated thermal shifting factors form the foundation for a reduced time material model capable of simulating creep with any combination of temperature and loading history. Moreover, the frequency domain experimental approach can be completed in a few hours, saving time and expense.

This presentation is focused on in-car vibrations and shows how KONE uses Abaqus in the chain of multi-physics simulations. The ride comfort depends on the elevator design, running parameters, installation quality, as well as the building behavior. From a finite element point of view the important challenges are:

- misalignment of the guide rails which typically are in the range of millimeters with a 10 cm distance between the peaks and lengths over 500 meters;
- compensating travelling cables introducing variable mass acting on the sling as a function of car position;
- suspending cables introducing variable forces as a function of car position;
- wind induced forces caused by the counterweight passage at high speed;
- sway shape and magnitude of the building; this induce guide rails variable movement.

Therefore, a special user defined elements has been developed to include guide rail geometry and movements as well as the varying load case of the rope. The advantage of this approach is that the elevator dynamics and stresses/deflections for the entire structure can be analyzed for every elevator component: car, sling, roller, roller’s stopper, for the entire travel.

Creating Effective Use of Simulation Results for Critical Design Decisions, Charlie Wood - Motorola Mobility
The growth and maturity of finite element simulations have influenced the way people design new products or even identify root causes to observed failures. In many industries, the size and scale of the models are large and produce an incredible amount of data over the life of the program or activity. Similarly, the amount of computational power available today is unmatched. Often, this leads to a situation where it can difficult to navigate through all of the data and arrive at a consistent and accurate conclusion. This is especially true when facing a design challenge in which simulation is a perfectly suited tool. However, without linking critical features of the model (such as material choice or some critical dimension in a complex part) to outcomes, decisions can be more conjecture than scientific. In this talk, we will go over some techniques to help aide in making better decisions at critical junctures using simulation results. These probability based analyses will help formulate our simulation test plans with more direction and more with the data we collect.
SIMULIA CUSTOMER PRESENTATION ABSTRACTS

Front-End Loader Linkage Durability Study Using Load Input from True Load, David Slowinski - CNH Industrial

This document describes the use of Abaqus/CRE, Rbaqus/Standard, True-Load, and Verity yield analysis in finite-element software to evaluate the durability of a front-end loader linkage. Dynamic field loading for FEM of construction equipment is not always easily replicated by traditional linear static loading methods. Wolf Star Technologies True-Load software along with unit load FER in Rbaqus was used to determine time history loads for a field operation with dynamic loading that resulted in fatigue failures of the linkage. The time history loading was then used in finite-element analysis of design alternatives and determined a design that met the simulation criteria. The methodology greatly improved the confidence of the design developed through simulation because the field loading was accurately reproduced.


Back extrusion processes provide an alternative to open-die upset forging processes for producing rings and cups due to its significantly less material loss. Rbaqus/Explicit was used to understand the strain distribution in a niobium (Nb) material under such bulk deformation processes. Adaptive meshing using arbitrary Lagrangian-Eulerian (ALE) method was employed to successfully simulate the back extrusion process. Die geometry for the back extrusion process was optimized for producing a part with uniform strain distribution within it. It is observed that greater strain uniformity is achieved by modifying the conventional back extrusion process by creating a notch on the outer walls of the die around which the material deforms. More strain uniformity is achieved through such a deformation process that the conventional back-extrusion process lacks. This may further lead to more microstructural uniformity within the part.

Development of the Subroutine Library 'UMMDp' for Anisotropic Yield Functions, Kai Oide - Mechanical Design & Analysis Corporation

To complement improvements in metal manufacturing technology, metal-forming simulations using advanced anisotropic yield functions are required in addition to common constitutive laws, especially in high-value-added application fields. Advanced finite element codes, such as Rbaqus, make it possible to apply anisotropic yield functions via user subroutines. However, this kind of technique requires securing human resources with professional skills and knowledge, and it's not yet widely available. Over the last 10 years, the Japan Association for Nonlinear Computer Aided Engineering (JANCRE) has developed a subroutine library, including not only practically recognized anisotropic yield functions (e.g. Yoshida's 6th-order polynomial, Bralat's Yld2000-2d and Yld2004, Cazacu's CPB2006, Vegter's spline), but also subroutines that provide multi-port capability to any finite element codes in order to implement these yield functions. In this case of Rbaqus, a framework for a UMRT subroutine is already prepared in the library as a common subroutine.

The working group of JANCRE sprang from the idea of providing a multidisciplinary forum for steel researchers, design engineers, and software engineers through the medium of advanced finite element codes. The subroutine library is called the Unified Material Model Driver for Plasticity (UMMDp). This paper introduces the development of UMMDp and discusses a simulation of a drawing and redrawing process using some yield functions for its applications with Rbaqus.

Mechanical Testing of FDM Parts for Process Simulation, Siddhartha Dev - Stratasys

Fused Deposition Modeling (FDM) enables the development of functional prototypes, tooling molds and fixtures of complex geometries which are difficult to manufacture using conventional manufacturing technologies such as injection molding. In the FDM process, thermoplastic materials such as ABS and ULTEM are extruded as filaments and deposited layer-by-layer to build a part directly from a CAD model. In this study, the relationship between the layer-by-layer build process and the resulting mechanical properties is investigated. The build orientations are selected based on the assumption that the printed parts demonstrate orthotropic behavior. This assumption is validated by mechanical testing of tensile and Izod specimens fabricated with layer-by-layer orientations orthotropic to the loading axes. Properties that are obtained from the mechanical testing include tensile modulus, tensile strength, shear modulus and Poisson’s ratio. In addition to mechanical testing, virtual testing is also performed using a three-dimensional Representative Volume Element (RVE). The Rbaqus micromechanics plugin is used to homogenize the linear elastic material constants. Periodic boundary conditions are used to represent the boundary effect of the microstructure in the printing process. Neither experimental nor virtual testing involve any empirical coefficients. The influence of the bead aspect ratio, bead volume fraction and geometry of the air gaps between the beads is investigated. A scaling factor is calculated to identify the correlation between the test material properties and simulation-driven results. Good agreement is found between experimental and virtual testing. The testing results are readily usable for part-level FDM process simulation and in-service performance simulation.

Predicting the Properties of Additively Manufactured Parts, Tyler London - TWI Ltd

Selective laser melting (SLM) is an additive manufacturing (AM) process whereby a laser produces a three-dimensional part by successively fusing regions of a metal powder bed in a layer-wise manner. SLM has the ability to produce complex, light-weight, metal parts with the potential to reduce costs and lead times over conventional manufacturing processes. However, the layer-by-layer nature of the SLM process introduces an almost unique thermal history at each location within the part as subsequent laser passes reheat the material. As a consequence, strong microstructural anisotropy and hence location-dependent material properties are features of SLM builds. The ability to simulate and predict the distortion and residual stress at each location within the SLM build that arise from this process is therefore crucial to establish SLM as a robust and reliable manufacturing route.

In this paper, thermo-mechanical-simulation models of Ti-6Al-4V parts produced by SLM are validated against experimental measurements. The work involves the simulation of the SLM process; the prediction of location-specific microstructural features (such as grain size, morphology characteristics and phase fractions), the implementation of a novel mapping between microstructural quantities and tensile properties at each material point, and the simulation of the bulk tensile behavior of test specimens removed from the SLM build. The results show the potential for Rbaqus to accurately reproduce the behavior of SLM parts and therefore enable process optimisation and the control of mechanical properties.

LIFE SCIENCES

Biomechanics of Craniofacial Fractures: A Simulation Study, SHOBHR E.S. - Dayananda Sagar College of Dental Sciences

The objective of the current research is virtual simulation of cranio-facial fractures under impact loading to understand its biomechanics. In particular, the focus of this work is on the complex part of a human body, the skull. One of the emerging areas of applications of computational biomechanics is to understand the behavior of the skull during a traumatic head impact due to its head impact accidents.

In this study, a 3D finite element model of the skull is created using CT scan data. All complexities of the skull geometry are simulated using Abaqus. This numerical model is then subjected to frontal, lateral, vertical, occlusal and angular impact load. Impact analysis is done and weak areas susceptible to fracture and hence failure is identified. Further fracture situations of various designs and materials used in cranio-facial fractures are also considered. The results show different fracture situations in the virtual model and subjected to different impact load conditions. This will enable the study of fracture and stability of the fracture of the skull under cranio-facial fracture conditions.

Results from the analysis can be used to come up with optimum locations of implants for different types of impact situations. This is expected to complement the existing treatment methodologies used by surgeons. Further, appropriate knowledge of fracture biomechanics can be used to develop safety measures in automobiles to prevent and reduce incidence of facial injuries.

Example Nonparametric Optimization Cases for Additive Manufacturing Using Tosca and Rbaqus, Juan Betts - Front Analytics LLC

Two example non-parametric optimization cases for Additive Manufacturing using Tosca and Rbaqus are presented. The first case is the optimization of a hip implant and the second case is the optimization of a disc brake caliper.

To optimize the hip implant, we evaluated the stresses between the femoral stem and its head during an extreme side loading condition. The paper discusses the loading requirements, envelope limitations for a given patient size, and the boundary assumptions for the entirety of the implant. Using Tosca embedded within Rbaqus, we established an objective function, the desired constraints, and the targeted design response, which drove the outer topology of the implant stem. The topology optimized geometry provided the basic geometry, and a subsequent shape optimization reduced the stress concentrations in the transitional areas between the stem and the head.

Similarly, for the second case, we optimized a disc brake caliper, where loading conditions were driven by the master cylinder, the contact areas of the disc rotor, and the envelope limitations within the wheel. The objective function, design constraints, and the targeted design response drove the outer topology of the caliper casting. Just as before, the initial topology optimization provided the primary geometry, and the subsequent shape optimization reduced the high stress concentrations around the corners of the caliper.
Trauma to the spinal column can result in severe injury, often leading to paralysis or death. In order to facilitate Three Dimensional Modeling and Finite Element Analysis of the Human Spine and Axial Skeletal Other than Tosca which has radii associated with beams, Element Free has nodal radii. To deal with this difference, Element Analysis (FEA) on the spine under dynamic loads. Loading was applied to approximate real world traumatic events in order to study the response of the skeletal system, and to study the effect of loads on the spinal cord and brain.

Personalized Medical Devices: Contact Lens, Robert Stupplebeen - Optimal Device

Personalized medical devices require 3 enabling technologies: biometric measurements, automated simulation and customized manufacture. Every customized device be it an external prosthetic or an implantable device will follow a similar workflow. For the purpose of this paper, a customized contact lens design will be investigated. A contact lens in essence is an optical suction cup which corrects the abnormalities of the patient’s vision. The lens surfaces with the anterior surface of the cornea by way of a suction pressure. This suction pressure is balanced by the energy stored in the deformed shape of the contact lens. The lens is free to move on the cornea finding its equilibrium position (which will be at the first order).

Many patients can be served by the shelf lenses however for some patients this does not work. There are a variety of conditions that could cause this. Trauma can cause irregularities. Severe astigmatism, where the optical power of the eye is different as you go around the eye. Keratocones, where the cornea thins and sags. The current standard of care, for these cases, has an eye care practitioner either optometrists (O.D.s) or ophthalmologist (M.D.s) measure the refraction error and the central corneal curvature with a keratometer. Working with custom contact lens labs, which CNC each lens individually with a diamond tool. They will design a contact lens for each unique patient through diagnostic fitting lenses, intution and trials. In this study we are investigating the use of corneal topographer data, simulation and optimization to improve the outcomes and reduce the iteration. From the data from the Eye Eye ESP corneal topographer has been imported into CATIA using Digitized Shape Editor. NURBS geometry for the corneas and white scleera were created with Quick Surface Reconstructon also within CATIA. Various contact lens geometries have been modeled in CATIA, SolidWorks and Rbaqus/CRC. This simulation includes multiple non-linearities such as hyperelastic materials, large deformation and sliding contact. Isight has been used to automatically modify lens geometry, optimizing comfort and fit of the contact lens.

Solution from Lattice Sizing Optimization to Additive Manufacturing, Luyao Cai - Purdue University

Lattice structures bear many desirable characteristics from design standpoints, such as better performance for stability, desirable weight characteristics, custom mechanical behavior and porous nature that could facilitate bone and tissue growth on medical implants. To efficiently optimize the structure of lattices, we can use Tosca software to vary the lattices to load for a total of several design objectives. However, the radii in Tosca are associated with beams which are not optimal to have a smooth stress transition on connected nodes. Additionally, the optimized results cannot be meshed smoothly enough in Rbaqus for additive manufacturing. In order to avoid these limitations, we proposed a workflow combining the Tosca structure and Element Free (nTopology) by using python scripts. As a demonstration, a sphere was created in solid. By meshing the sphere in Rbaqus and Element Free, a basic lattice structure with internal supporting beams was created and imported into Tosca for sizing optimization. Other than Tosca which has radii associated with beams, Element Free has nodal radii. To deal with this difference, the optimized results were converted into nodal radii models in Element Free. Then a STL file was created in Element Free based on the converted results. Additionally, a method to design the initial PVE lattice structure using topology optimization was also mentioned for those who might be interested. This whole process enabled the additive manufacture on optimized lattices and reduced the design time to one or two hours.

Parametric Study to Evaluate the Effect of Strut Geometry on PLLA Coronary Stent Recoil, Ross Blair - Queen’s University Belfast

Bioresorbable poly(l-lactic acid) (PLLA) stents present a clinically attractive treatment for coronary heart disease, providing tissue responses that result in a development of the arterial lumen. However, the use of light weight material in both vehicles and medical devices has been a major concern in the medical device’s frame standards that OEMs develop most less-fuel-efficient vehicles with lower emissions. To keep pace with the radical advances in powertrain technology, advanced cylinder head gaskets (CHG) are needed to achieve higher performance expectations by accommodating higher temperature exposure, the use of Aluminum
alloy hardware instead of traditional cast iron hardware, and extended warranty guidelines from the OEMs. Along with technological advances, time to market and first time right initiatives are pushing the development of CHG to utilize new levels of simulation and optimization processes. Increased use of insight for optimization of forming processes and stopper height along with improved analysis techniques for fatigue prediction enable the design of more robust CHG. This paper demonstrates how simulation is driving the complex development of technologically advanced CHG.

**Fatigue Life Prediction Techniques for Polymers and Polymer Matrix Composites, Amir K. Shojaei - DuPont**

Fatigue damage mechanisms in Polymers and Polymer Matrix Composites (PMC) are unlike metal fatigue mechanisms. In the context of metal fatigue methodology, once the fatigue crack is initiated, most of the functional life of the component is terminated. The crack initiation defines the lifetime for most of metallic structures. However, thermoplastic polymer systems are much more resilient compared to metals and they demonstrate more energy absorption properties compared to metals. Based upon these key differentiations of fatigue damage mechanisms, applicability of prediction techniques, which have been developed for metal fatigue, in PMC structural analysis is questionable. This work aims at summarizing the fatigue prediction technologies for polymer based material systems. In the framework of DuPont’s Advanced Predictive Engineering, the presenter has developed a fatigue computational platform that utilizes the Continuum Damage Mechanics (CDM) framework to link microscale damage mechanisms to macroscale fatigue fractures. The developed computational tool provides full coupling between elastic, plastic, thermal, damage and microstructure response of material and is implemented in Rbaqs through user-defined coding. Both feasible and the developed framework are utilized for lifetime prognosis. Pros and cons for fatigue prediction methods are discussed and some recommendations for future developments are proposed.

**State of the Art Hytrel® Material Modeling Development for the Design of Jounce Bumper, Helga Kuhlmann - DuPont**

This paper describes the material testing and development of a constitutive material model of Hytrel® thermoplastic elastomer (TPE-ET) used in application such as a Jounce bumper. The Jounce bumper is an important and active part of a vehicle’s ride behavior and handling and maintains a high level of comfort. It also offers additional spring function to improve vehicle behavior when negotiating smaller holes and irregular road surfaces such as cobblestones, or during sudden braking and quick change in direction. The part is expected to operate in the most demanding conditions while exhibiting good durability and dynamic fatigue and impact resistance. Indeed, long term jounce bumper reliability is essential to the vehicle’s road handling.

The objective of this program is to develop an accurate material model for the Hytrel® Jounce Bumper that addresses the following customer requirements/demands:

- Accurate prediction and design of the response curve from 0 to 500 or 1000 N loading, with the aim to improve the ride behavior of the suspension
- Prediction of the change in slope and second derivative to remain within the OEM requirements
- Allow re-design of the lip to avoid noise and response issues

The material model is implemented in Rbaqs using parameter optimization techniques available through PolyUMod® software. A hyper-elastic, visco-elastic material model, including permanent set and Mullins effect is implemented, which is an improved model from the hyper-elastic material model presented at the 2016 SIMULIA conference. Numerical results are compared to coupon test data as well as test data from an actual component.

**Analysis of DuPont Engineering Polymers - Challenges and Solutions, Pieter Volgers - DuPont**

Modern application development and design depends even more on more accurate predictive engineering methods. Critical in these analyses is the behavior of the material and its corresponding constitutive behavior. Engineering polymers demonstrate complex behavior, where traditional isotropic elastic-plastic models are often no longer acceptable for the precision required.

DuPont is one of the major suppliers of thermoplastic engineering polymers providing support and knowledge to its customers in the best use of its materials. Understanding and developing the state-of-the-art of material modelling of its portfolio, as well as the best use for the specific problem at hand, is key to this. These material models should capture the essential behavior of thermoplastic polymers, while at the same time avoiding too complex material characterization and computation times for reasons of efficiency and cost. In addition to state-of-the-art predictive engineering, material testing capabilities and data management must meet the data generation needs.

This paper describes the material behavior for different families of materials of the DuPont product offering, the performance of different material models for these materials using the extensive possibilities available in Rbaqs and the collaboration between DuPont and SIMULIA to advance the developments for a more accurate prediction of their behavior.

In addition to the modelling, the challenge of resource-effective data generation, state-of-the-art laboratory testing and corresponding data management on a global scale is discussed.

**Dynamic Simulation Methodology of Half-toroidal CVT Variator System with Feedback Control, Toshihiro Saito - Honda R&D Co., Ltd. Automobile R&D Center**

Dynamic behavior analysis technique for a half-toroidal variator system using a traction drive was developed. The frictional characteristics of a traction fluid dependent on surface pressure, sliding velocity, and contact surface temperature were identified from the dynamic behavior of the finite element analysis for a four-roller test apparatus together with experimental data. Since half-toroidal variator system was required to analyze simultaneously the vibration, displacement, and force and pressure of the variator system resulted from dynamic contact, Rbaqs/Explicit solver was used. Additionally, this approach provides arbitrary control of the variator speed ratio, a mechanical sensor-less control logic that combined a state estimation observer with feedback control was coupled with the Rbaqs/Explicit solver. These methods make it possible to quantity the torque capacity and power transmission efficiency of a half-toroidal variator and also enabled visualization of the traction surface frictional force spinning behavior and resulting heat generation.

**Improvement of Piston Skirt Scuffing Problem using 3D Piston Motion Simulation, Sangwoo Cha - Hyundai Motor Company**

This paper shows a study for solving a piston skirt scuffing problem in diesel engines. We propose to solve this scuffing problem by studying the piston dynamics analysis using Abaqus. The first approach was to develop the piston secondary dynamic analysis method based on the Finite Element Method. The second approach was to clarify the root cause of the diesel piston scuffing case. The last approach was to optimize the piston skirt profile and cylinder block design for reducing the scuffing index. The solution proposed in this study was adopted in the piston and block design.
FlowVision & Abaqus 2-Way Strongly Coupled FSI Simulation of Automobile Tire Aquaplaning, Ming-Feng Sung - Kenda Rubber Ind. Co. Ltd

Aquadplaning of automobile tires simultaneously incorporates various physical phenomena some of which are challenging to model with computer-aided simulation tools. Among those are complex patterned topology and rotational movement of tire, small clearances between ground and tire, free surface flow of water and air and inevitable the elastic nature of tire body and effect of its deformations on fluid flow characteristics.

In order to overcome these challenges, Abaqus FE software from SIMULIA and FlowVision CFD package from Capvadia are strongly coupled for bidirectional fluid structure interaction (FSI) co-simulation. This approach is applied to a 205/55/R16 sized tire and same conditions are realized in experimental environment for comparison purposes. Simulation and experimental results are in well agreement with each other and the corresponding outcomes are as bellow: maximum pressure build up is at the tire front and once this pressure starts to stabilize by lateral groove, longitudinal groove loses capability of effective draining. Additional observation is that the aquaplaning speed of tire decreases in correlation with an increase in the number of lateral grooves.

Welding Spot Simulate in Compression Analysis of New Energy Vehicle Battery Package, Lyu Guang - Pan Asia Technical Automotive Center

As the core component of the new energy vehicle, ensuring the security of the battery pack is an essential part during the design and development of new energy vehicles. The compression analysis of a certain model battery package has been conducted. Considering the complexity of battery package assembly and the limitations of the elements type in Abaqus/Explicit, this article use C3D8R to simulate the welding spot. The result shows that the stress around the welding spot is well distributed. And this method can effectively avoid large local stiffness and grid dependency problems compared to traditional method.

A New Fatigue Analysis Method for Bracket of Engine Exhaust System Based on Abaqus Scripting Interface, Deyuan Xu - Pan Asia Technical Automotive Center

As the working condition of engine exhaust system bracket usually involves long time high temperature and intense vibration, its structure strength and fatigue life need to be evaluated and optimized by simulation in the early stage of engine design. Using thermal and dynamic stress analysis module in Abaqus/Standard, the thermal and dynamic stresses of the bracket are firstly obtained. Based on the Python scripting interface of Abaqus and Hashtrid diagram of the bracket material, safety factors of bracket elements are calculated using element stresses automatically extracted from ODB files. In comparison to the third party fatigue analysis software, this method is not only fast and convenient, it is also beneficial for the subsequent closed-loop optimization process since its calculation engine and data storage plan share the same Abaqus platform.

Modelling Rubber Bushings using the Parallel Rheological Framework, Javier Rodriguez - Principia

Bushings are anti-vibration components used in vehicle suspension systems. They are typically made of rubber between two concentric metallic cylinders. Their analysis requires characterizing their complex impedance (dynamic stiffness and phase angle).

Their behavior is often difficult to predict without testing a prototype. The rubber undergoes large strains, usually involving strain stiffening and Mullins effect, time dependency and, possibly, plasticity. Thus the behavior depends on the loading frequency and amplitude, as well as the preload. To tackle the problem a methodology has been developed that combines Abaqus and Isight.

From the constitutive models in the material library the parallel rheological framework (PRF) is the more appropriate one for the rubber. It is based on a hyperelastic material model and consists of multiple parallel viscoelastic networks. The hyperelastic model is determined by evaluating strain energy potentials for a uniaxial quasi-static tensile test, dynamic tests are used to calibrate the parameters of the PRF model. The process combines Abaqus analyses on a single-element model with Isight runs for minimizing the differences between test and model results.

Based on this a model of the bushing is constructed and used for predicting its behavior under static and dynamic loads, self-contact, axial and radial displacements, and torsional and cardanic rotations. The model takes into account the assembly loads produced by thermal shrinkage and a subsequent plastic reduction of the radius.

Stress Analysis under Random Loading, James Guo - Robert Bosch LLC

To assess the durability of a structure under random loading, the frequency based random response analysis is performed to obtain the stresses. Since random response analyses ignore the element damping, the results of the random response analysis can be unrealistically twisted if significant local element damping exists. This paper presents an Abaqus FEA procedure to more realistically calculate the stress of the structure subject to random loading by submodeling and including element damping, which is a complementary method to the Abaqus random response analysis procedure.

Modelling of Accurate and Fast Heat Transfer Analysis using FILM Subroutine, Abhishal Puthigut Meethal - Robert Bosch LLC

In typical engineering applications, mixed convection is simulated using an assumed convection Heat Transfer Coefficient (HTC), but in reality, the HTC depends on various parameters like the orientation of geometry, the flow regimes, the temperature of the body, and the properties of the fluid. Hence the simulation using assumed HTC is less accurate. The thermal analysis can be made more realistic by taking into the account of variation of HTC due to several factors. A CFD analysis or a conjugate heat transfer analysis computes the HTC accurately but a major limitation is higher run time. On the other hand one can predict the HTC by using various correlations and dimensionless numbers. Keeping this in mind, a user subroutine (FILM) computing temperature state dependent HTC’s has been developed and implemented in Abaqus. This results in accurate and fast uncoupled heat transfer simulation without making assumptions.

For natural convection the Nusselt’s number is a function of Grashof’s number and Prandtl number, and for forced convection the Nusselt’s number is function of Reynolds’ number, Grashof’s number and Prandtl number. Where Grashof’s number and Prandtl number are functions of surface temperature and properties of fluid around the body. Once Nusselt’s number is known, HTC can be estimated by the characteristic length of the geometry and the conductivity of the fluid. These temperature dependent functions are fed into the FILM subroutine and recalculated the HTC at every increment of the transient heat transfer analysis. The advantage here is faster analysis compared to the conjugate heat transfer analysis. Thus, the complex convection phenomena are easily included in the simulation and enhanced the functionality of Abaqus.

This paper describes in detail how to calculate the temperature dependent HTC using FILM subroutine and the simulation results comparison with conjugate heat transfer analysis carried out in Abaqus.

Optimization of conformal cooling part by SIMULIA additive manufacturing solution, Wei Zhang - Shanghai Behr Thermal System Co., Ltd

Topology optimization by Tosca , Multi parameter optimization by Isight , 3D printing process simulation by Abaqus, Conformal cooling channel simulation and optimization by 3DE FLIR.

Reliability Based Design Optimization of Sub-systems / Component using Monte Carlo Simulation of Isight, Saxtry Bonala - Tata Consultancy Services

With the advent of HPC computing, both non- parametric and parametric optimization methods are gaining rapid momentum in order to find the optimum design that satisfies the performance requirements. Usually, each input design variable such as material property, load and manufacturing processes of a component / sub-system will have uncertainties which have to be accounted while performing the optimization. Since, the output parameters of mechanical sub-systems are highly nonlinear and sensitive to variation of input design variables. The Monte Carlo Simulation (MCS) method of Isight has been used to define variation of input design variables thereby to assess the risk factor of the optimum design. With the help of MCS method, the product designer was able to quantify risk involved in the optimum design instead of a qualitative assessment of the design. This paper also compares Robust Design Optimization (RDO) and Reliability Based Design Optimization (RBDO) formulations. This paper will summarize benefits of evaluating risk factor of the optimum design.

Development of the High-performance Bushing Model using Abaqus, Satoshi Ito - Toyota Motor Corporation

To obtain sufficient accuracy for analysis to predict vehicle dynamic performance, it is necessary to consider amplitude and frequency dependence as well as preload effect to the rubber bushing model. FE models are capable to reproduce these characteristics though it is difficult to obtain sufficient calculation accuracy within practical computation time. Therefore this paper proposes a new low DOF bushing model by utilizing the user subroutine of Abaqus. The new bushing model exhibited high performance maintaining its precision.
Vehicle Front-end Shape Optimization for Pedestrian Injury Risk Reduction, Atul Gupta - Waymo (Google self-driving car)

Vehicle front-end shape plays an important role in the pedestrian injury outcome. Prior studies on the effect of front-end vehicle geometry on pedestrian injury risk have focused on simple car profiles with uniform cross-section, lower fidelity pedestrian models and basic injury metrics. In this study, an Isight framework was developed to automatically sweep through pre-processing, analysis, and post-processing phases. In the pre-processing phase, a parametric car front-end shape is generated based on 48 parameters defined within Abaqus/CAE model to generate a wide variety of car shapes. In the analysis phase, vehicle-pedestrian impact simulations are conducted through a co-simulation interface with Madymo (multibody dynamics software used in pedestrian modeling). Finally, a detailed injury risk prediction for each body region and generation of a composite injury score, based on methods like Functional Capacity Index (FCI) and New Injury Severity Score (NISS), is performed using Matlab post-processing scripts. The framework was used to run a DOE of approximately 200 car shapes in simulated 25mph impacts with both adult and child pedestrian models. The results from the DOE were processed to determine the optimal car shape which would lead to significant reduction in pedestrian injury risk by simply varying the shape of the car front-end. This framework could be extended further to incorporate optimization of hood and bumper structure, material choices, and active protection mechanisms to further reduce pedestrian injury risk.

*For the purpose of this guide book, only first authors are listed.*