Your Essential Guide to the 30th SIMULIA UK Regional User Meeting

Park Royal • Cheshire, UK

Conference Dates: October 18-19

www.3ds.com/company/events/uk-regional-user-meeting/overview/

3DEXPERIENCE®

DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Contents

Your Personal Invitation

You're invited to join and celebrate the 30th Anniversary of the SIMULIA UK Regional User Meeting at the Park Royal Hotel, Cheshire.

We have once again created this e-Book full of all the vital conference information you need to make the decision to join us on the 18-19th October. We hope this e-Book will help to prepare you for the RUM and we look forward to welcoming you in Cheshire.

The preliminary agenda is now available and it is shaping up to be one of our best events yet. Due to the fantastic feedback received from last year's new conference format we will continue this into 2017.

Summary of what to expect at the conference:

- Registration to both days of the RUM will again be completely free
- Special Guest speaker from QinetiQ
- Learn from more than 16 customer paper presentations
- Choose from three advanced seminars; Topics include; Abaqus/CAE and Solvers , Tosca for Abaqus, Linear Dynamics with Abaqus
- Attend "Tune it all out" and learn the fundamentals of CST 3D EM modeling for EMC
- Hear how to integrate simulation with design from our CATIA colleagues
- The latest updates on the SIMULIA portfolio product suite
- See how to increase the value of Simulation Processes and Data using SIMULIA on the **3D**EXPERIENCE platform
- Attendance of the interactive exhibition area including academic showcase
- Free networking banquet on the evening of the 18th October

As this is a local conference, we are able to tailor the content towards local trends and industries, making the conference more relevant to UK users. This conference allows attendees to engage with other local users enabling them to share knowledge and experiences making networking more relevant and valuable.

2016 Conference Photo

Venue & Accommodation

This year the conference will be held at The Park Royal Hotel, Cheshire. More details regarding the venue may be found at http://www.qhotels.co.uk/our-locations/the-park-royal/.

To reserve your accommodation please contact the hotel quoting 'Dassault Systemes' in order to receive the discounted rate of £115 for bed and breakfast.

It is vital that you make your reservations as soon as possible as the discounted rate is available for a limited time only.

The Park Royal Hotel
Stretton Road
Stretton, WA44NS
Cheshire, UK
T. 01925 730 706

Registration

Continuing the success of last year's conference we are once again making the conference free to attend for both days including the conference banquet. Spaces are limited and due to the high number of expected attendees, it is essential that you only reserve a place if you will be attending and let us know as soon as possible if you are unable to attend.

Accommodation is expected to sell out fast. Please ensure you book directly with the hotel on 01925 730706. We have reserved rooms at the discounted rate of £115 B&B. These rooms are on a first come first served basis so please ensure you book as soon possible, quoting Dassault Systemes.

Registration link: http://www.3ds.com/events/simulia-regional-user-meetings/uk-regional-user-meeting/registration/

Technical Stream – Advanced Seminars

This stream has been specifically designed for SIMULIA users and provides the opportunity to learn from our specialists on a variety of advanced topics:

1. Topology Optimization using Tosca for Abaqus

When used in combination with finite element analyses, topology optimization techniques can help you improve your designs to produce lightweight, strong and durable components. In addition, using structural 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.

2. Hands-on Linear Dynamics with Abaqus: Effective Solution of Structural Vibration Problems

Linear dynamics has become standard practice in many industries to increase user comfort, increase product efficiency, decrease fatigue, and improve the overall experience of engineering products. 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 extract eigenmodes 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 eigenmodes, 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

3. Abaqus/CAE and Solvers

With Abaqus/CAE you can quickly and efficiently create, edit, monitor, diagnose, and visualize advanced Abaqus analyses. The intuitive interface integrates modeling, analysis, job management, and results visualization in a consistent, easy-to-use environment that is simple to learn. Abaqus/CAE supports familiar interactive computer-aided engineering concepts such as feature-based, parametric modeling, interactive and scripted operation, and GUI customization. Users can create geometry, import CAD models for meshing, or integrate geometry-based meshes that do not have associated CAD geometry.

Associative Interfaces for CATIA V5, SolidWorks, and Pro/ENGINEER enable synchronization of CAD and CAE assemblies, rapidly updating model with no loss of user-defined analysis features. Abaqus/CAE also offers comprehensive visualization options, enabling users to interpret and communicate the Abaqus results.

In this course, a general overview of the important features available in Abaqus/CAE will be covered, including creating parts using the feature-based modeler, importing parts into Abaqus/CAE, partitioning parts, meshing, defining analysis attributes, submitting and managing Abaqus simulations, and viewing the results of the simulations. This course is also intended to complement the Introduction to Abaqus Standard and Abaqus/Explicit course, which briefly describes how to perform linear and nonlinear analyses with Abaqus.

"Tune it all out" – Fundamentals of CST 3D EM modelling for EMC

Traditionally, Electromagnetic Compatibility (EMC) and Electromagnetic Interference (EMI) issues are often solved in an EMC lab. EMC is addressed when there is a problem or when a product is taken for certification. The adoption of 3D field simulation at the beginning of a project provides an insight into the root causes of electromagnetic resonance effects through field visualization and animations. EM simulations also help towards compensating and mitigating EMC issues, study for "what if" scenarios and develop best practices for fast design cycles and high product quality.

The main objective of this workshop is to provide a learning opportunity for those that are new to EMC/EMI as well as provide online demos and workflows to those who already have experience in this area. Topics included in the workshop are emissions, printed circuit boards, shielding, cables and EMC filtering.

Topics covered in the workshop (includes online demos):

- CST EMC Simulation for Early Stage Analysis and Troubleshooting
- Conducted emission of a motor control and a DC-DC converter for automotive applications
- Radiated emission of a DC-DC converter and the potential coupling to automotive AM/FM antenna
- Cable Simulation workflow
- Co-Site EMI Simulation Technology and Workflow
- Signal Integrity and Radiated Emission of Flexible Printed Circuits

SIMULIA Strategy Stream

This year we will again be offering a Strategy Stream to discuss how new technologies provide opportunities to drive process change in the deployment of simulation and the resulting business benefits.

Overview

In many companies the value of using advanced simulation technologies, such as nonlinear structural mechanics and computational fluid dynamics, to facilitate virtual product testing is well established. The need for physical prototypes has been reduced, and in some cases removed, from the product development process. Simulation is associated with savings in time, cost, improved product quality and can be regarded as a mature technology.

However, these valuable simulation tools are still largely deployed at the departmental level, as personal tools and there is a lack of management of the associated processes and data as a strategic asset.

In addition technology has been available for many years to augment the value of simulation through:

- Greater integration with the Design Process
- Automation of Simulation Processes
- Management of Processes and Data
- Automatic Optimisation and Trade Studies

These technologies have been shown to greatly increase the value of simulation. However they have yet to gain the wide usage of core advanced simulation technologies. Attend the Strategy Stream to:

- Explore the reasons for the limited take up of these technologies
- Review the opportunities to expand the value of simulation from the Department to the Enterprise
- Introduce the concept of using a Platform to manage Simulation Processes and Data
- Explore the business benefits of this approach
- Understand how new SIMULIA cloud based solutions can aid rapid deployment of new Simulation processes
- See new platform based solutions for Additive Manufacturing, Multi-scale and Multi-Physics Simulation and Virtual Human Modelling.

This session is suitable for attendees from organisations of all sizes and aimed at those responsible for the management and execution of simulation processes, managers of Simulation Practitioners and those responsible for Engineering Business Improvement.

Final Agenda – 18th October

9:00	Welcoming Remarks					
9:15	SIMULIA Executive Remarks					
9:45	Keynote 1					
10:15	Delivering Sustainable Innovation with the **3D**EXPERIENCE platform					
10:45	Break					
Strategy Stream		**Technical Stream**			**"Tune it all out" – Fundamentals of CST 3D EM modelling for EMC**	**Integrating simulation with design using CATIA**
11:15	SIMULIA cloud solutions	Topology Optimization using Tosca for Abaqus	Hands-on with Linear Dynamics with Abaqus: Effective Solution of Structural Vibration Problems	Abaqus/CAE and Solvers	• CST EMC Simulation for Early Stage Analysis and Troubleshooting • **Key Note:** EMI Test Receivers: Past, Present and Future: Andy Coombes, Rohde & Schwarz	Mechatronics & Model Based Systems Engineering
11:45	The Living Heart Model - A R2017x **3D**EXPERIENCE Industry Process					
12:15	Lunch					
13:15	Revealing the Power of an Open Platform for Simulation	Topology Optimization using Tosca for Abaqus continued	Hands-on with Linear Dynamics with Abaqus: Effective Solution of Structural Vibration Problems continued	Abaqus/CAE and Solvers continued	• Radiated emission of a DC-DC converter and the potential coupling to automotive AM/FM antenna • Co-Site EMI Simulation Technology and Workflow	Topology Optimisation with Functional Generative Design
14:15	A generative and integrated approach to designing functional additively manufactured parts					
14:45	Break					
15:15	The Journey to Simulation Driven Innovation	Topology Optimization using Tosca for Abaqus continued	Hands-on with Linear Dynamics with Abaqus: Effective Solution of Structural Vibration Problems continued	Abaqus/CAE and Solvers continued	• Conducted emission of a motor control and a DC-DC converter for automotive applications • Signal Integrity and Radiated Emission of Flexible Printed Circuits • Cable Simulation workflow	Conceptual Body Engineering with SFE
16:15	Rapid development of complex multiphysics and multiscale analyses					
17:00	Day 1 ends					
18:00	Reception & Banquet					

Final Agenda – 19th October

9:00	Welcoming Remarks	
9:15	Keynote 2	
9:45	Guest Speaker	
10:15	Break	
10:45	**User Paper 1 - Materials & Mechanics**	**User Paper 2 -General**
	Alex fergusson, FAC Technology - Virtual manufacturing and testing of syntactic foams	Bob Johnson, REAL - "A review of Abaqus element types for contact and plasticity"
	Caroline Graham, Leonardo UK - Using Abaqus FEA to show the critical effect of the composite FR4 out-of-plane properties	Stephen Gilmore & Ralph Collings - Crux Product Design - "Making all the right noises"
	Bob Andrews & Alex Brett, Rosen Group - Modelling of Real Crack Profiles Using Abaqus	Matt Hiett, Flow HD - The use of Xflow CFD for analysis and design of a shrouded rotor unit for use in a VTOL UAV
	Rizwan Choudry, University of Bath - Finite element prediction of tensile and buckling response of discretely stiffened composite panels	Alison McMillan, Wrexham Glyndwr University - Computational Tribology: Modelling the Lubrication of Rough Surface Elastic Bodies
11:45	SIMULIA Presentation - Abaqus Solvers (/Standard & /Explicit), Contact, Performance, Mechanics, and Materials	
12:45	Lunch	
13:45	**User Paper 3 - Materials & Mechanics 2**	**User Paper 4 - Optimisation & Processes**
	Brian Daniels, Amec FW - determination of creep strain and creep ductility material properties	Christian KNIPPRATH & Vijay RAMASAMY Airbus - Calibration of mesh-independent fasteners using Isight
	William Dearman, Frazer Nash - Modeling Rubber Subsea Air-Lock Seal	Jesus Blanco & Richard Tyrrel, Advanced analysis - Advanced connecting rod analysis using custom ABAQUS plugins
	Robert Luo, Trelleborg - IAVS Dynamic evaluation on rubber anti-vibration component using Abaqus	Tom Van Dijck, Ricardo UK - Vehicle Multi-Attribute Optimisation, demonstrated through a study on cross functional structural integration of a luxury Battery Electric Vehicle Battery Pack.
	Ioannis Thomaidisa, Alfredo Camaraa & Andreas Kapposa, University of London - Simulating the response of rocking rigid blocks using Abaqus/Standard	Ian Shilling & Jithu Jose Michael, Jaguar land Rover - Application of Structural Optimization in the development of main bearing caps
14:45	SIMULIA Presentation - The Expanding Scope of Simulation with Standalone Products	
15:30	Open Q&A - Closing Comments	
16:00	Close - Take away coffee & doughnuts	

User Presentations - The centrepiece of the RUM

The Regional User Meeting has become a time-honoured tradition in the UK. For 29 years industry and academia have met across the country to exchange success stories and challenges, and to debate and predict the future of simulation technologies.

The core of the conference is the shared user experiences and this e-Book details the abstracts that will be presented as technical papers during the conference. Some of the other features of the conference are also discussed in order to give as full a picture as possible of the benefits attainable from attending this event.

User 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 paper by providing unique user-specific viewpoints.

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 discussion of ideas.

User Paper Session One: Materials & Mechanics

Virtual manufacturing and testing of syntactic foams - Dr Alexander Fergusson - FAC Technology
A fully automated numerical model has been created to preprocess, run, and post-process a representative volume element (RVE) of a syntactic foam at volume fractions (Vf) approaching a jammed state (60% Vf). Abaqus was used to virtually evaluate the mechanical properties of syntactic foams of different Vfs in tension. A Monte Carlo simulation was used to account for the randomness of the packing arrangements of the microspheres within the RVE. The lower volume fractions were created via a Random Sequential Adsorption procedure, while a packing algorithm based on a molecular dynamics approach was used to achieve the high volume fraction microstructures. Representative elastic material parameters were selected for both the polymer matrix and the microspheres. Failure of the composite was modelled using a brittle fracture approach. For the epoxy matrix and high strength hollow microsphere syntactic foam studied, it was found that the maximum specific strength occurred around 50% microsphere volume fraction, and there was a tendency for the strength variability to increase with increasing microsphere content.

The numerical approach presented in this paper represents a powerful tool to greatly speed up the development of advanced lightweight structures by reducing the need for costly experimental tests to determine the optimal material composition for the specific stiffness and strength of the desired application. Numerical predictions are compared with experimental results with a particular focus on predicted vs observed experimental scatter.

Using Abaqus FEA to show the critical effect of the composite FR4 out-of-plane properties - Dr Caroline Graham - Leonardo MW Airborne and Space Systems
Fibre reinforced composites are commonly modelled as isotropic material. By their nature fibre reinforced composites are strong and stiff in-plane but can have low Young's modulus (E) and low shear modulus (G) in the through-thickness direction.
In microelectronic Ball Grid Array (BGA) shear tests solder joints are loaded through FR4 a glass reinforced composite. Load is not applied so as to put the fibres in direct tension but either in through-thickness tension or shear.
The work showed that the in-plane FR4 properties have little effect on the overall response (both in tension and shear) but that accurate through-thickness properties are required for satisfactory modelling. There is much disagreement in publications over FR4 composite through-thickness properties (this is assumed to be because it is notoriously difficult to measure, and is not normally of interest, particularly for thin material which is usually assumed to be in plane stress or bending). Abaqus FE modelling was used to derive out-of-plane values of E and G independently using published data.
These BGA shear tests are used to measure solder strain, assuming 100% of measured BGA sample displacement is solder deformation. Using substantiated through-thickness properties showed that the deformation within the BGA is more complex than has generally been assumed in the literature and in reality shear deformation of the solder ball could account for less than 5% of measured displacement. These laboratory tests have intentionally been isolated from harsh environmental concerns.

Modelling of Real Crack Profiles Using Abaqus - Robert Andrews and Alex Brett - Integrity Services Group, ROSEN UK
The usual practice when assessing the integrity of a cracked structure is to idealise the crack as an ellipse if buried or a semi-ellipse when surface breaking. The length of this idealized crack is set as the overall

length reported by the inspection and the through wall height is the maximum height reported anywhere along the length of the crack. This approach is simple and expected to be conservative. It is the approach of codified assessment methods such as BS 7910 and API 579.

In-line inspection technology for steel transmission pipelines has developed to the point where vendors can now report the actual profile of an axial crack. Whilst simple screening assessments will continue to use the maximum height and length, for critical cases modelling of the actual crack profile is now possible. This may remove excessive conservatism from the assessment, so avoiding unnecessary shutdowns or repairs.

This paper presents results from a study using both the XFEM and conventional focussed mesh methods to model the actual profiles of cracks found by in-line inspection. It is shown that for an elastic material XFEM is capable of giving J-integral results in good agreement with those from the focussed mesh approach, whilst reducing the complexity of model generation. For some crack profiles the maximum J-integral found anywhere along the crack was approximately half that derived from "handbook" stress intensity factor solutions, indicating a potentially large conservatism in the codified methods where failure is dominated by fracture. However, it was noted that for some crack profiles locally high values of J were obtained so that it was not possible to draw general conclusions. The XFEM approach was also used for elastic-plastic analyses but it was found that the convergence and agreement with the focussed mesh results was poor, so that it would not be possible to use XFEM for an Option 3 analysis in BS 7910, or to assess situations where failure is dominated by collapse.

Finite element prediction of tensile and buckling response of discretely stiffened composite panels - Rizwan Choudry – University of Bath

Stiffness tailoring of composite laminates by varying ply angles within the lamina-plane, can result in significant buckling performance enhancement without any weight penalty and is of particular interest to the aerospace-structures community. Practical consideration of high manufacturing rates, achievable through methods such as Advanced Tape Laying (ATL), require that the ply angles are varied discretely within a layer rather than continuously throughout. Such, discrete variations are termed, discrete stiffness tailoring (DST) and while DST can be optimized to maximize buckling performance, it may lead to reduction of transverse tensile strength. In this work ABAQUS standard was used to; (a) predict the performance enhancement in buckling due to DST, (b) study the influence of DST on transverse tensile strength. Two schemes of ply angle distribution were considered for tailoring stiffness across the width of the panel; (i) Half Seam; where half the layers in a laminate were subject to tailoring and (ii) Full Seam; where all layers were tailored. Non-linear compressive buckling simulations were carried out to determine the buckling loads (and strains) and to study the post buckling path. The transverse tensile strength was investigated using a detailed ply level model with cohesive zones for tracking the in-plane butt-joint failure as well as delamination. Comparison of simulations with experiments demonstrated that the model was reasonably accurate and the methodology can be extended for optimizing the DST panels.

User Paper Session Two: General

A review of Abaqus element types for contact and plasticity – Bob Johnson – REAL

Recent production work has revolved around the strength and fatigue validation of an offshore mooring system. Instead of one very complicated model of the complete system, Realistic Engineering Analysis Limited (REAL) has opted for a number of sub-assembly models. In all such models there has been conflicting requirements such as:-

i) Tight deadlines require that all FEA work is carried out as quickly as possible

ii) FEA must be accurate & reliable as the success of the whole system is at stake

iii) Loading at the minimum breaking load of the system causes significant plasticity

These criteria therefore indicate two main approaches: firstly convenient and relatively straightforward meshing direct from CAD using 10-noded tetrahedral elements throughout; and secondly, more onerous and difficult brick meshing using the 8-noded brick. It is generally accepted that an assembly model using brick elements will converge more readily and provide more accurate stresses (and plastic strains) than the equivalent tetrahedral model. That said the tetrahedral element model may take anywhere between 10% and 50% of the time required to build the traditional brick model.

With these conflicts in mind, REAL has designed an element test (between the 10-node tetrahedral and the 8-node brick) based on the contact analysis of the Chain Link as shown below. Four quarter models will be generated as follows:-

(1) Brick mesh using traditional meshing techniques such as sweep/copy/scale etc.

(2) Brick mesh derived by slicing up the CAD data until each cell is meshable

(3) Brick mesh derived by slicing up the CAD data but allowing a *TIE interface

(4) Tetrahedral mesh closely allied to all "features" of the CAD data

The results of this four-way comparison will be presented to the Regional User Group Meeting in October.

Making all the right noises - Stephen Gilmore - Crux

Traditionally device acoustics have often been overlooked as part of the design process. However, device developers are increasingly recognising the importance of sound. In tandem, recent developments are providing new ways to predict and optimise acoustic emission in-silico.

Crux has been leading the way. In the last year we have used Abaqus extensively to understand and optimise the loudness of complex medical device interactions and most crucially how the simulation data stacks up against real-life results. The process undertaken to simulate acoustic emission takes into account both the structural, dynamic behaviour of the system and the propagation of pressure waves in air.

Analysis of the simulated sound emission gives a measure of loudness for the component of interest. Repeating the process for each part of the mechanical system allows the key contributors of a particular noise to be identified.

A ball bearing was dropped from controlled heights onto one of two Delrin® plates. To provide a novel acoustic response, these were laser cut to unique spoked geometries. 50mm below the plate a laboratory microphone was positioned to capture sound from the impact.

Close correlation in peak amplitude was seen between physical and simulation data. For both plate components and various drop heights, the wave front generated through Abaqus was found to be an incredibly good match. This is key – the loudest noise comes in the first 0.1ms before material damping rapidly attenuates the signal. As a separate test, the structural response of the system was validated using high speed camera footage.

The use of Xflow CFD for analysis and design of a shrouded rotor unit for use in a VTOL UAV - Matt Hieatt -FlowHD

The development of a shrouded rotor drive unit for an unmanned UAV was undertaken by FlowHD limited for VTOL Technologies as part of a NATEP funded project.
This paper will aim to present some of the unique capabilities of Xflow that enable this device to be successfully modelled using moving geometry in a transient CFD simulation.
Key to capturing the physics of the device was correct modelling strategy for the small tip gaps between the rotor and the shroud, capturing the correct flow around the shroud, and ensuring the trailing wake turbulence was modelled to sufficient refinement to achieve successful correlation with both static and wind tunnel test results.
The presentation will explain these modelling strategies, what was successful and what wasn't, and will explain how the project allowed the end client to successfully improve the model and use results for overall flight performance prediction.

Computational Tribology: Modelling the Lubrication of Rough Surface Elastic Bodies - Alison McMillan - Wrexham Glyndwr University

The textbook approach to tribology separates the disciplines of lubrication and contact mechanics: Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) respectively. This compartmentalisation is reasonable for certain situations, for example, a CFD approach would be sufficient where the solid component can be thought of as rigid, but such an approach fails to recognise the opportunities which could be enabled if a multidisciplinary analysis approach were applied to the complete tribology problem.

Let's put this into context. Lubrication is an important topic in all machines with moving parts, particularly for components like gas turbine engines, where weight and dimensional constraints present challenges for bearing design. In the manufacturing context, fluid is used in machining for both lubrication and cooling. The cutting tool creates high shear stresses in the workpiece, giving rise to localised plastic flow, and leading to chip formation. It is clear that the FEA capability demands are high: plasticity modelling with a high degree of deformation, thermal gradients, localised stress gradients, and fracture mechanics.

There are unusual challenges for the CFD capability: while dimensional analysis would confirm that laminar flow is appropriate, the heat exchange role of the fluid suggests that variable fluid temperature,

and therefore viscosity and buoyancy, should be a feature of the CFD analysis. Chemical breakdown of the fluid is perhaps a challenge for future investigation!

In all, this investigation will explore the SIMULIA modelling capabilities, and present findings and further challenges.

User Paper Session Three: Materials & Mechanics 2

Dertmination of creep strain and creep ductility material properties for durability assessments - Brian Daniels, Amec FW

The objective of the following paper is to provide guidance on the determination of creep strain and creep ductility data required to perform a simulation of ductility, durability assessments for product design and quality control of high temperature components.
An example application is provided of the creep strain curve fitting and extrapolation processes using sparse results from Alloy 617 creep strain and rupture tests. It is demonstrated that the creep strain curve fitting of Alloy 617 test data gives a reasonable estimation of the applied stresses. Second order curve fitting of the estimated applied stress for constant-creep-strain against the Larson Miller parameter, represent a reasonable engineering approximation to the materials behaviour, aiding numerical interpretation and extrapolation of experimental data.

Modelling Rubber Subsea Air-Lock Seal - William Dearman - Frazer-Nash Consultancy Ltd

This project was carried out for a customer in order to better understand the behaviour of a rubber seal in a subsea airlock assembly. Specifically, how the seal performs at pressures above and beyond the design pressure to gain an insight into potential leak mechanisms.
The assembly consists of an axis-symmetric cylindrical chamber and three concentric rings which sit inside the chamber and compress a rubber sealing ring against the sealing surfaces. During use, the chamber must be able to be sealed and drained to atmospheric pressure or flooded to sea pressure before the chamber opens.

This analysis presented a number of challenges: Limited material properties for the rubber were available. Using Abaqus' built-in hyperelastic models, a pragmatic approach to bound the behaviour was taken. How to model fluid penetrating the sealing interface and establish at what point the seal leaks. Pressure Penetration functionality was used to model the sealing surface opening up and plot which pressure is being applied at all points along a surface at a particular moment in time. The large contact pressure between hyper elastic rubber seal and rigid parts leads to large deformation and fast changing contact surfaces. Additionally, the seal was overclosed in assembly drawings.

The customer gained valuable insight being able to visualise the seal as it deforms under pressure, something that they are not able to do during physical tests. This visualisation along with our understanding of the modelling techniques used can be used to inform future designs.

Dynamic evaluation on rubber anti-vibration component using Abaqus – Robert Luo - Trelleborg
There is no universal method to perform dynamic analysis on solid three-dimensional rubber anti-vibration components. Abaqus software is a leader in non-linear solid mechanics and provides a number of approaches for dynamic analyses. Viscoelastic approach is a usual method and has only achieved limited success. In this investigation, an equivalent damping ratio is introduced to account for rubber hysteresis using a resilience experiment. A new concept, NFR (Natural Frequency Region), is defined for dynamic evaluation based on natural characteristics of a rubber anti-vibration component itself. The verification procedure for this approach involves three parts: natural frequencies, harmonic spectra and impact responses. The dynamic responses from both simulation and experiment have been compared and have not only demonstrated that the proposed approach is reliable but that it could also be used to guide the design of rubber anti-vibration systems in industry. The key points for applying this approach are also presented through the verification procedure.

Simulating the response of rocking rigid blocks using Abaqus/Standard - Ioannis M. Thomaidisa, Alfredo Camaraa & Andreas Kapposa - University of London
Civil Engineering Department, Aristotle University of Thessaloniki, Thessaloniki, GR 54124
This work presents detailed Finite Element models developed in Abaqus/Standard to study the rocking response of free-standing rigid blocks, which is characterised by the alternation of impacts between the block and the foundation that suddenly change the contact point between both members. Series of nonlinear dynamic analyses with implicit time integration are carried out to study rocking initiation (uplift) and rocking attenuation under pulse-type motions and historic earthquake ground motions for rigid blocks of different slenderness. The latter is further studied by allowing the block to rock freely on the foundation. Solid homogenous sections are analysed with fully integrated 4-node plane stress quadrilateral elements (CPS4) both for the block and the foundation members. Rigid body constraints are used to refer the motion of each body to the respective centre of gravity. The contact is enforced using the small-sliding formulation and the resulting system of equations of dynamics is solved by means of the Abaqus/Standard HHT algorithm, examining the sensitivity of the results to the numerical damping (α), the mesh
default value of the numerical damping leads to different attenuation of rocking movement and unrealistic residual displacements in comparison with the analytical solution, but a smaller value of this parameter (α=-0.45) yields very accurate results. This suggests that the HHT algorithm in Abaqus/Standard is adequate to study the rocking of structures after selecting carefully its control parameters.

User Paper Session Four: Optimisation & Processes

Calibration of mesh-independent fasteners using Isight -Christian KNIPPRATH & Vijay RAMASAMY - Airbus Operations Ltd
In large scale Finite Element (FE) models mesh-independent fasteners offer a computationally efficient representation of rivets or bolts. Figure 1 shows an example of a wing model with fasteners highlighted for the outboard section.

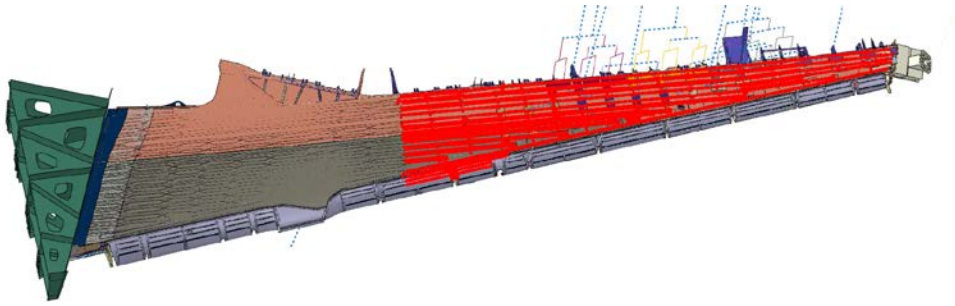


Figure 1: EW test model with highlighted fasteners in the outer wing section.

Here each individual fastener is represented by a construct which is capable of capturing the behaviour of a rivet or bolt in order to obtain an accurate representation of the structural FE model in the buckling and post-buckling domain. Therefore the behaviour cards can be assigned to an individual fastener or a set of fasteners with the same configuration (i.e. material, thickness and diameter etc.). However depending on the idealisation of the fastener the calibration process for the parameters governing the behaviour can be problematic and is often not a straightforward task.

This paper outlines the calibration approach for a mesh-independent fastener construct performed for metallic materials under quasi-static conditions. Here the software package Isight® is used to perform an inverse parameter estimation for the non-linear fastener behaviour based on experimental data sets. Figure 2 illustrates applied the Isight® workflow.

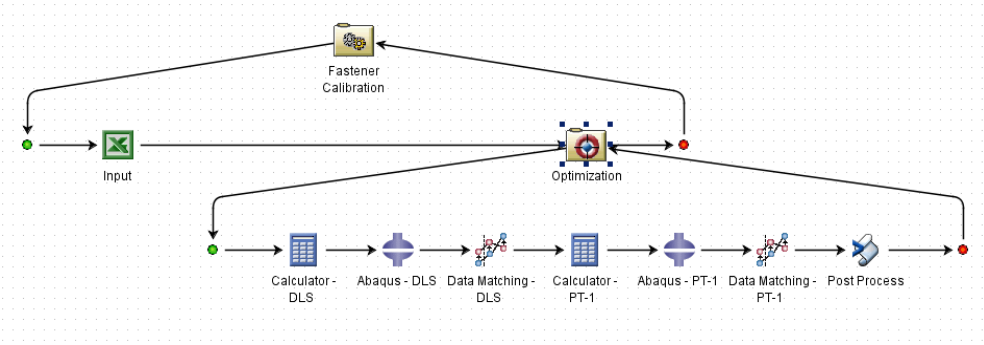


Figure 2: Example of an Isight® workflow layout for the fastener property calibration with a serial optimisation arrangement.

The identified parameters were then utilised to perform predictive Finite Element simulations for the wing structure. These results showed excellent agreement in the frame of the performed test campaign.

Advanced connecting rod analysis using custom ABAQUS plugins - Jesus Blanco & Richard Tyrrel - Advanced analysis

The paper describes the advanced use of ABAQUS and custom built CAE/Viewer plugins to analyse connecting rods. The methodology developed by AAL includes machining process effects, generation of oil film elements and a complex load calculation CAE plugin to represent all gas, inertial and dynamic loads on the connecting rod.

The procedure yields results for every 1-degree of crank angle per engine operating condition, in a single static analysis step. The results from the static analysis compare closely to the equivalent dynamic calculation, with significantly shorter run times.

Three custom built ABAQUS plugins were developed by AAL for this methodology:

- Plugin to write nodal coordinates and deformations from nodes displayed in Viewer. These are used to include the effects of the honing process via the *IMPERFECTION command.
- The 'Oil-Film' CAE plugin generates an oil film mesh with a parabolic distribution, used to represent realistic conditions at small and big end contacts.
- The 'Rod-Loader' CAE plugin performs complex loading calculations based on external (gas) loads, geometry and materials in the CAE model and all dynamic (inertial) loads. It writes Amplitude tables and a single static Step definition in CAE, which then represents the whole of the engine cycle.

The ABAQUS results can be post-processed with insightful animations in Viewer without showing rigid body motions of the rod - loads are calculated and applied to a vertical, nominally stationary rod. The results at every one-degree crank angle allow detailed fatigue analysis in Fe-Safe.

Vehicle Multi-Attribute Optimisation, demonstrated through a study on cross functional structural integration of a luxury Battery Electric Vehicle Battery Pack - Tom Van Dijck - Ricardo UK

Current advances in road vehicle development, such as the current push for Battery Electric Vehicle development, are accompanied by a range of new design engineering challenges in the form of new components, subsystems, and their associated vehicle integration requirements. There is a strong demand for these new, and previously existing, components and subsystems to be optimised. To get close to a true optimum design, the simultaneous consideration of multiple attributes is required. To achieve this through simulation, the joining of multiple tools into an automated process, capable of driving and optimising for a large number of variables is necessary.

As an example, Ricardo applied an integrated in-vehicle structural optimisation simulation process to a contemporary issue: the integration of a battery pack into a vehicle structure, with the aim of achieving competitive attributes at vehicle level. The attributes considered were mass, stiffness and NVH response at vehicle level, and resistance to crush loading at battery pack level.

The process incorporated several geometrically parameterised models of the battery pack, for assessment of attributes at multiple levels (stiffness, crush resistance), built in Abaqus/CAE; Response Surface Models generated using the parameterised models and Isight; Model translation tools; BiW stiffness analysis using MSC NASTRAN; Frequency Response analysis using Altair OptiStruct; and support tools such as MS Excel and Python. The work-flow automation required for optimisation and the optimisation itself were performed in SIMULIA Isight.

Application of Structural Optimization in the development of main bearing caps - Ian Shilling and Jithu Jose Michael -Jaguar Land Rover

With an increasing proportion of vehicles employing start stop and hybrid technologies, minimizing main bearing wear is an important requirement. Wear in main bearings is due to metal to metal contact between the crankshaft journal and main bearings shell, caused by the squeezing of the oil film at this interface. The main bearing cap can be a key parameter in minimizing this wear tendency, and ideally done in such a way to be compliant to the crankshaft deflection, thereby minimizing edge loading and consequently edge wear.

Global regulations aimed at minimising vehicle emissions have led to the downsizing and pressure charging of IC engines resulting in higher specific power. Which, coupled with the previously mentioned technologies, result in a more demanding duty cycle and an unprecedented challenge in the design of lighter, cheaper and durable systems. Main bearing caps are ideal candidates for optimization as a higher mass reduction for a given volume can be achieved as most high performance passenger car engines comprise steel main bearing caps bolted to / encased in an aluminum structure.

An optimization study was performed to develop a main bearing cap that is both compliant to crankshaft deflections and is optimized for mass, ensuring performance requirements whilst simultaneously satisfying the manufacturing, assembly, packaging, cost and commonality constraints. A topology optimization was undertaken in Tosca to identify potential mass reduction opportunity. Further shape optimization was carried out on the stiffness optimized cap to improve the fatigue safety factors.

Networking - Connect to your local community

We expect more than 100 attendees at the 2017 RUM. The attendees will come from across the UK and represent a wide variety of organizations. The agenda for the 2017 conference has been planned to maximise the opportunities for networking allowing delegates to learn from the experiences of others.

With delegates representing a wide range of industries and academic institutions you are sure to meet somebody working in a similar field to yourself. However, often the breakthroughs in simulation methods come from looking at work on quite different applications, so be sure to speak to a range of delegates.

In addition to the usual three breaks per day, the 2017 RUM will offer the following networking opportunities:

Wednesday 18th October

17:00 Join us in the 3DEXPERIENCE zone – Royal Suite

18:30 Drinks Reception – Garden Suite

19:00 Conference Banquet – Garden Suite

Partner Sponsors

Discover complementary technology

Partner Sponsors are a significant asset to our conference, offering innovative complementary solutions that will help you streamline your overall engineering process.

SIMULIA is pleased to recognise those partners participating in the 2017 UK SIMULIA Regional User Meeting.

Silver Sponsors

ssa

intrinsys
intelligent engineering

BETA
SIMULATION SOLUTIONS

SYNOPSYS®

EDEM™
Simulate material. Deliver results.

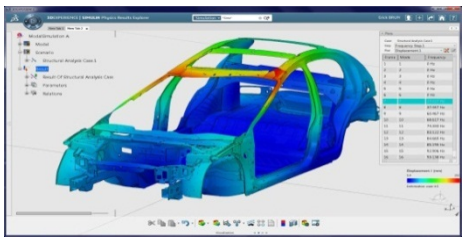
rescale

Conference Exhibition

In addition to representation from each of our partners the conference exhibition will also feature the following:

SIMULIA 3DEXPERIENCE

See the **3D**EXPERIENCE Platform up close and explore capabilities and potential with a SIMULIA expert.

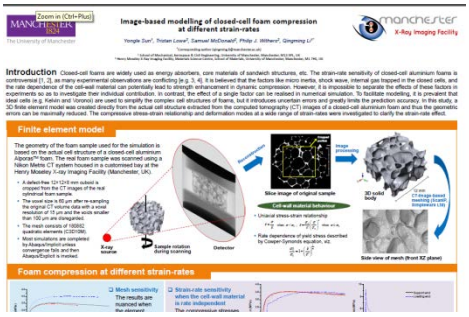


Visit the Power of the Portfolio

Get hands on with all that SIMULIA has to offer by, tackling a small workshop on Tosca, Isight and fe-safe!

Academic Poster Session

We have invited our academic users to showcase their work to industry in an academic poster section of our exhibition. Each poster will visually demonstrate the student's research, methodology and results.



The Living Heart Project

Z-Space experiences

Imagine a realistic 3D model of a beating human heart floating in the air right in front of you. Explore inside an automotive crash model or take a journey into the Living Heart or peel away the layers of a Geomechanics simulation using the unique Z-space tablets to investigate Abaqus output.

Immersive Visual Experience

Watch how we bring our products to life with CATIA **3D**EXPERIENCE Virtual Reality. Realistic Visulation improves communication and enables collaboration. Using this high-quality visualization, a common, shared mental image of the design intent can be created. Make sure you don't miss your chance to participate!