Your Essential Guide to the SIMULIA Regional User Meetings

Park Royal • Cheshire, UK

Conference Dates • November 4-5
Advanced Seminar • November 3

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

3DEXPERIENCE®

Contents

Your Personal Invitation

The Regional User Meeting (RUM) has been an annual event in the UK for the past 27 years, allowing SIMULIA users from all over the country to join together to exchange valuable industry knowledge and experiences. During these 27 years we have evolved from HKS to Abaqus to SIMULIA and have visited many different cities along the way.

The customer papers that are presented at the RUM characterise the local views of the users and provides attendees from a variety of engineering backgrounds with the chance to learn from their peers.

SIMULIA UK is responsible for providing technical support to our UK customers and this conference provides attendees with a fantastic opportunity to meet and spend some quality time with their local support team and other members of the UK organisation.

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

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/.

Registration

We understand that it is increasingly difficult to justify time out of the office to attend conferences, trade shows and seminars. It used to be the case that events such as these were the only way to find out about a product or subject area. However today, with so much information available via the web, this justification is less valid.

With this in mind we have compiled a variety of conference information into this e-Book to convey some of the wider benefits of attending the 2014 SIMULIA RUM, which go above and beyond that of pure product information.

We hope this information helps convince you that the benefits of attending the conference more than justify the travel and time out of the office and we look forward to greeting you at The Park Royal!

Conference Registration Fees:

Conference Delegate (Full rate – 13th September onwards)	£175
One Day Registration	£87
Academic Discount (must register with valid academic email)	£75
Advanced Seminar for conference attendees Advanced Seminar for non-conference attendees	£50 £150

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

Accommodation

To reserve your accommodation please contact the hotel quoting 'Dassault Systemes' in order to receive the discounted rate of £99 for bed and breakfast.

We advise you to make your reservations as soon as possible as the discounted rate is available for a limited time only.

Phone number: 01925 730 706

Key Note Speakers

Rod Giles, Principal Engineer at Triumph Design Ltd

Rod Giles has been involved in engineering from an early age, helping his father build 750 formula clubman racing cars. After graduating from the University of Bath in 1982, Rod joined Austin-Rover as a chassis engineer, and worked on the Metro, Montego, Rover 800 and 200 models. In 1990 he joined Lotus engineering and worked on the Lotus Elan and Esprit as well as projects for a number of clients, including Cadillac and Kia motors. In 1993 he joined Thyssen-Krupp Automotive, a tier-one suspension supplier where he set up and ran the analysis department. There suspension structures for Ford (Fiesta, Mondeo), Land-Rover (Freelander, Discovery), Opel/Vauxhall (Vectra) and Honda where developed. Rod then worked for an engineering consultancy from 1997 to 2008 where he worked on many different products in many different industries, including aerospace, defence, consumer goods, offshore, and industrial power generation. In 2008 he joined Polaris Industries, a manufacturer of recreational vehicles (snow mobiles, ATVs, off-road vehicles and motorcycles), where he was part of a small UK based team designing and developing an all-new 900 cc twin engine which powered the RZR XP900 – the most successful product in Polaris history. In 2011 he joined Triumph Motorcycles, responsible for powertrain CAE.

Rod is a NAFEMS advanced registered analyst, and has a BSc.(Hons) in Manufacturing from the University of Bath, a MSc. in Manufacturing Systems Engineering from the University of Warwick and is currently studying for a Mathematics degree with the Open University. His interests are motorsport, hiking and going to the gym. He recently completed his first triathlon.

Rod will present Advanced Analysis at Triumph Motorcycles at 11:15am on 4th November.

Simon Smith, Consultant at TWI Ltd

Simon Smith studied Mechanical Engineering and graduated in 1982 (Nottingham University). He then undertook post-graduate research using the Finite Element Method to develop models of creep crack growth. He received his PhD in 1986 (Nottingham University). He wrote Finite Element code and developed post-processing techniques for the determination of the path integral C* and for assessments using continuum damage mechanics.

Simon will present TWI, Structural Integrity, and Abaqus at 08:45am on 5th November

Silvia Schievano, Senior Lecturer, & Claudio Capelli, Research Fellow at UCL Institute of Cardiovascular Science & Great Ormond Street Hospital for Children

Silvia Schievano is a Senior Lecturer in Biomedical Engineering at UCL Institute of Cardiovascular Science & Great Ormond Street Hospital for Children. After graduating from Politecnico di Milano in Biomedical Engineering, she joined the Cardiovascular Unit of Great Ormond Street Hospital where she started setting up an engineering team embedded in the clinical environment. She was awarded a PhD in 2008 followed by a Royal Academy of Engineering/EPSRC Research Fellowship. Silvia's research has focused on structural and functional analysis of cardiovascular disease and device treatments, in-silico and in-vitro patient-specific modelling of congenital and acquired heart dysfunction, and the integration of biomedical engineering methods into decision making pathways for cardiovascular disease.

In January 2009, the engineering methodologies developed by Silvia and her team were successfully adopted to implant a new percutaneous pulmonary valve device 'first-in-man'. The development of patient specific models to test device implantation prior to the actual procedure has the potential not only to increase patient safety, but also to lead to more efficient and less frequent use of animal experiments. This work has contributed to the FDA's new regulatory strategy for integration of computer modelling into medical device approval processes.

Claudio Capelli is a Research Fellow in Biomedical Engineering. His research has been supported by the British Heart Foundation, the Rosetrees Trust and, more recently, the Heart Research UK. Claudio joined Silvia's group in 2008 for a PhD project focused on modelling cardiovascular devices, from engineering to clinical applications. Since then, his research studies have been focused on the translation of computational analyses into patient-specific clinical applications and personalised cardiovascular treatments

Silvia and Claudio will present FEM before FIM – Finite element modelling prior to first-in-man in heart valve technology at 13:45 on 4th November.

For full presentation abstracts visit: http://www.3ds.com/events/simulia-regional-user-meetings/uk-regional-user-meeting/speakers/

User Presentations - The centerpiece of the RUM

The Regional User Meeting has become a time-honoured tradition in the UK. For 26 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: General

Simulation of vacuum extraction during childbirth using finite element analysis

Rudy J. Lapeer, Vilius Audinis and Zelimkhan Gerikhanov, School of Computing Sciences, University of East Anglia

Vacuum extraction is an instrumental method used in obstetrics when childbirth labour fails to progress. The instrument used during vacuum extraction is the ventouse. It comprises of a suction cup attached to the fetal scalp through a vacuum, and a chord or chain to apply a traction force to expedite the delivery of the baby. It is claimed in the obstetric literature that incorrect placement of the cup, in particular across the anterior fontanelle, may cause serious injury to the fetal scalp. Here we put this theory to the test through a computerised simulation with finite element analysis. A fetal skull model comprising approx. 64K triangles (shell elements) and a model of the suction cup of a ventouse were used. Fetal bone and soft tissue properties were derived from previous work by the first author on the effect of labour forces on fetal head moulding. Calculation of the forces exerted by the suction cup was derived from the obstetric literature and their effect on the fetal scalp was calculated in custom-built software called BirthView. Once the material properties, forces and boundary conditions were established a non-geometric static analysis was run in ABAQUS 6.13. Results showed significantly higher deformations near the anterior fontanelle when the cup is placed in that area. Large deformations in this area may cause injury to the underlying anatomy, including blood vessels and may as such result into haemorrhage at different layers (depths) of the fetal scalp. These findings support qualitative observations from the obstetric literature.

Using topology optimization to generate personalised athletic footwear midsoles

Iain Hannah, Sheffield University

A number studies have shown that adding mass to a runner's shoes increases oxygen uptake (e.g. Frederick 1983) and thus the metabolic cost of running. However, further studies have also suggested that shod running may provide an energetic advantage over a barefoot condition for habitual rearfoot runners (e.g. Kerdok et al. 2002). As a result of these findings, lightweight performance footwear that still offers some degree of cushioning represents a large segment of the running shoe market (Nigg 2009; Asplund & Brown 2005).

A finite element footstrike model was developed and used with the Abaqus Topology Optimisation Module (ATOM) to output lightweight athletic footwear midsoles optimised for an individual athlete's personal loading profile. Biomechanical running trials were

performed by three healthy, male subjects (two rearfoot strikers, one forefoot striker) such that triaxial ground reaction forces and plantar pressure distributions could be measured. These loads were then applied to a generic midsole geometry via 20 rigid plates representing the plantar surface of the foot. An optimised midsole geometry for each subject's loading profile was subsequently generated with the objective function of each analysis being to minimise strain energy whilst simultaneously satisfy a constraint defined to ensure the midsole volume remained below a certain threshold.

As expected, each of the midsole geometry's output retained material at areas of higher loading whilst material savings were made in areas experiencing lower levels of loading. This resulted in a distinct solution for each subject's individual load profile. Geometric restrictions were also applied in subsequent analyses.

Use of Abaqus for efficient and accurate representation of axi-symmetric structures

Bob Johnson Realistic Engineering Analysis Limited

In many cases only detailed three-dimensional (3D) finite element analysis will achieve the reality that the engineer seeks. However we shouldn't go straight to 3D when there are simpler two-dimensional (2D) techniques at our disposal. Abaqus supports the usual variety of 2D solutions consisting of plane stress (for thin structures), plane strain (for long structures) and axi-symmetric structures (for structures "turned" about a central axis of symmetry). Of all these approximations, the most accurate, and in the view of the author, the most useful is the axi-symmetric formulation. Axi-symmetric analysis allows the engineer to address an enormous range of components and assemblies such as pressure vessels, threaded connectors, injectors, wheels/hubs and such forth in a very efficient and realistic manner. The proposed paper will show examples of 2D axi-symmetric analysis and the main thrust of the paper will discuss how these 2D models can be efficiently and accurately made from drawings and CAD data. The author will show competing technologies from the use of old-fashioned "ruled meshing" through to modern-day "paved meshing". The author will also show a combination of the old and the new in an attempt to get the best of both worlds. The paper will demonstrate relative accuracies and highlight ease of construction for real engineering assembly/contact models.

User Paper Session: Automotive and Manufacturing

Crack Prediction and Subsequent Simulation Driven Redesign of an Exhaust Manifold

Mark Stephenson, MAHLE Powertrain

With development times reduced to a minimum, MAHLE Powertrain (MPT) often finds that in the early phases of a project, when getting an engine running on a test-bed to confirm performance targets are achievable is paramount, CAD models for complex castings often have to be released for manufacture at the same time as to analysis. This was the case on a recent project where MPT had been tasked with uprating the customer's base engine to achieve a significant increase in power. The engine was to be turbo-charged and a new exhaust manifold was required as soon as possible to enable performance testing to begin. Thus as soon as the CAD model was completed it was sent for prototype casting as well as for analysis.

In order to assess the durability of the manifold a transient thermo-mechanical analysis was performed. The transient simulation cycle mimicked the exhaust manifold crack test which, as the name suggests, is a test designed specifically to test manifold durability. This test involves running the engine at maximum power for 3 minutes and then motoring it for 3 minutes. This cycle is then repeated. The test lasts 200 hours so the manifold undergoes 2000 thermal cycles.

Analysis of the initial manifold design showed unacceptably high cyclic plastic strains inside the manifold outlet region where the 4 manifold runners join together. When tested the manifold developed cracks in exactly the same area as predicted by the simulation. It was then a race against time in order to come up with an acceptable new manifold design before the manufacture of the next phase started. Simulation allowed several design variants to be virtually tested in a short timeframe leading to a manifold design that subsequently passed all durability tests, including the manifold crack test, without any failures.

Simulation of a Multi-pass Groove Weld and Clad Plate Using Abaqus 2D Weld GUI and Comparison with Measurements

C Parmar, CM Gill, PR Hurrell, BME Pellereau, Rolls-Royce plc

The computer simulation of multiple layering of welds is necessary to determine the distortion and residual stresses from the welding process. The welding simulation requires thermal and structural solutions, which are usually carried out in two simulations. Once solved, the thermal transient model temperature results are read into the structural model to solve for component stresses.

This paper describes the application of the AWI (Abaqus Weld Interaction) plug-in for simulating the residual stresses in ferrite-based weld specimens. The specimens were manufactured for an ongoing research programme within Rolls-Royce plc. The two test piece specimens were a Tungsten Inert Gas (TIG) welded narrow-gap ferritic plate with 239 weld bead passes and a multi-layer Metal Inert Gas (MIG) clad plate with 33 weld beads.

The Abaqus AWI Graphical User Interface (GUI) simplifies the generation of the Finite Element (FE) models, especially in regards to creating the heat transfer properties and setting up the surfaces for the welding bead sequences.

The simulation results were compared with Deep Hole Drilling (DHD), Incremental Deep Hole Drilling (IDHD) and Neutron Diffraction (ND) measurements in actual test piece specimens. The measurements showed similar results to the simulation when comparing peak stresses, however, there were some differences; these are believed to be attributed to the 2D plane strain assumption and the fact that phase transformation was not included in the simulation.

Multidisciplinary NVH optimization of Dual Core Lightweight Modular Engine

David Sziroczak, Ben Cudjoe, Integral Powertrain Ltd

The Dual Core Lightweight Modular Engine concept was developed in a Technology Strategy Board funded project, by a major automotive OEM, Intrinsys, its sister company Integral Powertrain and another engine development/manufacturing partner. The aim of the concept was to provide a modular engine solution for premium automotive product, which would be lightweight, and at the same time satisfy the stringent stiffness, durability and NVH requirements of the OEM.

Intrinsys successfully conducted all of the analysis work during the project using only SIMULIA simulation tools, running simulations in all the required disciplines including: durability; abuse loading; powertrain matching; structure borne and radiated noise.

A major challenge in the project regarded the radiated noise in the critical frequency range between 2-3 kHz, which was over the specified limit. This noise was identified to be caused mainly by the response of the structural Sump.

After many attempts to improve the design intuitively, Intrinsys analysts turned to Abaqus and TOSCA to help optimize the sump structure, and thus mitigate the excessive radiated noise. TOSCA's topology and beading optimisation schemes were used to optimise the sump panel shapes and give guidance to internal ribbing. The result was to shift the natural frequencies of the sump away from the critical region, or importantly reduce their responsiveness, and thereby reduce the radiated sound power from the engine surfaces. A python script was used to calculate sound power from the AVNSQ output in Abaqus.

In addition to reducing the radiated sound power, Tosca was also used to ensure the final design still met the mass budget, improved Sump stiffness, and satisfied the required functional requirements; such as the Sump volume required to hold engine lubricant.

User Paper Session: Offshore

Large Scale Prototyping in the Oil & Gas Industry: The Use of FEA in the Structural Capacity Rating of a Deep Sea Pipeline Clamping System

Dr. David Winfield, Laurence Marks, John Stobbart, Nick Long

Freudenberg Oil & Gas Technologies (FOGT) in Port Talbot, UK, provides complex metal to metal sealing solutions for the oil & gas and energy industries. FOGT is supplying two of its largest Optima® subsea connectors for use just inside the Arctic Circle. These will be the deepest of their kind anywhere in the world.

Weighing some 10 tons, the Optima® is a high precision, multi-piece clamping system using a FOGT Duoseal® metallic seal, tensioned by multiple leadscrew(s) activated via integral drive buckets. The resulting leadscrew tension positions the clamp segments on the hubs; as the tension increases, the opposing hubs are pulled together overcoming external forces and moments. Pressure energisation and plastic deformation ensure a high integrity double seal between the inner pipeline and the deep water environment.

Multi-body elasto-plastic finite element analysis (FEA) is used to simulate the interaction and contact between all parts of the Optima®, with focus on the stress and plastic strain of individual components during make-up and operation.

Fluctuating in-service loadings such as temperature, pressure and bending moment are also analysed to qualify the clamp segments, together with capacity analysis for the clamps and Duoseal®, where contact analysis is used to verify Duoseal® compliance. The Optima® is also required to overcome a range of hub misalignments, resulting from installation tolerances, friction and pipeline flexibility.

The FEA simulation results of the Optima® will be used to support experimental test data obtained during factory trials, prequalifying these components to the most extreme subsea loading conditions.

FE Modelling of Offshore Wind Turbine Grouted Connections

James Lockley, Senior Engineer, Atkins, Energy

The typical design of offshore wind turbine generator (WTG) foundations consists of a monopile (MP) driven into the seabed, a transition piece (TP) to which the tower and turbine are bolted, and a grouted joint between the MP and TP. Most offshore windfarms built before 2010 were built to design rules that were subsequently found to overestimate the axial capacity of the grouted joints, with the result that many have seen vertical slippage of the joints.

As the result of the slippage, in many existing turbines temporary installation attachments (jacking points) on the TP have come into contact with the MP. These now loaded jacking points are an initiation site of potentially life limiting fatigue cracking in the foundations.

To understand the mechanism of the grouted joints and the slippage, FE models of the foundations, and specifically the grouted joints and jacking points have been analysed. These have then served as the basis for substantiating remedial designs that provide alternative load paths. As the grouted joints are effectively reliant on friction to provide axial capacity, robust modelling of the contact within the grouted joint is required.

This presentation will present the modelling carried out and discuss some of the challenges in the models, as well as presenting some of the down-stream use of the FE results (fracture and fatigue assessments). In addition, current work to model the inclusion of shear keys in the grouted joint will also be presented.

Extreme response of Dynamic Umbilicals in Random Sea

R. Yasseri, S. Yasseri and Dr B. Wang, Brunel University,

Safe-Sight Technology

Umbilical systems provide power and control (electricity, hydraulic power, chemical injection) to deepwater subsea oil and gas equipment. Electrical cables and hydraulic tubes are arranged in helical bundles and placed in successive layers depending on the field requirements. Installations into increasingly deeper water place greater demands upon the structural components in terms of strength and fatigue. 3D finite element analysis is an applicable technique by which to assess the strain and stress distribution and the interactions between components in such complex built-up structure. FE enables the true geometry to be respected and the dropping of the simplifications of conventional approaches. However finite element models developed this way are complex to build and can reach a large size, hence an appropriate model construction and the simulation must be adapted in order to remain practical. It is proposed to use a global analysis, using pipe elements, to determine the internal forces and local analyses (sub-modelling), using shell elements, for detailed stress evaluation of interacting components. This paper addresses the global analyses of an umbilical in an extreme wave condition employing Abaqus/Aqua.

User Paper Session: Materials

Mullins effect on anti-vibration products with residual strain

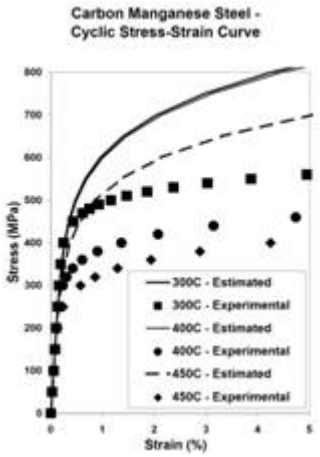
Robert Luo, Trelleborg IAVS

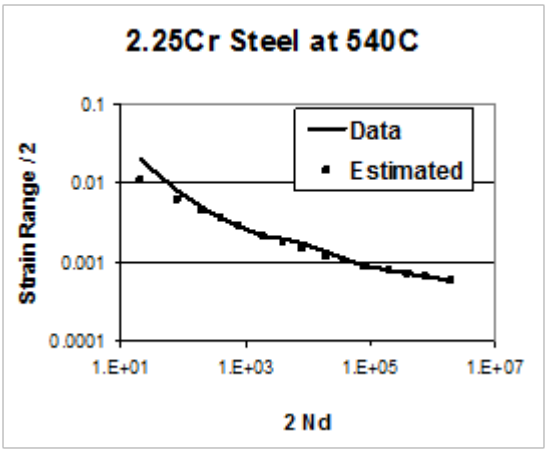
This paper presents an engineering approach to simulating the loading-unloading and reloading-unloading histories of rubber products with the Mullins effect in industry, including the stress softening and residual deformation. The work is a further study extended from Luo and Wu (2013) using rebound energy (resilience) approach. This rebound resilience is a key engineering constant in the functions, a physically meaningful concept. It should be noted that the proposed model does not need a negative stress state as a necessary pre-requisite for including the residual strain. In addition, they are continuous functions through the loading-unloading and the unloading-reloading points. The following parameters are defined in the functions: rebound resilience, unloading energy release rate, reloading energy rate, residual deformation variable and reloading decay function. The simulation was performed using well-known non-linear software Abaqus and validated by a well-defined experiment. It is shown that this method can be easily incorporated with FE software, e.g. Abaqus. The proposed approach provides the reliable prediction and can be used for engineering design and industrial applications. Nevertheless, the proposed approach should be further verified using more engineering cases.

Determination of Material Parameters

Brian Daniels, AMEC

The objective of the paper is summarise guidance on the determination of material parameters required to perform finite element analysis or durability assessments. In particular, the fitting of monotonic stress-strain test data and low cycle fatigue test data to the well-known Ramberg-Osgood (Figure 1) and Coffin-Manson (Figure 2) equations, respectively.





Overcoming mesh orientation bias in modelling of matrix cracking in composites

Supratik Mukhopadhyay, Stephen R. Hallett, University of Bristol

Continuum damage models are an attractive choice for modelling progressive failure in composites because they are straightforward to implement as user material subroutines and at the same time, computationally less expensive than other methods, e.g. discrete crack modelling techniques like the XFEM. However, they suffer from the inherent limitation of mesh orientation bias in which cracks typically propagate in a direction aligned with the mesh lines. This is more of a problem when modelling matrix cracking in multidirectional laminates, where matrix cracks in a particular ply should propagate along the fibre direction of that ply, being independent of the mesh. Often, in cases where matrix cracks are strongly coupled with primary failure mechanisms like inter-ply delamination, an incorrectly predicted matrix crack path fails to capture the damage interaction and hence is unable to predict the overall failure.

In this work, two different approaches have been taken to address the above mentioned problem by directly taking the fibre direction of each ply into account. The first one is a continuum damage model, implemented in Abaqus/Explicit as a user material, which uses a simple neighbour-search algorithm and the knowledge of fibre vector, to propagate cracks through the structure in a mesh-independent manner. The second one is a discrete cracking approach and is implemented as a user element routine, where, upon damage detection, an element is ‘split’ into multiple subdomains connected by cohesive segments which represent cracks. The orientation of these cracks is decided directly from the fibre vector, thus alleviating the pathological mesh orientation bias.

User Paper Session: Automation

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

Carlos Charnaux, Samuel Paul and Graeme Roberts, Subsea 7

The design phase for offshore rigid pipelines can be lengthy and demanding. A significant number of engineering hours can be burned before the various analyses attain target results. The assessment of the lateral buckling phenomenon is one aspect of the pipeline design that requires numerous finite element simulations. The purpose of such simulations is to determine if the pipeline has a genuine tendency for lateral buckling and, if so, whether control of the buckling behavior is required to ensure that the pipeline stresses and strains are within acceptable limits.

The process map for finite element based lateral buckling assessments follows a distinct procedure distributed into four generic steps: (1) gathering input information, (2) constructing finite element models, (3) running the simulations and (4) results post-processing. Typically, the process map is followed manually, and requires engineering analysts to develop quite complex FE models using typed data input with subsequent use of batched Python scripts to acquire model results, verify and validate the analysis runs and make tabular and graphical output.

Depending on project requirements and other circumstances such as availability of license tokens and disk space, the simulation process can be relatively inefficient and places a limit on the number of analysis runs that can be made and, therefore, the range of input that can be assessed. The latter point is important because the results of lateral buckling assessments depend heavily on input uncertainties (e.g. pipe-soil interaction data) and limiting the number of analysis runs may lead to conclusions that are not based on sound engineering judgment.

In modern offices, it is rare to find engineers who are conversant and comfortable with developing batch processes or writing computer code that would enable a large range of jobs to be completed and post-processed efficiently. However, Windows-based applications such as the user-friendly Isight program allow engineering processes to be automated in order to enhance the results fields and to better understand the influences of the uncertainties of some input parameters.

Subsea 7 has developed a Graphical User Interface (GUI) that works with Isight and which fully automates lateral buckling assessments, from the input gathering stage through model generation, post-processing, graphical output and model verification.

This paper demonstrates the current status of the automation process which is built around the “Splatter” GUI, Isight workflows and in-house Python scripts which are, admittedly, now rather old and in need of renovation. The process enables engineers with relatively little experience of Abaqus to create, analyze and post-process Abaqus models for lateral buckling, pipeline walking and on-bottom roughness assessments.

Automated Analysis in CATIA V5

Stephen Wallis, JLR

The traditional process for assessing the strength, stiffness and modal performance of a chassis component such as a suspension link, and iterating the design to meet targets in the optimum way, is time consuming. Some design engineers experimented with using integrated analysis in Catia V5 to assess and iterate the design more efficiently. Success was limited due to the complexity of setup and poor correlation with the results obtained by CAE specialists using the standardised process.

This paper outlines how a combination of Catia V5 analysis and scripting was used to solve these problems for a number of component types. The success of the tools released to date and the advantages of using them will be discussed. Guidelines for identifying processes which are suitable for automation will be presented

The Parametric Design and Automated Fatigue Analysis of a Double Pipe Heat Exchanger

William Kilpatrick, Prospect

Double-pipe heat exchangers are extensively used in the processing of raw natural gas extracted offshore due to their suitability for high pressure, high temperature operating conditions. The focus of this paper is the parametric design of a double-pipe heat exchanger using a multiphysics analysis approach and the subsequent fatigue analysis. The heat exchanger geometry was parametrically driven by classical calculations within Mathcad. A sequentially coupled analysis was performed by mapping the thermal operating cycle temperatures from a STAR-CCM+ Computational Fluid Dynamics (CFD) analysis to a linear elastic Finite Element (FE) analysis solved within Abaqus/Standard. The post-processing of the FE analysis model was automated through the application of a fatigue module developed by Prospect. The parametric design and multiphysics analysis conducted by Prospect was utilised to ensure the heat exchanger design satisfied the fatigue requirements specified by the American Society of Mechanical Engineers (ASME) for a 25 year continuous operating cycle.

Advanced Seminar: Maximize the value of your investment in SIMULIA technology and enhance knowledge

3rd November

Using SIMULIA Optimization Packages for Improved Product Design

Conference attendees: £50

Non-conference attendees: £150

This one day seminar will:

- Provide a detailed overview of the design exploration and optimization packages offered by SIMULIA
- Discuss in detail the more specialized non-parametric methods for structural optimization available in the SIMULIA Tosca product family.
- Discuss how these methods interact with industry standard FEA or CFD packages and can efficiently handle a very large number of design variables.
- Give an overview of Isight optimization methods which are parametric but can be applied to any mathematical calculation integrated in a process flow
- Explore how, together, Isight and Tosca can be very powerful for generating new concepts early in the design cycle or for fine tuning later when the design is mature. Used throughout the design process, both can lead to significant cost savings, product performance gains, innovative product design and a shortening of the design cycle in general.

For additional details visit: http://www.3ds.com/events/simulia-regional-user-meetings/uk-regional-user-meeting/seminar/

Networking - Connect to your local community

We expect more than 60 attendees at the 2014 RUM. The attendees will come from all over the UK and represent a variety of unique organizations. The agenda for the 2014 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 2014 RUM will offer the following networking opportunities:

TUESDAY 4TH NOVEMBER

18:00 DRINKS RECEPTION – GARDEN SUITE

19:00 CONFERENCE BANQUET – GARDEN SUITE

Round Table Session - Share your viewpoint

On Thursday the 5th November we will host a Round Table Session at 13:30 in the Royal Suite

The objective of each session is to participate in a live discussion with our technical experts and managers to discuss, hear and debate SIMULIA's strategy, capabilities and new functionalities. Conference delegates will have the opportunity to attend 3 out of the 6 round tables; each round table session will last for 20 minutes and will cover areas such as problem solving, debugging, mentoring, future direction and developments.

Round Table Sessions Planned for the 2014 RUM Include:*

- Strategy & **3D**EXPERIENCE
- Fatigue and Durability
- Contact & Convergence
- Process Automation
- Coupled Multi Physics Problems
- Materials & Material Behaviour

*Subject to change based on interest.

You will be asked to choose your desired round tables during your registration process.

We hope this session will allow delegates to raise specific points on the topics of most interest to them.

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. Learn more about select product offerings during the Complementary Solutions sessions taking place each day after lunch.

SIMULIA is pleased to recognise those partners participating in the 2014 UK SIMULIA Regional User Meeting.

FUJITSU

NVIDIA

simpleware

intrinsys
intelligent engineering

ssa

Applied

NAFEMS

Conference Exhibition

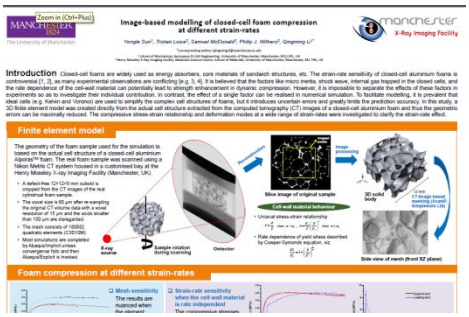
SIMULIA 3DEXPERIENCE

Come and try the **3D**EXPERIENCE Platform by tackling day to day modelling tasks in the new environment

Book a slot and give us your feedback.

Technical Support Desk

Do you have a technical question? Book a slot with the technical support desk to receive face to face advice and support on your technical problem.



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

Imagine a realistic 3D model of a beating human heart floating in the air right in front of you. Attendees of the RUM will not have to imagine, "The Living Heart Project Exhibit' will be featured in this year's conference exhibition area.

SIMULIA Technical Representatives Learn about the latest technology

The conference will be attended by representatives from SIMULIA's Research & Development, Technical Marketing and Worldwide Operations departments; they will provide a unique perspective on product capabilities and potential applications of our technology.

SIMULIA HQ Presenters

- Roger Keene, Vice President, Worldwide Operations

Roger is responsible for sales, technical support and services provided by the SIMULIA offices around the world, as well as supporting simulation sales by the DS PLM sales channels. He has worked for SIMULIA since 1991, initially as regional manager in the UK, then as head of marketing, General Manager Americas, and General Manager CATIA Simulation, before assuming his current position in January 2009. Roger has a Master's degree in engineering from the University of Cambridge and worked for Arup and MSC prior to joining SIMULIA.

- Juan A Hurtado, SIMULIA R&D, Technology Director of Materials and Explicit Analysis

Juan has been working for SIMULIA R&D (formerly HKS, Abaqus) for 14 years. During that time he has made many contributions to the advanced mechanics technology in Abaqus, especially in the area of constitutive modelling of materials. As leader of the Materials and Calibration Technology Area, Juan is directly involved in the long term planning and development of key simulation technologies aimed to address our customers' needs in the area of materials modelling. Currently Juan manages a team focused on development of Materials and Explicit Analysis capabilities. Prior to joining SIMULIA, Juan held several positions in academic research institutions, including Professor of Continuum Mechanics at the University of Granada (Spain) and Research Associate at the Division of Engineering at Brown University (USA). Juan holds a B.S. in Mechanical Engineering from the University of Seville (Spain) and a M.S. and Ph.D. in Materials Science and Engineering from North-western University (USA).

- Gergana Dimitrova, SIMULIA Optimization Portfolio Technical Specialist

Gergana Dimitrova is part of the SIMULIA Optimization Strategy Team, supporting the Global Optimization Initiative for integration of the optimization technology in SIMULIA Portfolio with focus on technical marketing. She joined FE-DESIGN in 2011 as CAE & Optimization Engineer after working in the field of Crash&Safety simulations. Gergana holds BSc. in Applied Mathematics from Sofia University in Bulgaria and a MSc. in Computational Science in Engineering from TU Braunschweig, Germany.

- John Draper, Dassault Systemes

John Draper is founder of Safe Technology Limited which was acquired by DS SIMULIA in 2013. John has over 45 years' experience in fatigue design and fatigue life assessment in the aerospace and ground-vehicle industries. He is a recognized authority in the industry, and a regular presenter at conferences and seminars worldwide - including conferences for ABAQUS and ANSYS users around the world, the ASTM Symposium on Residual Stress Effects on Fatigue and Fracture Testing, and more. John Draper is a former Chairman of the Engineering Integrity Society in the UK, and a member of the SAE (Society of Automotive Engineers). John Draper is also the author of Modern Metal Fatigue Analysis, published by EMAS Publishing

Preliminary Agenda

4th November	Title	Presenter
9:00	Welcoming Remarks	
9:15	SIMULIA Business Update	Roger Keene, Dassault Systemes
10:00	Abaqus 6.14 & fe-safe Update	Juan Hurtado & John Draper, Dassault Systemes
10:45	Break	
11:15	Advanced Analysis at Triumph Motorcycles	Rod Giles, Triumph Design Ltd
11:45	**User Paper Session 1 – General** Simulation of vacuum extraction during childbirth using finite element analysis Rudy Lapeer, University of East Anglia Using topology optimisation to generate personalised athletic footwear midsoles Iain Hannah, Sheffield University Use of Abaqus for efficient and accurate representation of axi-symmetric structures Bob Johnson, Realistic Engineering Analysis Limited	
12:45	Lunch	
13:45	FEM before FIM – Finite element modelling prior to first-in-man in heart valve technology	Silvia Schievano & Claudio Capelli, UCL Institute of Cardiovascular Science & Great Ormond Street Hospital for Children
14:15	Isight & Tosca Update	Gergana Dimitrova, Dassault Systemes
15:00	**User Paper Session 2 - Automotive & Manufacturing** Crack Prediction and Subsequent Simulation Driven Redesign of an Exhaust Manifold, Mark Stephenson, MAHLE Powertrain Simulation of a Multi-pass Groove Weld and Clad Plate Using Abaqus 2D Weld GUI and Comparison with Measurements, Chandrahas Parmar, Rolls-Royce Multidisciplinary NVH optimization of Dual Core Lightweight Modular Engine David Sziroczak, Ben Cudjoe, Integral Powertrain Ltd	
16:00	Coffee and Exhibition Break	
16:30	User Paper Session 3 – Offshore Large Scale Prototyping in the Oil & Gas Industry: The Use of FEA in the Structural Capacity Rating of a Deep Sea Pipeline Clamping System David Winfield, Freudenberg Oil & Gas Technologies FE Modelling of Offshore Wind Turbine Grouted Connections James Lockley, Atkins Extreme response of Dynamic Umbilicals in Random Sea	

	Sirous Yasseri, Wood Group Kenny	
17:30	Day One Ends	
18:00	Drinks Reception	
19:00	Conference Banquet	
5th November		
8:30	Welcoming Remarks	
8:45	TWI, Structural Integrity, and Abaqus	Simon Smith, TWI Ltd
9:15	Review of Advanced Materials Modelling Capabilities in Abaqus	Juan Hurtado , Dassault Systemes
10:00	Coffee and Exhibition Break	
10:30	User Paper Session 4 – Materials Mullins effect on anti-vibration products with residual strain Robert Luo, Trelleborg Determination of Material Parameters Brian Daniels, AMEC Overcoming mesh orientation bias in modelling of matrix cracking in composites Supratik Mukhopadhyay, Bristol University	
11:30	Partner Presentation	
11:45	Lunch	
12:45	SIMULIA 3DEXPERIENCE	Karl Kueres, Dassault Systemes
13:30	Round Table Session - Each round table will be led by a SIMULIA expert. Each session will last for approximately 25 minutes and will cover areas such as problem solving, debugging, mentoring, future direction and developments. Attendees will also be invited to participate in open discuss and debate key issues that are related to the subject.	Choose from three of the six sessions: Strategy & 3DX Fatigue and Durability Contact & Convergence Process Automation Coupled Multi Physics Problems Materials & Material Behaviour
15:00	Coffee and Exhibition Break	
15:30	User Paper Session 5 – Automation Increasing the efficiency of offshore rigid pipeline lateral buckling assessments using a dedicated GUI and Isight, Carlos Charnaux, Subsea7 Automated Analysis in CATIA V5, Stephen Wallis, JLR The Parametric Design and Automated Fatigue Analysis of a Double Pipe Heat Exchanger, William Kilpatrick, Prospect	
16:30	Closing Remarks	

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