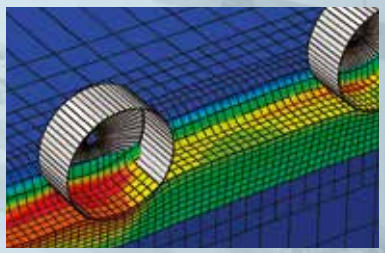
DASSAULT SYSTEMES

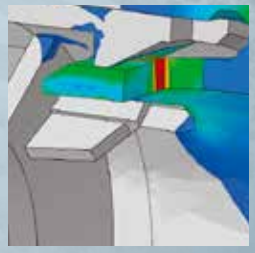
SIMULIA

SIMULIA COMMUNITY CONFERENCE

May 22—24, 2013
VIENNA, AUSTRIA
3DS.COM/SCC2013





3DEXPERIENCE

FOREWORD

This volume contains the abstracts of the papers submitted for presentation at the SIMULIA Community Conference held in Vienna, Austria, May 22–24, 2013.

We are grateful to the authors for their efforts in preparing the papers that make up these proceedings. The full paper can be found in the eBook.

CONTENTS

AEROSPACE & DEFENSE

CIVIL ENGINEERING

HIGH-TECH

INDUSTRIAL EQUIPMENT

LIFE SCIENCES

MANUFACTURING

MARINE & OFFSHORE

MATERIALS

NATURAL RESOURCES

TRANSPORTATION & MOBILITY

AEROSPACE & DEFENSE

A Methodology for Sensor Positioning Based on Eigenmode Data

P. J. Antunes, R. J. Guimarães, N. J. Ferreira, J. C. Viana, and G. R. Dias (Critical Materials S.A.)

The importance of sensor location is crucial for a correct assessment of the dynamic response of structures and, if early structural damage detection is a concern, for increasing the POD-Probability of Detection of Structural Health Monitoring (SHM) vibration-based methods. Also, the high cost of sensoring equipment (accelerometers, strain-gauges, fibre-optic sensors, etc…) implies the use of a minimum number of sensors that must be judiciously positioned in order to extract adequate and relevant information from the monitored structure. Critical Materials S.A. is developing algorithms for sensor positioning procedures, as an auxiliary tool for the deployment of vibration-based SHM techniques, based on Eigenmode and Eigenfrequency data retrieved from natural frequency extraction analysis procedures. Abaqus FEA-Finite Element Analysis platform, due to its capability in terms of dynamic analysis procedures and API-Application Programming Interface, contributes to the effectiveness and executability of the proposed sensor positioning methodology. The developed algorithms are capable of positioning accelerometers and/or strain sensors in function of the evaluation metrics used for the study and the specifications of the user. Several case-studies are presented and the proposed methodology is experimentally evaluated for the positioning of accelerometers in orthotropic CFRP-Carbon Fibre Reinforced Polymer composite based structures.

Keywords: Sensor Positioning, Modal Analysis, Eigenmodes, Abaqus Scripting

Multiphysics Simulations of Representative Hypersonic Vehicle Structures

Eric L. Blades, Parthiv Shah, Scott Miskovish, and Michael Nucci (ATA Engineering, Inc.)

This paper extends a previously developed computational framework for coupled aerothermoelastic simulations for hypersonic vehicle structures in extreme environments. An aeroheating verification and validation case is first conducted on a Mach 6.5+ spherical dome protuberance that was tested in a NASA wind tunnel. ATA's simulation includes aerothermoelasticity in the physics and captures the heating of the compliant dome structure during the first five seconds of exposure to hypersonic flow. This case is followed by a coupled quasi-static aerothermoelastic analysis of an acreage panel-level structural component for a sustained, reusable, air-breathing Mach 5–7 cruise horizontal-takeoff platform. The panels were consistent with the NHV outer mold line (OML), allowing the fully coupled simulation of two realistic structures in a realistic hypersonic environment. The paper presents results from a first quasi-static simulation of an orthogrid stiffened, curved panel that bows into the flow due to thermal stress and an assumed internal pressure. The effect of the stiffeners on the panel's response is profound and includes global and localized bowing relative to the undeformed panel and vehicle OML, resulting in the generation of multiple shock structures.

Keywords: Aerospace, Co-simulation Engine, Hypersonic Vehicle, Quasi-static Analysis, Unsteady Coupled Aerothermoelastic Analysis, Multiphysics Simulation

Modeling and Simulation of the Firing Cycle for an Automatic Small Arm

Lorenzo Castiglioni (Fabbrica D'Armi Pietro Beretta S.p.A.)

The presented work intends to illustrate how Beretta approaches the study of automatic small arms' firing cycles, from the launch of firing pin to locking mechanism being subjected to the gas pulse, to the recoiling of the moving parts operating the extraction and ejection of the spent cartridge case, and finally the feeding of a fresh cartridge into the firearm's chamber. Using finite element analysis for the study of such engineering problems benefits the product development cycle for its ability to provide key insights in the areas of reliability and durability. Through calculation of the dynamics of the moving parts, it is possible to qualify the functionality of the product, thus proving reliability. By using the load histories for each component during the firing cycle, an accurate estimation of fatigue life can be established,

proving durability of the design. Starting from the parasolid-based model assembly, this paper describes the use of Abaqus to create a complete mechanical model of an automatic small arm for a realistic simulation of the entire firing cycle.

Keywords: Bullet, Cartridge Case, Contact, Gas Pressure, Ejection, Extraction, Feeding, Locking, Recoil-operated, Rotating Barrel, Springs Preloading, Unlocking

Validation of VUEOS in Abaqus/Explicit and Application to Noble-Abel and Mie-Grüneisen Equations of State

A. Colletti and P. Carlucci (U.S. Army—ARDEC)
N. Dan and J. Hurtado (Dassault Systèmes Simulia Corp)

In the release of Abaqus 6.13-1, SIMULIA is adding a new user subroutine capability in Explicit termed VUEOS. This capability allows the user to define a hydrodynamic material model in which the material's volumetric response is determined by a user-defined equation of state (EOS). The introduction of this capability gives the user the ability define EOS's based on a constitutive relationship involving pressure, energy, and density, where the pressure may have parameter fits or empirical data. In order to validate this new user subroutine framework, the U.S. Army ARDEC developed two user EOS's for comparison with analytical and published data.

The validation is completed in two parts: first, a comparison to the analytical results of the constitutive equations; and second, application to other equations of state. The Noble-Abel EOS, a modified Van der Waals EOS, is implemented. This EOS is used for high-pressure gases such as those found in projectile launch generated from burning propellant. A Mie-Grüneisen equation of state is also defined and determines the pressure in shock compressed solid materials. This implementation uses a third order fit to the shock vs. particle velocity and a volume correction term. Validation results are presented, indicating a successful implementation of the new VUEOS subroutine interface.

Keywords: VUEOS, Ideal Gas, Nobel-Abel, Mie-Grüneisen, Equation of State, Us-Up, Constitutive Model, Shock Compression, User Subroutine, CEL, Abaqus/Explicit

A Plasticity and Damage Model for Fiber-reinforced Polymers

Ulrich Hofer (University of Innsbruck, Austria)

The Abaqus damage model for fiber reinforced plastics has been reimplemented as UMAT user material. As an extension, the plastic behavior of the matrix material is taken into account. Therefore, two modes are considered independently from each other: matrix shear and matrix compression. The original as well as the new material model have been verified by means of single element tests, mesh dependency tests, and the standard plate-with-a-hole problem. In addition, a shell-type structure, namely an industrial model of the Boeing-777 Winglet, has been analyzed.

Keywords: Composites, Constitutive Model, Progressive Damage, Plasticity, Polymer, User Material

System Integration as Key for Improving and Speeding up the Preliminary Design Phase of Aero Engines

Philipp Kupijai and Dieter Bestle (Brandenburg University of Technology Cottbus)
Daniel Kickenweitz (Rolls-Royce Deutschland Ltd & Co KG)

Usually the development of a new engine is initialized by the request of an aircraft manufacturer who formulates basic requirements for the propulsion system. Preliminary engine design is then the first design phase at the aero-engine manufacturer where the engineer's task is to develop a proper engine concept fulfilling all requirements and to respond to the aircraft company in a short time. At this early design stage only simple models can be used, but the decisions made are far-reaching and need to be precise. Therefore, engine component scaling

tools are applied to increase prediction accuracy. Accessing these methods, which have been developed by different component specialists, implies high manual effort due to manual data transfer and due to the fact that design is always an iterative process. This paper demonstrates an automated design process utilizing Java based Isight components which are a robust, secure, and fast alternative to common Excel sheets and may even be the superior solution. The process will be illustrated by design and optimization studies on two application examples showing the benefits in terms of saved runtime and better solutions than can be obtained by experience-driven, human search.

Keywords: Aero Engine, Automation, Multi-objective Design Optimization, Process Integration, Isight

Modeling of High-rate Ballistic Impact of Brittle Armors with Abaqus/Explicit

N. A. Nordendale and P. K. Basu (Vanderbilt University)
W. F. Heard (ERDC, U.S. Army Corps of Engineers)

Accurate modeling and simulation of brittle armor panels under projectile impact is a challenging problem involving costly experimental characterization of material properties and verification of ballistic impact response with actual test data. The nature of impact of brittle armors, specifically those made of cementitious materials, is a problem of much complexity, due to high-rate fragmentation caused by strong compressive shock waves followed by reflected tensile waves. Traditional Lagrangian finite element modeling (FEM) of these problems can introduce errors and issues of convergence as a direct result of the resulting large deformations of free surfaces. Introducing element erosion can resolve some of the issues related to excessive distortion of elements, but presents new problems involving conservation of mass. Regardless, traditional FEM cannot account well for secondary contact of spalled fragments from these panels. Smoothed particle hydrodynamics (SPH) is a more recent addition to Abaqus/CAE and resolves many of these issues. However, SPH is less accurate in general than traditional Lagrangian finite element analyses when the deformations are not too large. The main focus of the paper is to deal with these modeling issues related to high-rate impact of projectiles on high-strength cementitious composite panels, as well as to evaluate the situations in which a particular method would be more appropriate. A dynamic user-defined material model is used to accurately represent the complex behavior of this high-strength cementitious composite. The simulations of several example problems are validated with experimental results.

Keywords: Abaqus, Cementitious, Experiment, Impact, Material, SPH

Tire Debris Impact Modeling on a Composite Wing Structure

R. H. Rigby, M. Al-Khalil, and M. R. C. Fouinneteau (Airbus SAF)

This paper presents the implementation of an industrial FE methodology to predict delamination and debonding for a composite wing box subjected to tire impact. This numerical approach has been developed in Abaqus/Explicit v6.10 and is calibrated and validated using a building block strategy, where analysis has been correlated to test at each level. Ply-to-ply interface and bondline properties have been determined by an exhaustive test campaign to calibrate the cohesive element material cards. Intra-laminar damage model based on the Hashin model has been used together with cohesive elements to model the ply interface and bondlines. The methodology, successfully validated using the aforementioned building block approach, was then applied to tire impact on a composite wingbox structure. Of particular interest is the effect of including modeling of delamination on the prediction of stringer disbonding. For this large structural application a number of limitations were encountered, such as complex interaction of interfaces which created difficulties for run completion, for which a work-around was successfully applied.

Keywords: Aircraft, Composites, Damage, Delamination, Impact, Rubber, Tires

Multi-mode Failure Analysis of Scratched Laminated Carbon Fiber/Epoxy Composites

Seyedmohammad S. Shams and Rani F. El-Hajjar (University of Wisconsin)

Composite structures can be vulnerable to several types of damage. Abrasion and low-to-mid velocity impacts can result in deep scratches impacting the surface plies. In this study, the delamination, which is induced by scratches under tensile loading, is investigated using experimental and finite element techniques. A finite element approach based on progressive damage analysis implemented in VUMAT subroutine and a cohesive zone model using Abaqus are performed to simulate the behavior in the composite specimen. Scratches at different depths with different angles in carbon fiber-epoxy laminates are modeled to study the influence of the scratch size and the stacking sequence effects. Experimental results show the delamination propagates between layers. The results prove that the scratch tip location and size control the load carrying capacity as a result of the delamination response originating in undesirable bending and torsional deformations.

Keywords: Laminated Composites, Fracture Mechanics, Hashin Failure Criterion, Cohesive Behavior, Failure Theory

Prediction of Damage Evolution in Composites Using Abaqus

Wouter Wilson (Fokker Landing Gear B. V., The Netherlands)

Fokker Landing Gear has a history in development of composite technology for landing gear applications. In order to successfully design and qualify composite landing gear parts, it is essential to be able to reliably predict the mechanical and failure behavior of the material.

With traditional composite material models the mechanical behavior and failure moment of thin and undisturbed composite sections can be predicted relatively well. However, for thick composites and around stress concentration (e.g. open holes, pin loaded holes), these models substantially underpredict measured failure loads.

In recent years, Fokker Landing Gear has developed a proprietary material model to more accurately model composites. In this material model, the fibers and resin are included as separate materials with their own specific material and failure behavior, both with a limited number of material parameters and with clear physical meaning. The interaction between the fibers and resin is accounted for using a new analytical approach. Because the inclusion of fibers is done using a vector based approach the model is valid for any fiber structure.

In the current study, this material model has been extended to also include non-local damage mechanics (including damage evolution) and more realistic (physics-based) damage laws for fiber damage. Also a new damage initiation criterion for the onset of damage in cohesive surfaces (delaminations) based on both transverse shear stresses and out-of-plane stresses has been added. The implementation of this material model and cohesive damage laws is done in Abaqus/Standard. The extended material model can now also accurately predict the moment of failure and failure mode around stress concentrations.

Keywords: Composites, Constitutive Model, Damage, Delamination, Fabrics, Failure, Landing Gear, User-defined Material

Design and Analysis of a Foldable Wing Mechanism

Ali Yetgin and Bulent Acar (ROKETSAN Missile Industries)

Rocket and missile systems are often kept in launch tubes until they are fired. Dimensions of the tubes are determined by the munitions inside them. In some cases, due to space and weight limitations, launch tubes must match certain dimensions other than those imposed by the system. The inner diameter of the launch tube is mainly determined by control surface span. Thus folding the control surfaces decreases space occupied in the tube. In foldable wing mechanisms, wings are held in folded position in the tube. After munitions leave the tube, wings erect, and are locked secure in unfolded position. These mechanisms contain a spring like element that stores necessary energy to unfold the wing and a locking pin that secures

the wing in desired position. In this study, a foldable wing mechanism has been designed and analyzed under dynamic working conditions. Models were built using Abaqus/CAE and dynamic implicit solutions were obtained. In the analysis models, impact loads on the structure when the wing hits held piece were determined. The opening time of the wing, from folded position to fully erected position, was also examined. A series of iterations has been performed to optimize the spring parameters and fold positions. At the end of design study, optimized model was manufactured. To conclude the study, a test program was conducted. Test equipment that measures dynamic loads and opening time was manufactured. Following the tests, verification of the model was performed.

Keywords: Aerospace, Design Optimization, Dynamics

CIVIL ENGINEERING

Numerical Simulations of Quasi-Static and Rockfall Impact Tests of Ultra-High Strength Steel Wire-Ring Nets Using Abaqus/Explicit

J. P. Escallón (WSL Institute for Snow and Avalanche Research SLF)
C. Wendeler (Geobrugg AG Technical Department)

Flexible, light-weight wire ring-net barriers are used to stop falling rocks serving as protection for villages and roadways in mountainous regions. The steel wire-ring nets are difficult to model because of their complex geometry. The rings consist of a single steel wire strand wrapped together multiple times, and are connected to one another in a chainmail fashion to form the net. Their geometry and behavior represent a challenge to finite element modeling. This paper presents quasi-static and impact explicit FE simulation results of wire-ring net tests using an approach which relies on the general contact algorithm available in the FE code Abaqus. This approach allows a better description of the physics involved in impact problems related to rock falls. The model accounts for many complex physical processes: high-speed impact, contact with sliding friction, damage initiation and evolution, and strain-rate dependent material behavior. These physical aspects have been taken into account and the parameters involved have been calibrated with real test data. We demonstrate that the modeling approach of the ring net improves the accuracy and predictive value of the simulations.

Keywords: Ultra High Strength (UHS) Steel, Wire-ring Net, Strain Rate Sensitivity, Rockfall Impact, Mass Scale, Mesoscopic Ring, Macroscopic Ring, Damage, Ductility, Contact

Intermediate Crack Debonding Model of FRP-strengthened Concrete Beams Using XFEM

T. Mohammadi and B. Wan (Marquette University)
K. Harries (University of Pittsburgh)

The use of fiber-reinforced polymer (FRP) systems as externally bonded reinforcement has become very popular for the repair and strengthening of reinforced concrete (RC) beams as an alternative to the traditional techniques. One of the common failure modes of FRP strengthened RC beams is intermediate crack (IC) debonding of FRP initiated at the tip of flexural or flexural/shear cracks where the interface is subjected to mixed-mode loading. Because of the presence of the crack in the model, conventional methods are not efficient enough in this case; therefore, the fracture mechanic criterions can be applied to analyze the model. This study presents a numerical analysis method using the eXtended Finite Element Method (XFEM) implemented in Abaqus to model FRP intermediate crack debonding failure started at the tip of flexural crack in a concrete beam. XFEM models a crack as an enriched feature by adding degrees of freedom in elements with special displacement functions and does not require the mesh to match the geometry of the discontinuities. It can be used to simulate initiation and propagation of a discrete crack by getting help from fracture energy criterions along an arbitrary, solution-dependent path without the requirement of remeshing. The numerical results are validated against experimental data and there is a good agreement between them.

Keywords: Concrete Beam, Debonding Failure, Flexural Crack, Fiber-reinforced Polymer, FRP, eXtended Finite Element Method, XFEM

Transient Analysis of Air Pressure Loaded Lightweight Fabric and Beam Structures

Gerald Pichler (SinusPro GmbH)

SinusPro GmbH is tasked to help ensure the safety of removable lightweight structures which support banner advertisements and wind reduction sails at large sport events. The lightweight structures consist of fabrics, ropes, and masts and are built on-sight within a few days. They have to withstand wind forces higher than the event-terminating wind speed (18 m/s) and have to be "unloadable" under full wind load within minutes.

SinusPro uses Abaqus/CAE for model setup and postprocessing, and Abaqus/Explicit as the solver. To the customer, the rope and mast cross sectional forces were of special interest to provide proper anchoring and a comfortable safety margin against breaking of a rope.

Keywords: Buckling, CFD Coupling, Collapse, Connectors, Dynamics, Fabrics, Mechanisms

Experimental and Numerical Investigations of the Dowel Effect of Pile Grates in Quay Wall Structures

G. Qiu, K. Reimann, and J. Grabe (Hamburg University of Technology, Institute of Geotechnical Engineering and Construction Management)

The typical quay wall structures consist of combined sheet pile walls, pile grates and horizontal anchors in north Germany. Measurements have shown that the earth pressure on the sheet pile wall is shielded due to the dowel effect of the pile grates. The shielded earth pressure is transferred through the slab sections (reinforces concrete superstructure) to the anchor system. In the current design recommendation, a higher frictions angle is used to take the dowel effect into account. Firstly, 1g small scaled model tests have been conducted to obtain a better understanding of the failure mechanisms of the soil with regard to pile grates. Soil arches behind piles are observed in the model tests. In addition, the development of the shear bands is illustrated by using the Particle Image Velocimetry (PIV) technique. The numerical simulations using the Coupled Eulerian-Lagrangian (CEL) method have been carried out to estimate the resulted earth pressure on the sheet pile wall with proto type scale with regard to the shielding effect of the pile grates. The failure mechanisms shown by the numerical simulations match well with the observations from the model tests. The parametric studies have been performed to estimate the influences of the diameter of the piles, the height of the sheet pile wall, the block length and the pile-wall distance on the dowel effect.

Keywords: Quay Wall Structure, Dowel Effect, Sand, Arching, Active Earth Pressure, Coupled Eulerian-Lagrangian, Model Test

Concrete Constitutive Model, Calibration and Applications

Javier Rodríguez, Francisco Martínez, and Joaquín Martí (Principia)

A methodology has been developed for characterizing the mechanical behavior of concrete, based on the damaged plasticity model, enriched with a user subroutine (V)USDFLD in order to capture better the ductility of the material under moderate confining pressures. The model has been applied in the context of the international benchmark IRIS_2012, organized by the OECD/NEA/CSNI Nuclear Energy Agency, dealing with impacts of rigid and deformable missiles against reinforced concrete targets.

A slightly modified version of the concrete damaged plasticity model was used to represent the concrete. The simulation results matched very well the observations made during the actual tests. Particularly successful predictions involved the energy spent by the rigid missile in perforating the target, the crushed length of the deformable missile, the crushed and cracked areas of the concrete target, and the values of the strains recorded at a number of locations in the concrete slab.

Keywords: Concrete, Impact, Perforation

CONSUMER GOODS & RETAIL

Advancing Backgammon Strategy Using Isight

Axel Reichert (Dassault Systèmes Simulia Corp)

Backgammon is a game of luck and skill involving complicated strategic decisions. Since exact calculations are too complex to be performed over the board, simple and accurate heuristics are needed to guide these decisions. In this presentation, Isight has been employed to develop a decision criterion for important end-game situations that is both simpler and more accurate than the heuristics found in literature.

Keywords: Design Optimization, Isight, Python, Backgammon

CONSUMER PACKAGED GOODS & RETAIL

Integration of Moldflow and Abaqus in the Package Simulation Workflow

Eskil Andreasson and Leo Persson (Tetra Pak Packaging Solutions)
Eskil Andreasson (Blekinge Institute of Technology)
Henrik Jacobsson (FS Dynamics)
Johan Nordgren (Altair Engineering)

Tetra Pak has used numerical simulation tools for plastic injection molding (Moldflow) and structural analysis (Abaqus/Implicit and Abaqus/Explicit) for many years. Today, these two simulation tools are used independently of each other without any coupling. How these two disciplines can be combined to better predict the mechanical response of a polymer component is presented in this work. The manufacturing process, in this case injection molding, creates the mechanical properties of the produced polymer part. Process settings, material selection and molding tool geometry affect the polymer flow, material orientation and rate of crystallinity. A method to build a layered finite element model in Abaqus using results from Moldflow simulations regarding crystallinity growth and molecular orientation is proposed. Relatively simple material models were utilized and assigned for each individual material layer through the thickness in the polymer part. These constitutive models were derived phenomenologically from experimental test results and could adequately capture both the microscopic and the macroscopic behavior in a more realistic way. The numerical results showed a good agreement with the experimental results, both regarding visual appearance and force/displacement response.

Keywords: Abaqus/Explicit, Moldflow, Crystalline Orientation Factor, Constitutive Model, Damage Initiation, Damage Evolution, Fracture Mechanics, Material Modeling, Polymer, Injection Molding

Simulating the Pouch Forming Process Using a Detailed Fluid-structure Interaction

A. Mameli and A. Magnusson (Tetra Pak Packaging Solutions)
A. Aksenov, K. Kuznetsov, T. Luniewski, and I. Moskalev (Capvidia)

A deeper understanding of the interaction between machine, packaging material, and liquid product during the forming process of pouches is enabled by the use of numerical simulation. Tetra Pak pouch forming system forms the pouches in a continuous fashion starting from a tube of packaging material filled with the liquid to be packaged.

A bunch of jaws, moved with special laws, shape the tube of packaging material into pouches. The liquid into the tube is supplied through an inlet pipe, placed inside the tube. Additionally, a pouch forming systems has a counter-pressure flange and a floater that moves along the inlet pipe and floats on top of the liquid column. The floater is used to control the flow rate at the inlet pipe. Thus a complex problem with flexible walls, free surface flows and kinematic motion

of different bodies under hydrodynamic and gravity forces needs to be simulated. Simulating the pouch forming, including a detailed fluid-structure interaction is the goal of this paper. The simulation is performed coupling Abaqus for the structural side and FlowVision for the fluid side through a Direct Coupling interface. The definition of the problem as well as the results of the simulation is presented in the paper.

Keywords: Package, Forming, FSI, Code Coupling, FlowVision

ENERGY, PROCESS & UTILITIES

Numerical Modeling of Spudcan Footing Penetration in Stand

K. Abdel-Rahman, P. Gütz, and M. Achmus (Leibniz University of Hannover, Germany)
P. Peralta (Fugro GeoConsulting, Belgium)

Spudcans are a type of footing foundation to provide a secure operation of mobile jack-up rigs. They are connected to each of the independent legs of a jack-up rig. The current practice to assess the penetration depths of spudcans and the risk of geotechnical failure (punch-through) involves calculating the bearing capacity of the footing at discrete depths. The finite element method offers the possibility to model the complex spudcan penetration process while involving detailed soil parameters and spudcan geometry. In this paper, the use of Coupled Eulerian-Lagrangian (CEL) model elements, available in Abaqus, to simulate large soil deformations during the spudcan penetration process, is described. The novel approach allows the separation of the material movement from the finite element mesh and thus overcomes numerical problems associated with large distortion of elements. This method combines Lagrangian elements, which can represent solid structures (such as a spudcan), with Eulerian elements that idealize the soil. The application of CEL elements in a finite element model of the spudcan penetration process allows large-strain failure mechanisms to be observed, better understood, and additionally, the effects of the penetration process to be estimated.

Keywords: Offshore, Spudcan, Soil-structure-interaction, Abaqus, Coupled Eulerian-Lagrangian (CEL)

Enhancing Operational Flexibility of Fossil Power Generating Assets Based on Nonlinear Numerical Thermo-mechanical Damage and Fracture Analysis

A. Bagaviev, A. Karpunow, and M. Schneider (E.ON Anlagenservice GmbH)

A power generation toggle from the conventional sources to the becoming rapidly prioritizable renewable energy production poses a challenge for the fossil fired power plants. The strong dependency on the weather conditions makes such issues as non-predictable availability and variable loading to the ad hoc asset management tasks in terms of increasing operational flexibility. The existing fossil power plants were not designed to generate the energy according to the real time demand. The paper focuses on the results of this tendency and their impact on the backbone of all power plants—power plant equipment—suffering from the accelerated accumulation of the combined creep-fatigue damage because of the increasing tightened transient start-up and shutdown conditions, increased heating rates and number of operational cycling. Some practical examples of the remaining life time assessment of the rotating and non-rotating high temperature components based on the computational analysis shows the applicable ways to increase the component life time expectation and, therefore, leading to the noticeable flexibilization of the power plants.

Keywords: Power Plant, Turbines, Crack Propagation, Low-cycle Fatigue, Creep, Damage, Heat Transfer, Thermal Stress

Mechanical Behavior of a Pipe-in-pipe in a Span

Theofilos Giagmouris, Xinhai Qi, Andy Tang, and Jianxia Zhong (Genesis, North America)

In a free span, the inner pipe of a pipe-in-pipe system (PiP) bends substantially due to internal pressure and temperature and possibly contacts the outer pipe. The study is motivated by the

need to understand how the interaction between the two pipes affects the PiP behavior at a free span during thermal expansion and how a PiP can be modeled adequately. The pipes were simulated with thick wall pipe, elbow, and shell elements. The interaction between the inner and outer pipes was simulated with multi-point constraints (MPC), tube-to-tube contact elements (ITT), and contact surfaces. Finite element (FE) model accuracy and cost effectiveness were compared in a case study.

Keywords: Pipe-in-pipe, Free span, Pipeline, Bending

Comparative Analysis of Floor Response Spectra in Civil Structures of NPP Using Different Methods of Dynamic Analysis

V. A. Korotkov, D. V. Poprygin, and A. P. Ivanov (JSC Atomenergoproekt)

JSC "Atomenergoproekt" has great experience of NPP design. In this design, the problem of floor response spectra calculation for analysis of loading on the equipment has great importance. It is well known that this analysis is implemented in two steps. On the first step, the accelerogramm in points of equipment location (replying accelerogramm) is determined, and after that on the second step maximum value of absolute acceleration for nonconcervative oscillator under different values of frequency and in fixed value of damping is calculated as floor response spectrum. In this case, replying accelerogramm from the first step is used as initial action.

In the research, the most important problem is to determine kinematic parameters of the 'soil-structure' system through the use of the following methods from SIMULIA (Abaqus): direct integration method, modal method, and advanced modal method (SIM-architecture), where projected damping matrix is considered as completely filled.

This paper offers theoretical description of the above-mentioned methods, their advantages and disadvantages, and the comparative analysis of their results for the schematic model of the 'soil-structure' system. Here, one can find results of the calculation for a typical NPP building.

Keywords: Civil Structures of Nuclear Power Plants, Floor Response Spectra, Direct Integration Method, Modal Method, SIM-architecture, Soil Dashpot, Rayleigh Damping, Modal Composite Damping

Validation of Abaqus Virtual Simulation Model for an Expandable Liner Hanger

Ganesh Nanaware, Leo Gomez, and Andy Tom (Baker Hughes)

Expandable liner hangers used for wellbore construction within the oil and gas industry are complex mechanical systems. The consumable nature of the expandable products makes the accuracy and reliability of virtual simulation predictions important for reducing the time and cost to introduce a reliable and robust product to the competitive marketplace. This paper summarizes the seven-step methodology for validation of Abaqus virtual simulation model used for the performance predictions of an expandable liner hanger. The virtual simulation model of the liner hanger system comprises an adjustable swage to expand the hanger body, a slip ring to hang, and a packer to seal in a variable-diameter casing. The Abaqus/Explicit solver is used to evaluate and predict performance parameters such as required expansion force, hanging capacity, and seal integrity. The methodology presents the steps taken and challenges faced to validate the Abaqus simulation model from its initial build and refinement to the final model. Also included is how the validated simulation model helped us to study and optimize the effect of the various design parameters and investigate diverse design iterations with increased confidence. The seven-step validation methodology can be used as guideline by the simulation community to apply simulated predictions with confidence to achieve significant reductions in product development time and cost.

Keywords: Expandable, Liner Hanger, Oil and Gas, FEA, DOE, Reliability, Validation

Identification of Ductile Damage Parameters

Antonin Prantl (SKODA - JS a.s.)
Jan Ruzicka, Miroslav Španiel, and Milos Moravec (Czech Technical University in Prague)
Jan Dzugan and Pavel Konopík (Comtes FHT a.s.)

This paper describes the calibration process for uncoupled material model of ductile damage by Johnson-Cook and Rice-Tracey that is implemented in FE package Abaqus. As the work was supported by the grant, "Identification of ductile damage parameters for nuclear facilities," calibration was done for typical steel used in the nuclear power plant industry. The project includes both design and realization of experiments, and application of experimental outputs in the calibration process of material constants of mentioned ductile damage models. The calibration process of the material model uses fifteen types of experimental specimens corresponding with literature. The result of the calibration process was verified through the comparison of FE simulation of each specimen with experimental response. As the material model describing ductile damage within the wide range of stress states was searched in this project, the model was tested on several other specimens that exhibit higher stress concentration.

Keywords: Phenomenological Material Modelling, Ductile Damage, Fracture Locus

Multi-GPU Computing with Abaqus: Benchmarking and Scaling for Multiphysics Applications in Mechatronics

Aaron D. Price (ABB Corporate Research Center, Germany)

Mechatronic systems encountered in the power and automation industries exhibit very complex behavior for a variety of applications. These problems require solutions derived from diverse physical phenomena, and hence are considered to be multiphysics problems. One such problem includes computing the coupled electromechanical response of electroactive polymer actuators. Due to the complex nature of the problem, obtaining the transient response of electroactive polymer actuator-enabled mechanical systems is a computationally expensive task. In this study, we investigate how multi-GPU acceleration using Abaqus/Standard 6.12 may improve the overall simulation speed. Benchmark calculations are performed on a 16-core workstation with 3 GPUs. This approach is applied to two classes of problem size in order to elucidate scaling effects. As a result, some best practice guidelines will be given for efficient use of GPU acceleration in the given context.

Keywords: GPU Acceleration, Dielectric Elastomer Actuator, UMAT, User Subroutine, Multiphysics, High Performance Computing

Modeling and Analysis of Salt Creep Deformations in Drilling Applications

Jueren Xie and Gang Tao (C-FER Technologies Inc.)

Drilling through salt has created increasing challenges for the oil and gas industry. In many cases, deepwater exploration around the world requires drilling through salt layers above oil and/or gas reservoirs. Borehole closure and instability due to salt creep are two major concerns in designing completions for production wells drilled through salt layers. Similar issues also occur where gas storage caverns are built in salt formations. Pressure changes that occur in the salt cavern during operation can cause cavern closure and instability. As such, predicting salt creep deformations is essential for designing completions for wells drilled through salt and for storage caverns built in salt formations. This paper presents a literature review of material models describing the behavior of salt creep. Using an established material creep model, a finite element analysis (FEA) methodology using Abaqus is then presented for analyzing salt creep deformations. Application examples are included to demonstrate the use of this methodology.

Keywords: Casing, Cavern Storage, Collapse, Constitutive Models, Drilling and Completions, FEA, Geomechanics, Salt Creep, Stability

Study of Feedback Controlled Variable Cone Expansion Process

Allan Zhong and John Gano (Halliburton Company)

This paper will discuss a variable cone expansion of casings that is realized through a two-cone expansion system with a smaller sized, fixed cone at the front, and a variable cone at the back. The variable cone can be moved between an expanded position and a retracted position. During expansion, the variable cone is enlarged to its expanded position and advanced through the casing until a restriction (e.g. smaller ID wellbore outside the casing) is reached. At the restriction, the variable cone is automatically retracted to allow passage through the restriction without incurring a large force. Once passing the restriction, the variable cone will return to its expanded position. The movement of the variable cone between its two possible positions is realized through a feedback controlled mechanism.

In this study, Finite Element Analysis (FEA) is used to investigate the potential influence of frictions between different parts within the system during the movement of the variable cone. Ways to increase the critical friction coefficient via fixed cone modification are also evaluated.

Keywords: Wellbore, Feedback Control, Friction, Critical Friction, FEA

HIGH-TECH

Parameter Survey for Implicit Dynamic Time Integration under Contact Condition

Satoshi Ishikawa and Atsushi Koguchi (IDAJ Co., Ltd., Japan)

Dynamic problems involving contact are of great importance in industry related to mechanical and civil engineering, but also in environmental and medical applications. Regarding the implicit dynamic structural analysis, Abaqus/Standard has the HHT (Hilber-Hughes-Taylor) method which is a family of unconditionally stable one-step time methods for the direct time integration. This method is extended from the well-known Newmark beta method. The HHT method has three parameters; the parameters of beta and gamma are commonly of Newmark beta method, and the parameter of alpha is an additional one. While Abaqus/Standard classifies the implicit dynamic analysis applications into three types; "Transient Fidelity" and "Moderate Dissipation," in that are related to the HHT method. In this paper, we clarified the spectral radius of integration approximation operator for various time integration methods including Newmark beta family and the central difference method. Also, we investigated a variety of HHT's three parameter differences with the contact problem. Furthermore, we illustrated the practical application regarding impact analysis of a ratchet device.

Keywords: Implicit Dynamics, Impact, Direct Time Integration

Validation Study for Flow over a Sphere Using the CEL Method in Abaqus

Ahmad Firdaus Ismail (Mechanical & Human Factor Development, MIMOS Berhad, Malaysia)

The study of fluid dynamics has evolved so much from mathematical modeling to numerical calculations. Recent advancement in numerical calculations introduced by Abaqus for fluid analysis is the Coupled Eulerian-Lagrangian (CEL) method. The new method provides a quicker solution for a Fluid-Solid Interaction (FSI) analysis. However, when a new method is introduced, it is important for the users to know the accuracy, efficiency, and even the limitation of the method. The objective of this study is to validate the famous flow over a sphere using CEL method in comparison to the experimental data.

Keywords: CFD Coupling, Coupled Analysis, Experimental Verification, Fluid Mechanics

INDUSTRIAL EQUIPMENT

Automated Systematic Strength Calculation of High Duty Bolted Joints

Bettina Berlet, Oliver Trossmann, Ralf Waterkotte, and Martin Stief
(Schaeffler Technologies AG & Co. KG)

FE modeling of high-duty-bolted joints, such as those found in wind turbines, is in many aspects a very complex task. On the one hand, the bolt analysis must comply with standardized guidelines, such as VDI 2230. On the other hand, FE models are usually highly detailed and contain a large number of bolt connections. This makes the whole design process error-prone and cumbersome for the FE engineer, since modeling all these connections by hand usually takes a large amount of time and diligence.

Schaeffler Bolt Studio comprises a set of tools, which supports FE Engineers in the design process. With the Abaqus kernel interface, an integrated tool for the automated creation of bolt connections has been developed, which is based on recent research at the Schaeffler Group and the above mentioned guideline. The tool operates on standardized data and offers the possibility of creating bolt connections and all the necessary load cases in a minimum of time, making the model ready for strength analysis. Furthermore, several post-processing tools help the engineer to interpret the results and perform further calculations with the data gained from FE.

Keywords: High Duty Bolted Joints, Schaeffler Bolt Studio, Abaqus, Strength Analysis

Absolute and Relative Phases in Twin-tube Structures and Performance Criteria for Coriolis Meters

Jörg Gebhardt (ABB Corporate Research Center, Germany)

The main technological problem in Coriolis meter design is to achieve robust flow and density measurement signals. In this paper, it is demonstrated that Abaqus is a well-suited tool for implementation of related design performance criteria. It has played a major role in the development process of the new ABB Coriolis Master FCB330/350.

Keywords: Coriolis Meter Design, Abaqus, Design Performance Criteria, Zero Phases, Sensitivity Analysis, Finite Elements

Damage Assessment of Casting Materials in High-temperature Applications Influenced by Varying Plasticity Models

F. Laengler (BorgWarner Turbo Systems Engineering GmbH)
K. Naumenko (Institute of Mechanics, Otto-von-Guericke University of Magdeburg, Germany)

The turbine housing of an exhaust turbocharger is exposed to extensive cyclic thermo-mechanical loading. Consequently, the design of the turbine housing becomes a major challenge in ensuring the guaranteed lifetime in relation to the high-temperature behavior of the materials. In a first step, a phenomenological lifetime approach together with a rate-independent constitutive plasticity model of Chaboche-type was developed and validated for application on casting materials. The present study deals with the more detailed analysis of rate-dependent plasticity applied in the finite-element analysis with respect to the subsequent creep-fatigue damage calculation procedure. For capturing time-dependent effects both a creep law was additively superimposed to the rate-independent plasticity model and a constitutive viscoplastic model was developed for the casting material Ni-resist D5S. The creep model is by default implemented in Abaqus, whereas the viscoplastic model has to be linked via the programming interface. Thermo-mechanical fatigue tests on specimens under characteristic load conditions have been used to validate the influence of these different kinds of material models when describing the deformation behavior. Lifetime estimations are strongly dependent on the finite-element analysis output variables, and furthermore, the damage calculation procedure itself has to conform to the phenomena previous modeled.

Keywords: Plasticity, Creep, Low-cycle Fatigue, Damage, Thermo-mechanical Fatigue, Thermal Stress, Ni-resist D5S

Multidisciplinary Optimization of Turbine Components with the Aid of Surrogate Modeling Techniques

Iain Niven and Mark Willetts (ALSTOM Power)

Turbomachinery components design and optimization process requires a multidisciplinary approach. These components are designed and optimized for aerodynamic efficiency with robust mechanical integrity requirements. Further analyses are also required for thermal stresses, rotor dynamics, acoustics and other design requirements. These analyses require different tools—in-house and commercial—which must be integrated within a fully automated process. The calculation time for these calculations, particularly for CFD, could be a limiting factor for the number of iterations used. In order to ensure the widespread adoption of this automated process throughout the design and engineering organization, a generic process framework is being developed together with supporting software tools. This allows different types of analysis to be included in the framework process and the resulting analysis process to be easily applied to different design problems. This paper presents the approach to integrate multidisciplinary tools within an automated process and also describes the use of surrogate modeling techniques to reduce the time to execute these calculations. The development and use of the analysis framework process and supporting tools is also described.

Keywords: Coupled Analysis, Design Optimization, Multi-optimization, Seals, Turbines, Isight

Collaborative Robust Engine Design Optimization

Roland Parchem and Peter Flassig (Rolls-Royce Deutschland)
Holger Wenzel (Dassault Systèmes Simulia Corp)

This paper describes some of the major contents of the set up and technical coupled simulations within an extended enterprise as developed during the European FP7 research project CRESCENDO (2009-2012). Focus is given on the execution of simulation processes (workflows) for collaborative engineering design tasks in the aeronautical industry towards to ability to perform optimization and robust design studies. Different types of simulation workflows will be explained, such as event- or process-driven under the aspects of low, medium or high frequency access and execution. An industrial example of the implementation of a high-frequency process-driven workflow in the extended enterprise shall give an impression of the developed workflow execution capabilities in case of an aero engine design application.

Keywords: Aero Engine, Design Optimization, Robust Optimization, Collaborative Engineering, Isight, SEE

DOE Analysis for Internal Cooling Configuration of Gas Turbine Blade

Davide Pinna (Ansaldo Energia)
Francesco Cittadella (Exemplar)

Today, manufacturing companies are more and more often characterized by a growing product and processes complexity. Projects need the participation of a team that have to collaborate in a multidisciplinary and integrated way. The gas turbine blade is a typically multidisciplinary process where the variables of a discipline can strongly influence the results of other disciplines. For this reason, it is important to have a single simulation process that can correlate the different I/O and allow to investigate the entire Design Space, in order to verify even those solutions hardly predictable. The application of optimization techniques and simulation-driven design improve and accelerate the multidisciplinary simulation analysis, in order to obtain the best product in terms of technological innovation, performance and quality since the early design conception.

This paper is focused on a gas-turbine blade where both thermal and the mechanical disciplines are involved. The model of the complete process, implemented in SEE-Fiper environment and involving all the calculation tools, is described. The target of the project is the Design

Space Exploration (DOE Analysis) to identify a better internal cooling configuration in order to achieve a blade with simpler machining and that satisfies the criteria of minimum creep life and minimum fatigue life. The results obtained considering many optimization variables are reported.

Keywords: Collaborative Engineering, Design of Experiments (DOE), Design Optimization, Gas Turbine Blade, Isight, Multidisciplinary Process, Data and Process Management, Process Automation, Turbomachinery

Strength Analysis on Load Rejection Working Condition of Steam Turbine

Gongyi Wang, Kai Cheng, Jiandao Yang, Deqi Yu, Luyin Gu, and Zeying Peng
(Shanghai Turbine Works Co., Ltd.)

The load rejection is regarded as a vital process for a steam turbine performing commission operation or encountering some emergent accidents. Under the combination of the boiler and the turbine control valve, the steam temperature could be rapidly reduced and the turbine load could be cut back. In order to analyze the stress of the steam turbine during load rejection, the first stage rotating blades and grooves are taken as research subjects and the commercial software "Abaqus" is employed to perform the FEA process. The stress, which is strongly impacted by rapid cooling process in four periods, is calculated in this paper. The result indicates that the stress of rotating blade and rotor groove is sensitive to temperature change in short time. The stress increases quickly when the steam temperature drops rapidly. When the steam temperature drops to a certain level, the internal stress of the model is redistributed, and it always takes a rather long time for the stress to restore a uniform profile where the local peak stress occurs. In gradual cooling, the stress gets accumulated and tends to become higher. Although the redistribution of the stress takes a very long time, it is recommended to let the temperature drop to a low level before the next cooling process. During all four cooling steps, the stress of the components is well controlled within the elastic range of the material noted in this paper.

Keywords: Steam Turbine, Load Rejection, Heat Impaction, Strength Analysis, Low Cycle Fatigue

LIFE SCIENCES

A New Understanding of the Application of Tissue Strain by Negative Pressure Wound Interfaces through 3D Finite Element Analysis

Salim Mirza (Element Materials Technology)
Steve Bishop and Ian Tabron (ConvaTec Inc. Global Development Centre)

Within the wound care arena, the understanding of the effects of Negative Pressure Wound Therapy (NPWT) upon tissue and the processes of healing are still being developed. Optimization of the level of strain applied to the tissue to affect the healing process, without causing undue stress on the tissue and pain to the patient, remains a key clinical objective.

To provide additional understanding of the tissue strain and forces applied by different NPWT tissue interface materials, physical assessments of open cell reticulated foam (OCRF)+ and a polyester dressing with void spaces on contact surface (PDVS)++ were performed to confirm their three-dimensional structures and material behavior. This material structure information was then used to produce three-dimensional finite element analysis (FEA) models. These FEA models were then used to assess the predicted levels of strain applied to the tissue by these dressing materials under different levels of applied negative pressure.

The findings of this study have been compared with those of Wilkes et al1. Key differences in the findings from this new study are that the tissue strain distribution predicted for the OCRF dressing is highly non-uniform with localized high strains. Compared to this, the tissue strain distribution predicted for the PDVS is highly uniform with only small differences between the average and maximum values. Another important difference between the two systems

is that the strain distribution across the tissue surface for the OCRF dressing will be random because of the random nature of the foam struts, whereas the tissue surface strain distribution produced with the PDVS system is more uniform because of its stable and uniform structure under applied negative pressure.

These differences in the homogeneity of the tissue response and in the degree of stress applied to areas of tissue to which NPWT is applied are important factors for clinicians to consider in wound management. Within the recognized limitations of current viscoelastic material models of tissue and skin, the predicted tissue strains as a function of time for one of the interface materials assessed in this study compare favorably with previous clinical findings2.

Keywords: Wound Management, Negative Pressure Wound Therapy (NPWT), Tissue Strain, Material Behavior

Numerical Method for Laser Welding Simulation

S. W. Opolski (Atlantic Engineering, Inc.)

Numerical methods using Abaqus to simulate welding processes have been available for several years. These methods typically are based on arc welding procedures where one is depositing a bead or beads and the interest is in the thermal/stress history in the local heat affected zone. The significant thermal cycle found during welding does introduce residual stress in the structure. This residual stress must either be managed with a post weld heat treatment or the welding process and variables must be adjusted to minimize residual stress.

This method is different in that laser welding does not typically deposit any material and the thermal/stress zone of interest is not directly in the weld zone but some distance away where the combination of heat and material differences can cause material failure. In this paper, a method is described showing how to simulate a laser welding process using capabilities and tools found in Abaqus and Abaqus/CAE. The method allows sequential pulsing of the laser energy with all the typical laser welding variables such as pulse energy, pulse duration, travel speed, welding pattern, material and thermal properties.

While the laser pool is not modeled directly, the heat and stress effect throughout the rest of the structure is determined. The finite element thermal results are validated against direct temperature measurements. Examples are presented and compared with each other and the direct measurements.

Keywords: Laser, Welding, Thermal Stress, Heat Transfer, Implantable Medical Device

Reverse-engineering of Contact Lens Mechanical Properties from an In Situ Compression Test

Gary Richardson and Robert Stupplebeen (Bausch + Lomb)

Contact lenses correct the optics of the ocular system by placing a refractive element over the cornea. Modern lens materials are comprised of 30-90% water and exhibit viscoelasticity, hyperelasticity and thermal dependencies. Contact lens materials have traditionally proven to be extremely difficult to characterize for a number of reasons, primarily because the properties are dependent upon the processing conditions of the lenses themselves and the size and geometry of contact lenses makes them unsuitable for use in standard test apparatuses. In order to better characterize our contact lens materials, we have developed an In Situ lens compression testing instrument, wherein a lens is placed in solution and compressed between two flat surfaces while applied force is recorded. Contact lens geometries are reverse-engineered with CAD software (Dassault Systèmes CATIA), Finite Element Analysis software (Dassault Systèmes Abaqus) is used to model the force response of the lens and the optimization and data matching capabilities of a process integration and optimization software package (Dassault Systèmes Isight) are used to modify the material model parameters in order to match the test and model curves of force vs. displacement. This experimental system has allowed us to develop a much greater understanding of our own lens materials and those of our competitors, and it has provided new insights into the behavior of the materials vary with both temperature and the rate of loading.

Keywords: Biomechanics, Biomedical, Contact Lens, Abaqus, Abaqus/CAE, Isight, CATIA, Hyperelasticity, Nonlinear Elastic, Modulus, Material Properties, Reverse-engineered, Material Matching, Optimization

Applications and Impact of the New Beam-to-beam Contact Capability to Industries

Peter Zhang and Wei Gan (Medtronic CRDM)

Xiaoliang Qin, George Ang, and Subham Sett (Dassault Systèmes Simulia Corp)

In this paper, we discuss applications and impacts of the new beam-to-beam contact capability from Abaqus V6.13 to industries – in particular to the medical device and automotive industries. The applications that we interested in the medical device industry are bending simulations on the implantable leads, especially, bending simulations on coils and multi-strand cables. The applications in the automotive industry include simulations on bending of electrical cables, wires, and wire bundles. Being able to accurately and fast model the stress or strain to the coil filar and cable strand level for the problems above are important for design engineers to predict the field performances and reliabilities of the products in both medical device and automotive industries.

Without the beam-to-beam contact, the simulation problems mentioned above are either intractable or take very long computational time. With the new beam-to-beam contact simulation capability, the intractable problems could become tractable, and the computational time of the problems could be reduced by orders of magnitude. Because of the significant reduction of the computational time, we could potentially model problems that were not possible to do such as exploring the design space and create response surfaces or creating automated interactive modeling tools that can be accessed by design engineers.

Keywords: Beam-to-beam Contact, Medical Device, Cardiac Implantable Leads, Coil and Cable Bending Simulations

MANUFACTURING

Failure Analysis of Vise Jaw Holders

Essam A. Al-Bahkali and Adel T. Abbas (King Saud University, Department of Mechanical Engineering)

Sometime vise jaw holders of the hacksaw are broken due to accidental heavy loads or machine misuse. These broken parts are the stationary and movable vise jaw holders and the connecter power screw between them. With the aim of investigating the failure of these components, a three-dimensional finite element model for stress analysis has been performed. It starts by identifying the broken parts of the hacksaw machine. In addition, the material of the broken parts is identified, the CAD model is built, and the hacksaw mechanism is analyzed to get the accurate applied loads on the broken parts. After analyzing the model using Abaqus/CAE software, the results show that the location of the high stresses are identical with the original broken parts. Furthermore, the power screw was subjected to a high load which let the power screw be deformed. Also, the stationary vise jaw holder was broken by impact. This is because it is not touched by the power screw until the movable vise jaw holder is broken. A conclusion of the failure analysis is drawn and improving the design of the broken parts is suggested.

Keywords: Vise Jaw Holders, Hacksaw, Failure, Abaqus/CAE

Simulation of Beneficial Compressive Residual Stress Fields as a Design Tool in Manufacturing Internally Pressurized Parts

H. Brünnet and D. Bähre (Saarland University, Institute of Production Engineering)

Internally pressurized steel parts, such as components of fuel injection systems, are often treated with manufacturing processes which selectively induce beneficial compressive residual stresses. An available manufacturing technology to do so is hydraulic Autofrettage (AF). With

the help of a single over-pressurization, the compressive residual stress field is established right at the most failure critical area of the components, e.g. around the bore in a thick-walled cylinder or at the bore intersection in a pressure distributor block. This allows for the full exploitation of the material potential and the successful implementation of lightweight design concepts. However, the simulation of the material behavior during the process is a difficult task, as anisotropy effects like the Bauschinger effect lead to complex stress-strain relations. This paper presents several approaches to model the material behavior and the corresponding residual stress generation during AF for two representative geometries. The elastic-plastic material behavior is described with different available plasticity formulations in Abaqus/CAE. Additionally, different 2D, 3D, and axis-symmetric modeling approaches are compared and the performance of the chosen element types is discussed. Finally, two ways of modeling material removal after AF are presented (implicit is Abaqus/Standard and explicit is Abaqus/Explicit) and suggestions are made to consider the existing interactions along the manufacturing process chain.

Keywords: Residual Stress, Autofrettage, Material Removal, Abaqus/CAE

Thermo-mechanical Fatigue FE Analysis Applied to High Temperature Metal Forming Processes

A. Escolán, J. M. Bielsa, and M. A. Jiménez (Instituto Tecnológico de Aragón)
R. Allende (Productos Tubulares)

In high temperature metal forming processes, strong mechanical and thermal gradients in the formed material and in the tooling coexist. Such gradients have a negative influence in the tool life cycle. That influence occurs due to mechanical and thermal cyclic loads undergone during the production process. This factor leads to the appearance of cracks on the tool surface because of thermo-mechanical fatigue phenomenon. The presented work is focused on a specific manufacturing process of seamless steel tubes, which includes three basic steps: backward extrusion, perforation and Pilger rolling mill. This process transforms a blank or ingot into a final seamless steel tube with specific dimensions and characteristics depending on market requirements. The objective of this paper is to present thermo-mechanical fatigue FE analysis. These studies have been carried out in order to improve tool life and to know the effect of some key parameters of the main manufacturing process. For this end, several series of coupled thermo-mechanical FE simulations of the three analyzed conforming steps using Abaqus/Explicit solver were completed. Moreover, a post-processing subroutine has been incorporated that enables a prediction of the number of cycles until crack appearance (tool life). This subroutine includes a cumulative thermo-mechanical fatigue damage method of analysis where the damage is treated as a combination of three factors: multiaxial fatigue, oxidation and creep. In the multiaxial fatigue, it has been studied for each element that belongs to the surface, the critical level of crack initiation and propagation; in the oxidation and creep phenomena, the ratio between mechanical and thermal cycles dependence has been evaluated (i.e. in phase or out of phase).

Keywords: Ultra High Strength (UHS) Steel, Wire-ring Net, Strain Rate Sensitivity, Rockfall Impact, Mass Scale, Mesoscopic Ring, Macroscopic Ring, Damage, Ductility, Contact

Periodic Media Transport and Its Application at voestalpine Krems

Christian Hauser and Philipp Biegel (voestalpine Krems GmbH)

voestalpine Krems is a roll forming corporation constantly working on the improvement of product quality as well as the development of new production processes. Abaqus is used in both areas. Ever since the introduction of periodic media transport in Abaqus 6.11, we were immediately drawn to it as it is very close in nature to our main production technology. Thus we intended to make use of it in order to decrease the time consumption while on the other hand being able to utilize the full capabilities of Abaqus. Traditionally, when modeling any kind of rolling process, a reasonable long strip of sheet metal is pulled through the mill or the production line, requiring a large number of "passive" elements not currently involved in the

region of interest. The new feature "periodic media transport" enables the possibility to have only a small meshed region covering the interesting area. Within this paper, both approaches are shown and compared to each other. The first example problem shows a planetary rolling mill where we performed a feasibility study. The second example problem covers the whole roll forming process. The results are directly fed back to our engineering team to optimize the process. Critical further processing steps can be modeled afterwards by using pre-damaged parts.

Keywords: Cold Rolling, Design Optimization, Experimental Verification, Forming, Springback

A Numerical Prediction of Particle Contaminations in the Seal Design for a Bearing by One-way Coupling

Kangseok Kim and Daeyong Lee (Schaeffler Korea Corp.)
Yoochul Lee (CD-Adapco Korea)

In recent years, the tightening of fuel economy regulations and the trend towards environmentally-friendly products has increased the demand for compact, lightweight, and more efficient automotive transmissions. At the same time, the bearings in the transmission are always more often subject to harsh lubrication conditions. Contaminants in the lubricant have a strong life-reducing effect on rolling bearings. Past practices for protecting bearings from contamination include the use of lip seals make direct contact with raceway, but this solution lead to high frictional torque in the bearing system. Another practice includes the use of labyrinth isolators, but this solution also is not sufficient to prevent premature bearing failures due to contaminants. Therefore, a simple analysis method has been developed for predicting the penetration of contaminants from outside of bearing with non-contact lip seal. The method is fit for labyrinth isolator based on the geometry of the lip seal and the raceway that show the passing through the contaminants during bearing operation, which is driven largely by the flow of the lubricants. The system is analyzed with computational fluid dynamics where the lubricant is modeled with solid grids, while the lip seal and the raceway are represented with stationary and rotational wall. Through a better understanding of the mechanisms that lead to reduced bearing life owing to the effect of such factor, we can improve bearing design and operation.

Keywords: Contamination, Labyrinth, One Way Coupling, Co-simulation, Design Optimization

Structural Optimization of a Cleat for a Springfree® Trampoline

Guido M. Quesada (Matrix Applied Computing, Auckland, New Zealand)
Hamish McIntyre (Springfree NZ Ltd., Christchurch, New Zealand)

This paper reviews the structural optimization of a cleat from a Springfree® Trampoline. In this innovative trampoline, instead of using metal springs the mat is suspended on flexible rods attached to cleats made of fibre glass reinforced polypropylene. A physical testing device was built which applies the load from a rod at the centre of the cleat and supports it at the sides. Nonlinear simulations were used to replicate this experiment and evaluate the effect of changes in the cleat geometry before prototyping. The density and orientation of the glass fibre has an effect on the anisotropic material properties and orientations. Also residual stresses have an effect on structural performance. Therefore an injection simulation was performed using MoldFlow®. The results were imported into Abaqus/CAE to add the cleat to an assembly including two rigid surfaces which represent the testing device and the model was completed by editing the input file manually. This procedure was effective but too slow for an iterative optimization process. Therefore average isotropic material properties were estimated and modal eigenfrequency was chosen as the design response to maximize. A spring-mass analogy was used to relate stiffness to natural frequency and mass, particularly for a mode that resembles deformation under the physical test. Two other deformation modes were evaluated as well which provide an idea of the overall robustness of the cleat. Once a series of iterations lead to a satisfactory improvement in stiffness and mass, the nonlinear simulation procedure was repeated to verify the results.

Keywords: Design Optimization, Modal Analysis, Abaqus Topology Optimization Module, ATOM, Abaqus Interface for Moldflow

Advanced Finite Element Analysis to Tackle Challenging Problems in Pipeline Geotechnics

F. Van den Abeele, B. Spinewine, J-C Ballard, and R. Denis (Fugro GeoConsulting, Belgium)
M. Knight (Dassault Systèmes Simulia Corp)

Offshore pipeline design is a multidisciplinary field of engineering that covers route optimization, mechanical wall thickness design, fracture mechanics, flow assurance, stress analysis, geotechnics, etc. For decades, pipelines have been designed pursuing a stress-based approach, based on analytical methods and semi-empirical rules of thumb. However, the challenging conditions in oil and gas exploration and production (remote locations, seismic risks, ultra deep water developments, arctic conditions…) dictate the use of sophisticated numerical tools to assist the pipeline design engineer. In particular, pipe-soil interaction is a complicated phenomenon that governs the response of the offshore pipeline to operational load patterns. Interaction of the subsea pipeline with the seabed can give rise to buckling, pipeline walking or self-burial, and berm formation. Analytical approximations and simplified numerical models fail to capture those complex interactions, and hence, are no longer suitable to predict the offshore pipeline behavior. Solving such challenging problems in pipeline geotechnics requires more advanced finite element analysis tools, which are demonstrated in this paper.

Keywords: Geotechnics, Pipelines, Offshore Technology, Soil Mechanics, Pipe-soil Interaction

Fatigue Assessment of Welded-structures Using Continuum Element Meshes

W. Vonach, N. Friedl, and P. Löffler (CAE Simulation & Solutions GmbH)

Regarding FEM for welded structures, shell elements are mostly used to capture the structural behavior very efficiently. Depending on the choice of the element order, mesh size, etc., local nominal or even hot spot stresses can be calculated. These values can be assessed directly according to typical standards, i.e., Eurocode, DVS, FKM-Guideline, etc. In practical applications, there can be strong arguments against the shell element approach. The effort to generate an appropriate mid-surface shell model often is significant. Automated mid-surface generation is a common standard, but the correct joining of the parts (i.e., the 'welding' of the model) together with the necessity for a good mesh quality and size in these most critical welding areas still cannot be automated in a satisfactory manner. For complex structures costly manual work is always necessary.

Thus, it is inviting to mesh the CAD volumes directly and work with continuum elements. The corresponding meshes are available in much shorter time, e.g., using Abaqus/CAE. But when it comes to the fatigue stress assessment, the gain in efficiency is usually lost. Mostly, one ends up with stress results that are difficult to categorize regarding fatigue, since they are a mixture of nominal and partly resolved structural hot spot stresses or even singularities. The permissible values from fatigue standards are thus not directly applicable to these FEM stress results.

In our contribution, we will present a method, which allows an efficient fatigue assessment of welded structures, based on deriving relevant stress quantities from the results of continuum models. This method has been implemented in the commercial software LIMIT for typical standards.

Keywords: Fatigue, Postprocessing, Welding, Structural Hot Spot Stress, Nominal Stress

MARINE & OFFSHORE

High Performance Yacht Design with Abaqus

Ray Tsai (Simutech Solution Corporation)
Rey-Yie Fong (Tiny Machine and Mechanics Laboratory)
Yu-Chieh Lin and Chia-Chuan Ou (Ship and Ocean Industries R&D Center)

Unified analysis platform is the core value for SIMULIA Abaqus with realistic simulation. This paper demonstrated a successful integration of design process with various design requests where CEL dominated slamming wave impact transient analysis, CFD deals with ventilation and thermal analysis, and CMA was in charge with composite structure layup and manufacturing process management. This unified platform dramatically decreased yacht design cycle time and brought highly accurate simulations with complex physical problems.

Keywords: Abaqus, CEL, CFD, CMA, Yacht Design, Ventilation, Thermal, Composite Layup

MATERIALS

Strip-cooling Optimization by Means of Fully Coupled Thermo-mechanical-metallurgical 3D-model

Juan H. Bianchi and Luigi Langellotto (Centro Sviluppo Materiali S.p.A.)
José Díaz Alvarez (Arcelor Mittal España SA)

Advanced High Strength Steels (AHSS) are the fastest growing segment of strip products due to their high demand in the automotive industry. Because they represent valid alternatives to the use of expensive ferroalloys, steel producers are pushed to improve the processing routes and guarantee uniform properties conforming to specific targets.

During the cooling phase on the Run Out Table (ROT), thermal and phase transformation inhomogeneities throughout the strip could lead to some sectors of this not conforming to target mechanical properties. Moreover, the differential shrinkage between neighboring fibers might induce plastic deformation leading to permanent loss of flatness.

CSM metallurgical module microViewTM has been fully coupled in Abaqus/Standard Thermo-mechanical analysis and used in 3D modeling under generalized cooling conditions. First, it was applied to simulate rolling experiments done at CSM pilot mill using different cooling rates; the predicted volume fraction of constituent phases were compared against Metallographic measurements with good agreement. Secondly, the model was applied to an Arcelor Mittal España SA industrial line configuration and a sensitivity analysis was carried out to assess the effect of different cooling rates at the upper and lower face of the strip and finishing mill conditions (Temperature and Austenite grain size) on both breakdown of constituent phases and flatness. The model can be effectively used to optimize industrial parameters to guarantee dimensional/microstructure uniformity and absence of flatness. Moreover, an "early-quick" strategy that involves finishing water cooling and coiling the strip earlier were also examined and industrial practice modified accordingly to the predictions, with improvements in the strip quality

Keywords: AHSS, Multiphase Steels, High Strength Steels, Hot Rolled Strip, Hot Rolling Technology, ROT, Run Out Table, Cooling Bed, Coiling, Strip Flatness, Microstructural Coupling, Austenite Decomposition, Austenite Evolution, SRX, MDRX, DRX, Grain Size, Multiphase Properties

Analysis of Crack Propagation in Polyurethane Panels Subject to Cyclic-compressive Buckling

Steven M. Cron (Michelin Americas Research Company)
Tim D. Leister (École polytechnique)

When a panel of an elastomeric material, such as polyurethane, is subject to large cyclic in-plane displacements which result in cyclic compressive buckling, cracks will be observed to form at the center and edges of the panel. These cracks tend to propagate across the panel in the region of maximum deflection. If the magnitude of the applied cyclic deflection is held constant, the crack growth rate is observed to be nearly a constant over much of the width of the panel. Interestingly, a constant crack growth rate implies that the strain energy release rate is not a function of the crack length. This raises the following question: What are the geometric parameters that influence the crack growth rate for this type of structure? This paper will present a method for evaluating the strain energy release rate in compressively flexed elastomeric panels and present results showing the influence of basic geometric parameters on the strain energy release rate.

Keywords: Polyurethane, Panel, Buckling, Fatigue, Strain Energy Release Rate, J-Integral

Combining Detailed Experiments, PolyUMod®, Kornucopia®, and Abaqus to Create Accurate FE Scratch Simulations

Ted Diehl, Li Lin, Ye Zhu, John J. Podhiny, Richard T. Chou, and Jeffrey A. Chambers (DuPont)
Jorgen S. Bergstrom and Xiaohu Liu (Veryst Engineering, LLC)

Creating a viable simulation approach capable of representing key fundamental aspects of scratch of polymers is an extremely challenging task. The simulation must include complicated material response, such as nonlinear viscoelastic/viscoplastic behavior of the polymeric material at large strains. In addition, the model must include a sufficiently detailed representation of the scratching event using frictional contact between the indenter and the sample, and the model must take into account the indenter loading history. And perhaps the most challenging of all, the simulation must properly and efficiently represent the long recovery time after the scratching event during which the highly deformed polymer can partially recover. This paper discusses how Abaqus/Explicit models using the PolyUMod® advanced Parallel Network Material models were used in conjunction with detail experiments processed by Kornucopia® to create scratch simulations that correlated well to several physical scratch tests. In particular, various technical challenges that were solved during the development of a successful modeling strategy are described and discussed in detail.

Keywords: Constitutive Model, Creep, Damage, Experimental Verification, Plasticity, Polymer, Post-processing, Viscoelasticity, Viscoplasticity, Material Model Calibration, User-material Model, Nonlinear Analysis, Large Deformation, Remapping, Data Smoothing, Data Filtering

Recent Developments in Mechanical Characterization (Deformation and Failure) of Materials

Amos Gilat and Jeremy D. Seidt (Ohio State University, Department of Mechanical Engineering)

Several new testing methods that have been recently developed for mechanical characterization (deformation and failure) of materials are presented. The data from these tests can be used for the development and calibration of material models (constitutive relations) in Abaqus. The first method involves the use of Digital Image Correlation (DIC) in tests that are used for generating data needed for failure models. In these tests specimens with different geometries are loaded and DIC is used for measuring full-field strains and displacements. The second testing configuration is a shear test for sheet metals. The experiment is done by using a flat-notched specimen in a tensile apparatus. The shear strain is measured by using DIC within and on the boundary of the notch. The third development is a high-strain-rate tensile testing technique for Kevlar cloth and Kevlar yarn in a tensile Split Hopkinson Bar (SHB) apparatus. The Kevlar cloth/

yarn is attached to the bars by specially designed adaptors that keep the impedance constant. In addition to the traditional method of determining the specimen's stress and strain from the recorded waves in the bars the strain is also measured with DIC. The fourth development is an apparatus for testing at intermediate strain rates in compression. In this apparatus, the specimen can be deformed at strain rates ranging from 20 s-1 to 200 s-1. The apparatus is a combination of hydraulic actuator and a compression SHB. The stress in the specimen is determined from the stress wave in a very long (40 m) transmitter bar and the strain and strain rate are determined by using DIC. The results show very clean (no ringing) stress strain curves.

Keywords: Material Characterization, Plasticity, Failure, Digital Image Correlation

Multiphysics Simulation of Metal Solidification Processes with Abaqus

Seid Koric, Brian G. Thomas, and Lance C. Hibbeler (National Center for Supercomputing Applications-NCSA & Mechanical Science and Engineering Department, University of Illinois at Urbana-Champaign)

A coupled thermo-mechanical model of solidifying shell (Koric, 2006, 2011), (Hibbeler, 2009) in Abaqus/Standard is combined with turbulent fluid flow in the liquid pool and thermal distortion of the mold to create an accurate multiphysics model of steel continuous casting. The new model is applied to calculate temperature stress and deformation in a commercial beam blank caster with complex geometry. Results from the complete system compare favorably with plant measurements of shell thickness.

Keywords: Material Processing, Solidification, Steel Continuous Casting, Thermal Stress Turbulent Fluid Flow, Enhanced Latent Heat, Multiphysics, Finite Elements, Abaqus

Hot Rolling and Accelerated Cooling Simulations Using Abaqus—A Fertile Basis for Fast Online Algorithms in Heavy Steel Plate Production

E. Parteder (voestalpine Grobblech GmbH)
A. Binder (MathConsult GmbH)
G. Wollendorfer (GEWOTECH Technology GmbH)
T. Antretter (Institute of Mechanics, Montanuniversität Leoben)
K. Zeman (Institute of Computer Aided Methods in Mechanical Engineering, Johannes Kepler University of Linz)

Increasing customer requirements concerning product quality (e.g. dimensional tolerances, mechanical properties) in heavy steel plate production, require permanent enhancement of online control systems. Focusing on the main production steps "hot rolling" and "accelerated cooling," the prediction of the local thermo-mechanical behavior applying physically based modeling techniques is of vital importance.

An Abaqus model was developed for calculating the thermo-mechanical history during hot rolling and accelerated cooling processes. Constitutive laws describing the material behavior were implemented via user subroutines. For both production steps, material parameters of the rolled stock were identified from laboratory experiments. For the hot rolling process, the kinetics of softening (recrystallization and recovery) between consecutive rolling passes and work hardening within the roll gap are essential physical effects. During cooling the dynamics of phase transformation has to be accounted for. Simulation runs for the production cycle of one plate were automated using python scripting, including preprocessing, mapping of results between single pass models and process steps, output reporting and graphical visualization.

In a further step, customized algorithms based on the Abaqus model were developed. This strategy reduces the computation time for the thermo-mechanical history to a few milliseconds with an accuracy equivalent to the Abaqus model. The fast algorithms can be used in online routines for process control systems. In addition to the main process variables such as rolling force, torque, and power consumption, the accurate prediction of the evolution of local temperature and deformation provides the opportunity to couple these variables with modeling concepts predicting the mechanical properties of the final product.

Keywords: Heavy Steel Plate, Thermo-mechanical Rolling, Accelerated Cooling, Python Scripting, Abaqus

Improved Quality Prediction of Injection Molded-fiber Reinforced-components by Considering Fiber Orientations

Sascha Pazour, Wolfang Korte, and Markus Stojek (PART Engineering GmbH)

For fiber reinforced parts, the consideration of anisotropic material behavior is required to receive reliable results. In the scope of this fact, a procedure is described how to consider these effects in terms of process-structure interaction and how to achieve possible benefits such as weight reduction and shorter development cycles shown for examples of the industry. The developed procedures show how to consider the anisotropic mechanical behavior of injection molded short-fiber-reinforced plastics parts in FE analysis. It is shown how the existing information, which is provided by injection molding simulation software, can be processed and transferred into mechanical simulation models. The procedure is outlined with practical applications.

Keywords: Composites, Fiber-reinforced, Injection Molding, Process-structure-interaction, Anisotropic

Analysis of Cracking Characteristics for Various Indenter Shapes Using the Cohesive Zone Model

Hong Chul Hyun, Felix Rickhey, Minsoo Kim, and Hyungyil Lee (Dept. of Mechanical Engineering, Sogang University, Seoul, S. Korea)
Jin Haeng Lee (Division for Research Reactor, Korea Atomic Energy Research Institute, Daejeon, S. Korea)

During indentation of brittle materials, cracks may be forming around the impression, depending on load conditions, material, and indenter geometry. We investigate the effect of indenter geometry (half-included angle, number of edges) on crack characteristics by indentation cracking test and finite element analysis (FEA). Considering conditions for crack initiation and propagation, an FE model is established featuring cohesive interfaces in zones of potential crack formation. After verification of the FE model through comparison with experimental results for Berkovich and Vickers indentations, we study the crack shapes for diverse indenter geometries in Abaqus and establish a relation between crack size and number of indenter edges. Based on this relation as well as the effect of indenter angle on crack size, a formula is established which enables us to predict from the crack size obtained with a reference indenter (such as Berkovich or Vickers) the crack size for an indenter of different geometry.

Keywords: Indentation Cracking Test, Brittle Materials, Cohesive Zone Model, Indenter Geometry, FEA

Numerical Simulation of Severe Plastic Deformation during High Pressure Torsion Processing

P. Verleysen and J. Degrieck (Materials Science and Engineering, Ghent University, Belgium)
F. Van den Abeele (Vikar R&D, Ghent, Belgium)

The principle of achieving high strength and superior properties in metal alloys through the application of severe plastic deformation has been exploited in the metal processing industry for many decades. The High Pressure Torsion (HPT) process is one of the most promising techniques for imposing very high strains to a bulk solid without introducing a significant change in sample dimensions. The HPT process involves large shear and compressive plastic deformations, and offers the possibility to deform the material under very high-hydrostatic pressures (up to several GPa), with continuous control of the degree of deformation. This process can produce exceptional levels of grain refinement, and provides a corresponding improvement in mechanical properties. It is of paramount importance that the shape of the sample is retained during the high-pressure torsion process, by means of special tool geometries which effectively

prevent free flow of the metal to obtain high-hydrostatic pressures. The design of the mold geometry and the corresponding sample dimensions is an iterative process, which is governed by the constitutive behavior of the metal alloy under investigation. In this paper, we highlight the added value of Abaqus as a powerful numerical tool to assist in the design of high-pressure torsion experiments.

Keywords: High-pressure Torsion, Plastic Deformation, Metal Plasticity, Design Optimization

Modeling of Composite Materials in SIMULIA Abaqus with the Help of Analytical Solutions of Generalized Eshelby Problem

Dmitriy B. Volkov-Bogorodsky and Sergey A. Ryzhov (TESIS Co Ltd.)

In this work, a numerical-analytical approach for the static analysis of composite materials filled by inclusions with various geometrical form and mechanical properties essentially distinguished from the properties of the matrix is presented. It is considered two basic cases of spherical and prolate spheroid inclusions, which can contain additional interface layers. For this problem, an analytical method for a two-scale analysis of composite construction (micro- and macro-level) under SIMULIA Abaqus was developed. For the micro-level problem, a special class of high accuracy analytical elements constructed on the base of spherical and generalized spherical functions has been developed. These elements analytically describe the stress/strain structure near inclusions with interface layer and strongly accounting various conditions on the curvilinear interface boundary, including slip and loss contact conditions. On the base of these elements are estimated effective properties of composite material for the macro-level problem. These elements are included into SIMULIA Abaqus through the standard UMAT technique and allow high accuracy modeling of constructions of composite materials with detailed accounting of its internal microstructure.

Keywords: Dispersed Composites, Interface Layer, Strong Analytical Representations of Stress/strain Structure, Special High Accuracy User Elements

Using Abaqus to Understand the Orthotropic Material Interactions that Enable Zero Thermal Expansion Laminates to be Designed

Colmar Wocke (The Dow Chemical Company)

Poisson's ratio effects, to a large extent, control the thermal expansion characteristics of angle-ply laminates. It is possible to design laminates that have been manufactured from some highly orthotropic laminae, to exhibit a zero coefficient of expansion along one of its directions, even though none of the expansion coefficients of the laminae themselves are negative. It is often mistakenly believed that at least one of the expansion coefficients has to be negative in order for this effect to be achieved. However, it is the internal coupling of the individual expansion coefficients via the very high laminate Poisson's ratio that actually enables this. The paper illustrates the effect via a simple Abaqus model and, in so doing, graphically cements the idea in the minds of the FEM analyst.

Keywords: Abaqus, CTE, Thermal Expansion, Orthotropic, Lamina

NATURAL RESOURCES

Forecasting Seismicity, Stability and Stress in Underground Mining

Stephan Arndt (Coffey)

Simulation methods used in open pit and underground mining have evolved from simple stress analysis to forecasting tools providing realistic simulation for challenging geomechanics problems. Whilst simulations with in-elastic constitutive models, very detailed 3D models and life-of-mine analysis scenarios spanning decades become less limited by the growing availability of computer resources, more emphasis is given to the determination of model and material parameters. The main input parameters that can be obtained from testing are

rock mass strength and in-situ stress. Tri-axial rock mass testing of samples under varying confinement pressure can be used to obtain parameters for in-elastic strain softening behavior for a Mohr-Coulomb constitutive model. Various methods exist for the determination of the in-situ stress, divided into those that require access to the location of measurement, typically associated with higher costs, and those performed on core obtained from drill holes. Ideally there is a balance between testing and simulation that ensures an efficient approach, where value for the industry lies in the predictive capability of the simulation even before data for calibration and back-analysis becomes available. Examples using Abaqus in this paper show applications including stability, stress and seismicity in planned and operating underground mining projects.

Keywords: Mining, Underground Mining, Rock Mass Plasticity, Mine-scale Models, Mohr-Coulomb, Life-of-mine, In-situ Stress, Seismicity

TRANSPORTATION & MOBILITY

Thermal Performance of Monolithic and Coated-disc Brakes Using Abaqus and Matlab Software

Abdulwahab Alnaqi, David Barton, and Peter Brooks (University of Leeds, Department of Mechanical Engineering)

The brake system is a critical item of safety equipment in a vehicle and provides the driver with adequate control over the speed of the vehicle and is classified as one of the most crucial parts in any vehicle. Rotor reliability and the thermal behavior of various small scale disc brake rotors were investigated using a small scale brake dynamometer rig and a number of different numerical approaches. Abaqus and Matlab software was first used to construct a one dimensional (1D) thermal model to investigate the thermal performance of disc brakes. The 1D model was validated with literature and experimental results. An axisymmetric thermal model was then developed using Abaqus in order to investigate the temperature distribution through the brake rotor. The effect of a coating layer on a disc brake rotor was investigated using Abaqus implicit analyses which included an enhancement of the modelling of the coating layer and as assessment of the results accuracy. Matlab code was constructed to communicate with the Abaqus input file in order to perform a repetitive braking events and optimization analysis. This code was developed in order to save time and to utilize the optimization toolbox in Matlab. The numerical results showed good agreement with the experimental results obtained from the literature.

Keywords: Heat Transfer, Disc Brake, Small Scale, Abaqus, Matlab

A Study of Vehicle-road Interaction Using Abaqus Co-simulation Approach

Amol Apte (TATA Motors Ltd.)

Early prediction of road loads is vital in vehicle development program to allow maximum flexibility for optimum design of structure. In this paper, Abaqus co-simulation-based approach is explored and successfully applied to study vehicle-road interaction. The co-simulation approach provides the user with tremendous flexibility in modeling vehicle system. The power of explicit and implicit codes can be unleashed to extract maximum benefits out of them. This combination allows user to simulate complex elements of system without putting over burden on system requirements. The ease with which highly nonlinear elements like tires and bump stoppers can be represented, free use of substructures, selective sub-cycling are the vital features of this study.

Understanding role of suspension in road load signature is equally important. Early inputs in this regard to suspension designer provide him an opportunity to manage road loads effectively. In this exercise, sensitivity of various suspension elements towards road loads is studied.

In this paper, both these aspects are discussed. It is demonstrated that how changing few suspension parameters can affect road load signature of vehicle. Currently this finding is validated subjectively. Objective validation is planned in the future. The study is planned to extend further to cover various suspension types and vehicle configurations to come up with design rules for suspension design.

Keywords: Vehicle-road interaction, Suspension, Road Loads, Co-simulation

Design and Finite Element Evaluation Using Abaqus of a Tri-axel Suspension Frame for a Passenger Bus

Ismael I. Ayala, Edgar Ramírez, Osvaldo Ruiz, and Armando Ortiz
(Universidad Nacional Autónoma de México)

The usage of modern suspension systems in passenger buses increases the performance of the vehicle to include a new kind of suspension in a passenger bus. This requires a careful examination of the packaging needs, load transfer, and structure characteristics, which leads to a safe design to be manufactured. The main objective of this paper is to provide a design proposal for a passenger bus frame to incorporate a commercial tri-axle pneumatic suspension, a system-level CAE evaluation is presented to validate and optimize the proposed design.

The main initial inputs for the design process were a set of Finite Element Analysis Simulations of the suspension to be implemented. The space requirements were determined by the design of the global bus structure. With these inputs, a series of concepts for the frame design were proposed and evaluated to determine the basis of the design to be constructed upon; a detailed design for each one of the three axles was presented and refined to a final stage where they were evaluated as assembled into the final structure.

Following the aforementioned methodology, a structure that is capable of sustaining the selected commercial suspension was obtained. The final proposal consists of detailed CAD files for each of the portions and bus frame assemblies to be constructed.

Keywords: Design Optimization, Passenger Bus Analysis, Frame Design, Structure FEM Models

Pirelli Formula1 Tire Modeling Application with Abaqus

Massimo Donatellis (Pirelli Tires S.p.A.—Formula1)

As the sole tire supplier for Formula1 Championship, Pirelli Tire provides the teams with a virtual tire model to help simulate racing conditions. The highly sophisticated body shape of a F1 car is driven by aerodynamic efficiency and tire deformation plays an important role in this context. Tire shape prediction at speeds up 300kph has been achieved by using a combination of Abaqus/Standard and Abaqus/Explicit. The numerical results have been verified using indoor test bench results. Tire shape is also the key factor for wind tunnel analysis. The same modeling technique has been applied to wind tunnel FE tire model.

Keywords: High Speed, Tire, Rubber, Viscoelasticity,*SST

The Latest Application of Abaqus for the Expansion of Simulation Ability in TMC Chassis CAE

Satoshi Ito, Phan Vinh Long, and Kouhei Shintani (Toyota Motor Corporation)
Noriko Ohtsuka and Izumi Kato (Toyota Technical Development Corporation)

TMC Chassis CAE has used Abaqus successfully for many years. This paper presents three recent examples of Abaqus analyses performed by TMC Chassis CAE.
1. Shape Optimization of Suspension Arm comprehending material and geometric nonlinearities.
2. A new method to identify material coefficients of static and dynamic characteristics of rubber using Abaqus. (Reproduce stress relaxing behavior, strain rate dependency, frequency dependency and amplitude dependency using one set of material coefficients)
3. FE Analysis Method to simulate the Static Stiffness Characteristic of Air Suspension Stiffness.

Keywords: Assembly Deformation, Design Optimization, Hyperelasticity, Minimum-weight Structures, Optimization, Rubber Bushing, Suspension, Viscoelasticity

Shape Optimization of Rubber Bushings Using Differential Evolution Algorithm

Necmettin Kaya (Uludağ University)

The main functions of a rubber bushing are basically to join the elements between rigid structures in the vehicles, isolate vibrations through to the chassis and avoid the transmission of noise. Due to the increasing interest of multibody simulations of complete vehicles or subsystems, it is important to develop and effective models to represent the static stiffness of these rubber bushings. During the vehicle development process, shape optimization of rubber products is also need to have desired stiffness curves. The objective of this study is to optimize rubber bushings to meet the stiffness curve requirements. A Fortran code has been developed to find the optimum shape parameters of 3D bushing models using differential evolution algorithm. Abaqus software has been used to create the hyperelastic finite element model corresponding to the geometric variations. Stiffness curves were obtained from the finite element analysis results, and chi-square values were used as an objective function for shape optimization.

Keywords: Rubber, Hyperelasticity, Optimization, Differential Evolution Algorithm

Application of Thermoplastic Polyester Elastomer for Honeycomb Shock Absorbing Components

Takaya Kobayashi and Yasuko Mihara (Mechanical Design & Analysis Corporation)
Katsuhisa Yamashita, Chisato Nonomura, Katsuhiro Kodama, Yumiko Isogai, and Tomoko Kanaya (TOYOBO Co. Ltd.)

Thermoplastic polyester elastomer (TPEE) possesses the properties of both rubber and engineering plastic. The most important feature of this material lies in its combining of the superior repulsion elasticity and flexibility of rubber with the rigidity of engineering plastic. This enables it to exhibit durability against fatigue, even when exposed repeatedly to large deformation. The most remarkable feature of TPEE is such that it can realize material properties with very small strain rate dependency. The authors have had their ample experiences with applying TPEE to large damper members in the civil engineering structures, such as fenders surrounding the piers of bridges to absorb the impact energy due to ship's collision, and the aseismatic connectors built in bridge structures. This paper attempts to develop shock absorbing components of small size with light weight for automobile applications. In order to achieve weight saving of the shock absorbing components, it is an effective way to utilize the buckling behaviors of thin-walled structures. With regard to buckling simulations of cylindrical shell structures, the authors have been working to develop analysis procedures for obtaining stable solutions utilizing the latest general-purpose finite element technology. This report shows that the experimental results could successfully be reproduced with applying such the simulation technique to honeycomb shaped shock absorbing components. For realizing desirable load-displacement relationship, this report also refers to how to modify the shape of the components.

Keywords: Buckling, Crashworthiness, Honeycomb, Polymer

The Use of Optimization Software TOSCA in a Standard Flexplate Design Process

Diwakar Krishnaiah (GM Powertrain)
Sreedhar Patil (TCS)
Matthias Friedrich (Dassault Systèmes Simulia Corp)

A drivetrain is only as strong as its weakest link. One of those links is the flexplate. It is used with automatic transmissions to connect the engine to the torque converter/transmission. A flexplate is a thin metal plate that operates at high RPM. Flexplates are optimized for stiffness,

strength, and mass. Stiffness controls the axial forces transmitted between the engine and transmission. The complexity of flexplate loading from the engine and transmission make it an ideal candidate for applying optimization techniques for design of a robust plate.

Keywords: Topology, Optimization, Flexplate, Torque Converter, Crank Flange, Lugs, Strength, Automatic Transmission, Design Process, Axial Stiffness, Torsional Stiffness, Bending Stiffness, Thrust, Ballooning, Load Requirements, Load Fluctuations, Torque, Misalignment, Parametric Strength Factor, Mass Reduction, Lightening Holes, Baseline, Symmetry, Constraints, Design Parameters, Design Proposals, Hybrid Applications, Sensitive, Volume Constraint, Design Variant, Feasible

Solution of Inverse Problem of Elastic-Plastic Deformation Using Isight and Abaqus

Masashi Kuwano and Makoto Saito (Nabtesco Corporation)
Yasumasa Shoji (Shoji Technology Consulting)

As a rule, compression spring shapes are designed to conform to the required compression force using general theories. Under some conditions, lateral forces on the springs can be an issue. Lateral forces are greatly affected by contact conditions between springs and structures and the spring's self-contact. Numerical calculations to determine the spring shape must take these factors into consideration. However, the manufacturing method causes a high degree of plastic deformation by a process called "prestressing," which means that the final shape of the spring is different from the coiled-metal piece required for manufacturing. So even if a shape with small lateral force is derived by calculations, the original shape of the spring before prestressing must be determined from that shape.

The authors propose an iterative method for solving this problem. The method estimates a reasonable candidate for the original shape and refines that shape using the results of repeated elastic-plastic analyses. An automated system integrating each process with Isight and Abaqus was designed. Although it is generally difficult to change the shape of a part in Abaqus, an analysis template was developed and made the automation of this procedure possible.

Keywords: Spring, Inverse Problem, Analysis Automation, Plastic Deformation, Isight

Development of Analytical Prediction Method for Pulley Bolt-loosening

Chang-Ho Lee, Chi-Un Kim, Young-Jin Kim, and Hyun-Soo Ryou (R&D Center, Hyundai Motor Company)
Sung-San Cho (Department of Mechanical & System Design Engineering, Hongik University)

Many auxiliary parts (e.g. the water pump, the oil pump, the alternator, etc.) in an engine are driven by the crankshaft. For this, a pulley is assembled to the crankshaft by a bolt. A bolt-loosening of the pulley can cause noise and vibration and bring engine breakdown in severe case. A finite element analysis procedure and a rig test were developed to predict a bolt-loosening of the pulley. The FE analysis results agree with the rig test results and verified with the real engine test. The parameters making worse the bolt-loosening are investigated. They are the torsional and lateral vibration of the crankshaft, the belt tension, the belt torque variation and the pulley inertia. Authors thought this procedure developed here can be applied to solve a bolt-loosening in other system.

Keywords: Engine, Crankshaft, Pulley Bolt, Bolt-loosening, FEA

The Use of Abaqus in an Engine Bearing Design Environment

David R. Merritt and Omar Mian (Mahle Engine Systems Ltd., UK)

Mahle Engine Systems has developed, and maintains, an internal software code called SABRE-EHL for Elastohydrodynamic (EHD) bearing simulation. The target components are oil-lubricated journal bearings within an internal combustion engine—primarily the crankshaft bearings—small end bushes, camshaft and balancer shaft bearings. The SABRE-EHL tool is used to optimize bearing design and investigate customer problems.

Abaqus is an integral part of the analysis process, proving data related to the structural stiffness of the housing, assembly deformations, inertia effects, and journal/crankpin misalignment. This paper provides an overview of the different types of Abaqus analysis which have been developed and deployed in order to extract the required input data for SABRE-EHL.

In addition to providing input data to SABRE-EHL, Abaqus can also be used to apply bearing pressures back onto the structural model. SABRE-EHL oil film pressures are mapped onto the Abaqus bearing mesh and used to perform a nonlinear engine cycle analysis where housing stresses and detailed housing-bearing contact interactions can be investigated.

Keywords: Automotive, Journal Bearings, Assembly Deformation, Bolt Loading, Elasticity, Experimental Verification, Powertrain, Substructures, Elastohydrodynamic, Lubrication

Weld and Adhesive Optimization Process Development and Implementation on Vehicle Body Structure Development

G. Nammalwar, B. Shahidi, N. Kochhar, A. Chator, L. Sivashankar, B. Barthelemy and R. Frank
(Ford Motor Company)
R. Krishnan and Rakesh K. D. (DEP Inc.)

Passenger vehicle structural performance is extremely sensitive to welds and adhesive bonds. Traditionally, multi-disciplinary optimization across different disciplines is performed largely with thickness, shape and material grade as variables. The objective of this study was to optimize the spot weld count and linear length of adhesives in the body while balancing vehicle structural performance and weight. Various optimization scenarios were carried out such as a.) maintain current structural performance but minimize weld count, adhesive length and body weight, b.) maintain current weld count and adhesive length but maximize structural performance and minimize weight, etc. Including welds and adhesives as variables in the Multi-Disciplinary Optimization (MDO) processes provides additional design space to improve structural performance and reduce cost through spot weld and adhesive minimization. This paper presents an automated Weld & Adhesive Optimization process developed at FORD to improve vehicle structural performance using a unique parametric technique developed by DEP, Inc. Variables include weld pitch and its alternative forms such as number of welds in a given weld line and % change to the weld count, type of welds, layers of adhesive bonds and adhesive bond dimensions. This process was implemented successfully on a passenger car currently in production with the primary objective to increase modal frequency separation between body-in-prime bending and torsional modes while maintaining the current number of total welds in BIP structure. Majority of the design variables were weld parameters at rocker, front and rear rails, roof liners, seat cross members, roof bows and sled runner locations. Parameters were automatically created using DEP's Meshworks software. It generates weld/adhesive lines automatically once parts are identified. Variables are created quickly and Design of Experiments (DOE) techniques are used to generate multiple designs by integrating the parametric model with Isight. Over 1000 designs were generated rapidly and automatically and evaluated across functions and attributes. Scripts were created for automatic post processing of results. Subsequently, an input-output matrix was tabulated and Response Surface Models (RSMs) were created using Isight's RBF tool. Isight's Pointer Algorithm was set to determine the optimal solution. The validated optimal design showed the required frequency separation with optimal placement of welds and adhesives. This process is a key enabler to execute weld and adhesive optimization with minimum resources to achieve optimum number of welds, placement of the welds, and adhesive length in a fast turnaround time demanded by current product development cycle.

Keywords: Design Optimization, Isight, Weld and Adhesive Bonds, Structural Performance, Multi-disciplinary Optimization, MDO

Structural Topology Optimization of Car Body Using ATOM

Seehe Oh (Hyundai Motor Company)
Sang-gyu Park (Dassault Systèmes Korea Corp)

The weight reduction of components and systems is of utmost importance in the automotive industry. Reducing weight translates into higher performance and lower fuel consumption.

In this paper, we suggest an optimization process for linear and nonlinear load cases of a car body. In order to achieve this goal without sacrificing the current performance of selected model, we use Abaqus Topology Optimization Module (ATOM). Current geometry models are simulated with the specified load cases to establish a baseline performance in terms of stiffness and reaction force. We then generate design envelopes representing the available space for topology optimization and the same load cases are applied to the optimization models with the volume as upper limit constraints, the reaction force as lower limit constraints and minimum strain energy as objective functions.

Optimization results suggest new conceptual sections for important components and new positions of structural adhesives compared to the current model with significant increase in stiffness and reaction force.

Keywords: Nonlinear Analysis, Strength Optimization, ATOM, Abaqus Topology, Optimization Module

Analysis and Optimization of a Passenger Bus Frame through Finite Element Software

Carlos A. Reyes Ruiz, Edgar I. Ramírez Díaz, Osvaldo Ruiz Cervantes, and Armando Ortiz Prado (Universidad Nacional Autónoma de México)

The objective of this work is to analyze and optimize a passenger bus frame using finite element software, in different static and dynamic conditions.

Through a static analysis, torsion and bending constants were extracted using boundary conditions different sets for each one of them. With a linear perturbation step, free and forced vibration modes of the system were obtained, and a frequency response analysis was developed.

For dynamic analysis, boundary conditions were defined from dynamic automotive equations. Evaluated conditions were suspended weight, acceleration, braking, cornering, and cornering and braking together.

The bus body geometry as well as different suspension components was obtained directly from the CAD files. The degrees of freedom were also identified from the original drawing, either movements between the suspension components as between suspension and bus body elements. Most components were modeled by wire elements, and some others as shell, to which the mechanical properties of steel and different cross-sections were assigned. Additionally, connectors were used to model dynamic components such as air springs, shock absorbers and tires, whose behavior was described by characteristic curves. This modeling methodology allowed us save time and computational capacity.

The most critical elements were found to be air springs fasteners and surrounding elements, which according to dynamic analysis carries a load elevated percentage of the load, while for curving conditions the torsion bars fasteners and nearby elements became critical components. After optimization process, structural frame weight was reduced 7.76% and, a decrease up to 50% of stress concentration at drive and auxiliary axles fasteners was reached.

Keywords: Design Optimization, Minimum-weight Structures, Multi-body Dynamics, Bus Structure Models, FEM Modeling

On the Simulation of Short Fiber Reinforced Engine Components

Tamás Schmidt (Audi Hungaria Motor Ltd., Engineering Development Department)

Lightweight technology is a very important part of today's engineering practice. The discussion of CO2 emissions and fuel consumption gets more and more important. Hybrid vehicles could save fuel, but are also heavier. AUDI has made a mark with innovative new technologies in the hybrid models Q5, A6, A8, and has shown, with the new A6 and A3, that a car could be lighter than its previous release. All these leading technologies are summarized in Audi Ultra. There are several ways to get a lighter construction; for example, with new materials and technologies, to perform more realistic simulations, or to use the capabilities of the material as much as possible. Another way is to optimize structures and manufacturing processes. This paper is intends to describe the advanced technology of simulating short fiber reinforced plastic engine parts. With the coupling of injection molding simulations and structural FEA, it is possible to use the anisotropic material behavior resulted from fiber orientation. Identification of manufacturing process induced weak points becomes possible and a more realistic behavior of the part can be predicted. By the simulation of composite parts, many more questions have to be considered. Problems of mesh density, material model description, tolerances, failure methods, and indicators may arise earlier than in case of simulating metallic components.

Keywords: Automotive, Anisotropic Material Model, Composites, Converse, Digimat, Injection Molding, Eigenfrequency Calculation, Failure, Fiber Reinforced Composite Material Model, Finite Element Simulation, Integrative Simulation, Micromechanics, Moldflow, Orientation Mapping

Application of CAE to Aluminum Wheel Impact Test Analysis Using Dynamic Implicit

Juntak Shin, Yeonsang Yoo, and Sungjin Yoon (Hyundai Motor Company)

Aluminum Wheel Impact Test (13 Degree Lateral, SAE J157) is the way in order to guarantee the aluminum wheel strength. The CAE Team at Hyundai Motor Company has been carrying out the wheel impact test analyses using CAE (Abaqus/Standard), and developing the accuracy of analysis results by comparing with the tests.

Before this study, we carried out the aluminum wheel impact test analysis with only wheel model (without test jig). And, we used static analysis. The impact load used in static analysis was calculated by an equation related to the impact theory. But, this analysis was not sufficient condition in order to simulate the real test. It must be considered the test jig stiffness. Sometimes, not considering test jig analysis, results didn't show the highest stress area (especially, nearby bolt hole). Also, dynamic behavior in analysis is more effective if considering.

In this study, aluminum wheel impact test analysis will carry out with test jig model. We will use dynamic implicit analysis (Abaqus/Standard) in order to simulate the rear impact test.

Keywords: Aluminum Wheel, Aluminum Wheel Impact Test (13 Degree Lateral), Aluminum Wheel Impact Test Jig, Dynamic Implicit Analysis

Using Abaqus/Standard and Abaqus/Explicit for the Development of New Bi-level Trains at Stadler Rail

Alois Starlinger, Benedetto Castelli, and Daniel Egger (Stadler Rail)

In the development of the new bi-level train, KISS, Abaqus/Standard and Abaqus/Explicit have been intensively applied to optimize the structure with respect to stiffness and strength as well as to crashworthiness. Models up to 3 million shell elements have been used to prove the strength of the load carrying structure with respect to the standard EN12663-1. By using Abaqus/Standard, the calculation times were minimized allowing for a significant number of iterations to optimize the structure with respect to lightweight engineering and reduction of manufacturing time. Additionally, the Abaqus/Standard result files were used to investigate the fatigue strength of the structure, especially of the welds, with FEMFAT, a

post-processor code from Magna Powertrain. The fatigue strength was proven for typical load collectives representing the operational life for 40 years of operation. In simultaneous analyses with Abaqus/Explicit, the crash energy management structures have been optimized. All requirements of EN15227, the European crashworthiness standard, were successfully fulfilled. Comparisons with dynamic test results have shown that the numerical results are very close to the test data. Abaqus/Standard and Abaqus/Explicit have helped to minimize the overall development time of the structure of the new Stadler bi-level train KISS.

Keywords: Crashworthiness, Damage, Design Optimization, Dynamics, Elasticity, Experimental Verification, Failure, Fatigue, Fatigue Life, Impact, Minimum-weight Structures, Optimization, Plasticity, Postprocessing, Railcar, Seam Welding, Shell Structures, Substructures, Visualization, Welding

Simulation Creep Behavior of Plastic Parts During Complex Thermo-mechanical Loading

Jan Steklý (IDIADA CZ a.s.)

The paper describes the use of Abaqus for evaluating the behavior of car plastic parts which are exposed to high temperature and self weight (plus another load) for long time. Car plastic parts, inside and outside, are subjected to complex loads – temperature, vibration, and environmental conditions. It is a challenge to prepare simulations which correctly describe the complex behavior - temperature dependent, time dependent of stress-strain behavior. To capture visco-elastic behavior, we chose a creep description of the material. We discussed choosing suitable material in Abaqus for the description of creep behavior. Base on measurements (of specific load cases), we define a process for defining material Abaqus cards. And finally, we describe the results validation of simulation on real parts. There were some specific requirements to evaluation of result – effectively measure the change of gaps between parts in dependency on position of measurement lines which have to be solved.

Keywords: Viscoelasticity, Creep, Power Law

The Extremely Easy External CFD Tool for Stylists

Takashi Takiguchi, Kenta Ogawa, Tateyama Hiroyuki, and Tatsuya Oda (Honda R&D)

For early stage car development, combining the car package and the external aero performance is crucial. To investigate the external aero performance for each package, fast and accurate prediction is required. In this time, an extremely easy external CFD tool for stylists called FAST using SEE was established. The only thing to calculate CFD is to create the external car shape. This tool prepares the floor and tire; the meshing and calculation will be finished automatically. The CFD result will come to the user, which is system feature one. The user should create only external car shape, which allows for zero human time for CFD. The user inputs the geometry preparation of car dimension, height and wheel size, etc. To begin, the system creates the floor by stretching from existing floor. The tire is located in a suitable position by the wheel base. The second step is meshing, calculations, and post processing. The assembled geometry (exterior, stretched floor and tire) will be meshed and calculated, and then some pictures are created as a standard post process. Fast Whole process will finish in two hours by parallel computing. Reasonable accuracy capturing CD trend is enough for early stage car development.

Keywords: CFD, Computational Fluid Dynamics, Isight, SEE, Car Shape, Post Processing

NOTES

INDEX

DASSAULT SYSTEMES

Delivering Best-in-Class Products

CATIA
Virtual Product Design

GEOVIA
Model and Simulate our Planet

SOLIDWORKS
3D for Professionals

EXALEAD
Information Intelligence

SIMULIA
Realistic Simulation

NETVIBES
Dashboard Intelligence

DELMIA
Virtual Production

3DSWYM
Social Innovation

ENOVIA
Global Collaborative Lifecycle Management

3DVIA
Online 3D Lifelike Experiences

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 150,000 customers of all sizes, in all industries, in more than 80 countries. For more information, visit www.3ds.com.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Visit us at
3DS.COM/SIMULIA