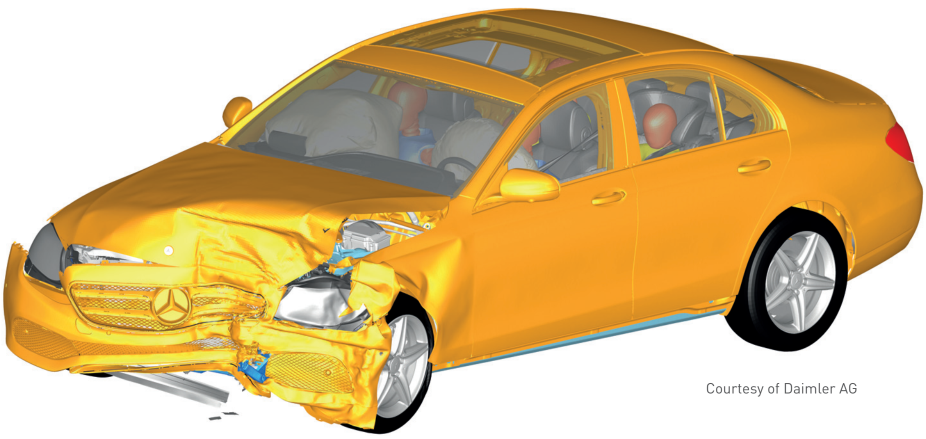
Book of Abstracts

11th EUROPEAN LS-DYNA CONFERENCE

9 - 11 May 2017 – Salzburg, Austria

Copyright Tourismus Salzburg GmbH



Courtesy of Daimler AG

PLATINUM SPONSORS

Book of Abstracts

11th EUROPEAN LS-DYNA CONFERENCE

9 - 11 May 2017 – Salzburg, Austria

The conference is organized by

in association with

DYNAmore GmbH, Industriestr. 2, D-70565 Stuttgart, Germany
Tel. +49 (0)711 - 459600 – 0, E-Mail: info@dynamore.de, Web: www.dynamore.de

ISBN 978-3-9816215-4-9

SPONSORS

Platinum

Gold

Silver

CRASH – PLASTICS FAILURE **PAGES 39 - 43**

L0 – Room A, Tuesday, 9 May, 15:40 – 16:55 Page

OCCUPANT SAFETY I – *ANDREAS HIRTH MEMORIAL SESSION* **PAGES 44 - 45**

L0 – Room B, Tuesday, 9 May, 15:40 – 16:55 Page

PROCESS – METAL FORMING I **PAGES 46 - 48**

L0 – Room C, Tuesday, 9 May, 15:40 – 16:55 Page

CRASH – METAL FAILURE **PAGES 77 - 82**

L0 – Room A, Wednesday, 10 May, 08:30 – 10:10 Page

OPTIMIZATION – GENERAL **PAGES 83 - 85**

L0 – Room B, Wednesday, 10 May, 08:30 – 10:10 Page

PROCESS – CONNECTIONS **PAGES 86 - 89**

L0 – Room C, Wednesday, 10 May, 08:30 – 10:10 Page

IMPLICIT MECHANICS — PAGES 193 - 194

L0 – Room B, Thursday, 11 May, 10:40 – 12:20 — Page

PROCESS – MISC. — PAGES 195 - 198

L0 – Room C, Thursday, 11 May, 10:40 – 12:20 — Page

MATERIALS – LAMINATED GLAS — PAGES 199 - 202

L1 – Room D, Thursday, 11 May, 10:40 – 12:20 — Page

FUJITSU

A success story for almost 40 years

For almost 40 years Fujitsu provides HPC users the computing resources they need to study scientific and technical problems on the basis of computer simulations. Our cutting-edge technologies enable them to make vital decisions, to promote product innovations, to speed up research and development, and to reduce time to market maintaining competitive advantage.

Today Fujitsu is the most experienced and largest provider of HPC solutions in the Asian market, and has been a leading HPC vendor in Europe almost from the beginning. Fujitsu has established close collaboration with leading ISVs and has jointly developed PRIMEFLEX for HPC cluster solutions optimized to application specific workload requirements. Furthermore, a global HPC competency network supports and further develops the full range of Fujitsu's HPC solutions.

High Performance Computing with Fujitsu HPC Cluster Solutions

High Performance computing tasks are not alike – nor are the requirements of users. PRIMEFLEX for HPC cluster solutions from Fujitsu embraces these diverse needs: from workgroup-sized configurations up to divisional-sized HPC cluster environments. Building on highly reliable hardware components supported by a fully integrated dedicated HPC solution stack – Fujitsu's Software HPC Cluster Suite – PRIMEFLEX for HPC Cluster solutions are designed to provide an optimal price/performance and faster time to value for a variety of operational and workflow scenarios. Ready-to-use delivery of HPC infrastructures guarantees rapid deployment for production and includes Intel Cluster Ready certification for proven performance and trouble-free operation.

Making the IT simple is only the first step. PRIMEFLEX makes using the applications simple. Fujitsu's Software HPC Gateway, integrated within the PRIMEFLEX software stack, is a wide-ranging application platform that provides a workplace adaptable for all users, from novice to expert. Its web desktop is intuitive and ready for mobility. And a growing range of optional add-ons are available to support the business, from automated workflow to accelerated remote visualisation, from search to analytics.

www.fujitsu.com/hpc

intel

Intel Corporation is a world leader in computing innovation.

Intel Corporation is a world leader in computing innovation.

We design and build the essential technologies that serve as the foundation for the world's computing devices. From workstations to the world's most powerful supercomputers, ever-higher performance for technical computing applications is needed to speed time to results, handle today's unprecedented growth in data volumes, and improve the accuracy and precision of modeling and simulation applications. Intel® architecture is designed to address the heavy demands of technical computing at every scale, so users can continue to push the boundaries of discovery.

To learn more about Intel in technical computing, visit

www.**intel.com/hpc and @IntelHPC**

LSTC was founded in 1987 by John O. Hallquist to commercialize as LS-DYNA the public domain code that originated as DYNA3D. DYNA3D was developed at the Lawrence Livermore National Laboratory, by LSTC's founder, John O. Hallquist.

A team of engineers, mathematicians, and computer scientists are engaged in the development of LS-DYNA, LS-PrePost, LS-OPT, LS-TaSC (Topology), and LSTC's Dummy & Barrier models for use in various industries, including Automobile Design, Aerospace, Manufacturing, and Bioengineering. LS-DYNA development is focused on one code methodology that includes Implicit, Explicit, SMP and MPP solvers. It is optimized on all platforms including clusters running Unix, Linux, and Windows operating systems.

LSTC is focused on technical excellence and support. The worldwide growth of the LS-DYNA user base is attributable to both the quality of LSTC's software and the extraordinary efforts regularly made by the technical staff to ensure customer satisfaction.

With regard to LS-DYNA pricing, LSTC consistently offers a more competitive product because 90% of the work force is solely dedicated to technical endeavors. LSTC directs its resources towards producing high quality software and supporting licensee's projects.

www.lstc.com

Rescale's™ turn-key cloud platform, ScaleX™, allows engineers to run LS-DYNA and other simulation software instantaneously on the cloud. ScaleX's SaaS graphical user interface puts the world's largest multi-cloud HPC infrastructure network at engineers' fingertips. A broad selection of HPC hardware, including the latest GPUs and InfiniBand interconnect, allows users to customize their hardware configurations for each job. Users can use traditional annual or paid-up licenses or purchase hourly, on-demand licenses to meet variable simulation requirements. On-demand hardware and software allow LS-DYNA users to fully leverage the elasticity of the cloud.

ScaleX incorporates a number of collaboration and administrative features that make it an enterprise-grade solution. Users can easily share workflows, input/output files, and logs to ensure a seamless, scalable, worry-free user experience. The ScaleX Enterprise administrative portal allows managers to set permissions and controls around users, budgets, hardware, and license server connections. Organizations can tailor resources to meet the diverse budgets and compute needs of their internal divisions.

The ScaleX platform is the most secure cloud solution available. Rescale's multi-layered security framework ensures complete data confidentiality based on the highest standards of data encryption. Rescale meets or exceeds industry standards, is independently audited, and conforms to assurance programs and controls. To comply with legal, privacy, and export compliance requirements, data is bound to the region in which the platform resides—the United States, Europe, Japan, or South Korea.

For more information on Rescale's ScaleX cloud platform, visit

www.rescale.com

4a engineering offers the testing system 4a impetus which enables the automatic mechanical characterization of dynamically loaded test specimens or components. 4a impetus is the first testing system in the world that constitutes a self-contained process creating validated material cards (“reverse engineering”). In the beginning specimens are cut out from real parts and loaded under realistic conditions. Finally validated material cards are determined using numerical methods. The implemented methods are intelligent analysis of analytical models, numeric FE-Solvers like LS-DYNA, as well as optimization like LS-OPT. The testing system can be operated on a common office desk and provides an intelligent, flexible and user-friendly system to shorten development time and to enable cost reduction.

www.4a-engineering.at

BETA CAE Systems offers state-of-the-art simulation solutions that exceed the requirements of all engineering disciplines, for many sectors, including the automotive, motorsports and aerospace. Our passion for engineering and excellence empowers us to develop solutions by listening to your needs, taking up even the most demanding challenges.

www.beta-cae.com

CEI - founded in 1994 with Headquarters in Apex, NC, USA – has worldwide subsidiaries in Germany, China, India and Japan. CEI is the creator of EnSight, the leading software solution for visualizing, analyzing, and communicating data from complex mono- and multidisciplinary numerical simulations. It supports an extraordinary number of solver data formats. The open interface architecture allows for easy integration with any data format, and a Python API empowers users to customize the software to meet their specific needs. With support for parallel data processing and distributed graphics rendering, it is the leader in handling very large models.

www.ensight.com

Through “our wisdom, our future”, Shanghai Hengstar Tech. and Shanghai Enhu Tech. mainly provide CAD / CAE / CAM cloud service, including: hardware cloud service, software cloud service, technical cloud service, and secondary development and application, etc. With internet technology and global technical source, Hengstar & Enhu focus on helping large, medium and small enterprises in China and other countries to learn and share the advanced design strategies and methods, to effectively use the advanced CAD / CAE / CAM tools (e.g. LS-DYNA) to achieve product design and manufacturing innovations, to improve product quality and consequently competitiveness.

www.hengstar.com
www.enhu.com

JSOL Corporation provides cutting edge CAE technology and technical consulting as a distributer of LS-DYNA and our own software products. We also offer expertise in modeling and its know-how as well as a software based on the expertise. JFOLD is an analytical software which simulates complex airbag folding and deployment that plays a critical role in the analysis of a car crash. J-SEAT designer is an integrated simulation tool for seat design, which contains information about regulation and assessment for automotive safety. We also develop simulation tools for press forming - JSTAMP for analyzing metal forming and J-Composite for composite material. Please come to visit our exhibition booth for more details.

www.jsol.co.jp/english/cae

NEC is a leading provider of HPC solutions, focusing on sustained performance for real-life scientific and engineering applications. To achieve this goal NEC delivers technology and professional services to industry and academia. Linux-based HPC clusters as well as our high-end vector systems meet the different needs of different customers in the most flexible way. Energy-efficiency is one of the key design objectives, addressed by advanced cooling technologies or by the high-bandwidth SX vector architecture, which delivers unprecedented efficiency on real world code. The service capabilities from the operation of complex systems to the optimization of scientific codes and NEC’s storage-appliances complete our solution offering.

www.emea.nec.com

Oasys
LS-DYNA ENVIRONMENT
The Software House of Arup

Oasys Ltd is the software house of Arup and distributor of the LS-DYNA software in the UK, India and China. Oasys Ltd develops the Oasys Suite of pre- and post-processing software for use with LS-DYNA. The Oasys team has been working with Livermore Software Technology Corporation (LSTC), the developers of LS-DYNA, for over twenty years and have an in-depth knowledge of this powerful analysis tool. During this period, Oasys has been involved in developing some of the features within LS-DYNA such as seatbelt system modelling, staged construction, and various material models.
The Oasys Suite of software, written exclusively for LS-DYNA, is at the leading edge of the pre- and post-processing markets and is used worldwide by many of the largest LS-DYNA customers.

www.arup.com/dyna

SCALE
IT-Solutions for CAE

SCALE offers software solutions and IT services for process and data management in the automotive and other industries. As a subsidiary of DYNAmore GmbH, SCALE has a strong background in CAE applications and processes. In particular, our range of services includes software development for process and data management, finite element method development, and numerical optimization for the functional design of vehicle components.
SCALE's lineup includes our standard products CadMe, LoCo, CAViT and Status.E for simulation data, process and requirements management, as well as IT services for individual software solutions on request. Our software products support the entire lifecycle of the typical CAE design workflow:

CAD data meshing model -> assembly -> solving -> post processing -> reporting and monitoring

If requested by the customer, the software modules can be individually combined or integrated as desired. In addition to software development and standard products, SCALE offers consulting services for assessing and optimizing your IT environment, requirements analysis, IT architecture design, project planning and management, etc. The name SCALE stands for „Scalable Solutions in Simulation Data and Process Management“ and our staff at SCALE are a mix of experienced CAE engineers and professional computer scientists. The majority of our employees are based at the Dresden site and benefit from the excellent scientific environment there.

www.scale.eu

Keynote Sessions

Recent Developments in LS-DYNA – Part I

J. O. Hallquist, R. Grimes, J. Xu, G. Cook (LSTC)

On Computational Strategies for Fluid-Structure Interaction: Algorithmic Developments and Applications

Prof. D. Peric, W. G. Dettmer, C. Kadapa (Swansea University)

CAE-Based Safety Development of the All-New Volvo S90/V90/V90CC

J. Jergeus, D. Macri, P.-A. Eggertsen, I. Jenshagen, U. Westberg, M. Khoo (Volvo Cars)

You want me to do what?!

E. DeHoff (Honda R&D Americas)

Simulation of Mechanical Watches at IWC

P. Steinhaeuser (IWC Schaffhausen)

Technical Challenges in the Integration of Hybrid-Components in New Automotive Concepts*

D. Moncayo, Prof. C. Glöggler (Daimler)

Modelling of Adhesively Bonded Joints in CAE-Models at Porsche – Look behind the Scenes

F. Burbulla (Dr. Ing. h.c. F. Porsche)

Recent Developments in LS-DYNA and LS-OPT – Part II

J. O. Hallquist, P. L'Eplattenier, N. Stander, Y. Huang, S. Bala, F. Del Pin (LSTC)

Tuesday Afternoon Sessions

15:40 – 16:55

Implementation of a VE-VP Material Law for the Simulation of Energy Absorbing Thermoplastic Components

P. Du Bois (Consultant);
M. Feucht, J. Irslinger (Daimler);
T. Erhart (DYNAmore)

The predictive simulation of the deformation and energy absorption of thermoplastic components during a crash event has been an elusive goal for the last decades. One reason for that in many cases is that the rate dependency of thermoplastics is not limited to the domain of permanent deformations but also exhibits itself during elastic deformation processes. Whereas classical viscoplastic (VP) material laws are perfectly able to cope in one way or another with the viscous stresses beyond the yield surface, they are limited to purely elastic (non-viscous) response inside the yield surface.

One of the more frequently used material laws for the simulation of thermoplastics in LS-DYNA is MAT_187 or SAMP (1). This is a classical VP material law allowing for separate consideration of uniaxial strength, shear strength and biaxial strength using a multi-linear yield surface and allowing for the simulation of permanent volumetric straining through the use of a non-associated flow rule. Our current approach consists of extending the SAMP capabilities by making the response inside the yield surface viscoelastic rather than linear elastic. This is done through the introduction of a viscoelastic stress component defined by a single Prony series term. The decay constant is internally recomputed every time step in order to allow for a non-linear dependency of the Young's modulus upon the strain rate corresponding to experimental data. Verification and robustness testing was performed and a first set of real material data was determined for Polypropylene. The proper functioning of the new module was then confirmed by the simulation of dynamic tensile tests at different speeds.

This presentation will describe the VEVP extension of SAMP, point out its potential as well as its limitations and illustrate the practical application using the example of an interior head impact simulation.

Literature

[1] DuBois, Kolling, Feucht, Haufe, A semianalytical model for the simulation of polymers 4thLS-DYNA Forum, Bamberg, 2005.

Modelling of Ductile Polymer Model for Crash Application

Y. Ngueveu, S. Miyagano (Toyota Motor);
F. Lauro (Université de Valenciennes);
R. Balieu (KTH Royal Institut of Technology)

Due to their lightweight and mechanical properties, ductile polymers are widely used in automotive industry. Those polymers play an important role during pedestrian protection performance as they are designed to absorb some of the energy induced by the collision with the pedestrian. To achieve a good and robust design fulfilling our target, CAE simulations are used.

However, a mismatch between CAE results and experimental test can arise, leading to an extra loop in the development process. Root cause analysis showed us that it occasionally comes from the CAE model of polymer. Current CAE material model for polymer still needs some improvements in order to capture mechanical phenomena exhibited by polymer during those impacts (Visco elasticity, pressure dependent plasticity, etc…).

We have developed a new material model considering all those phenomena [1] [2]. To validate this model, simulations were performed using LS-DYNA, and correlated with experimental tests. Tests were conducted, firstly to validate features implemented in this new model, secondly to check its effectiveness with complex loading case, and finally to verify its accuracy for vehicle development. The newly developed model correlated well with experimental test and improved our CAE accuracy for pedestrian protection performance. This shows that we are in the good direction to improve our CAE capability. This comes however with a trade-off which is the increase of CPU cost. A more thorough investigation is being done to check whether or not the CPU cost can be reduced.

Literature

[1] BALIEU R., LAURO F.: A fully coupled viscoelastic-viscoplastic damage model for semi-crystalline polymer, PhD Thesis, 03rd December 2012, Valenciennes, France.
[2] BALIEU R., LAURO F., BENNANI B., DELILLE R., MATSUMOTO T., MOTTOLA E.: A fully coupled elastoviscoplastic damage model at finite strains for mineral filled semi-crystalline polymer, 12th February 2013, Valenciennes, France.

Characterization and Modeling of the Deformation and Failure Behavior of Neat Thermoplastic Homopolymers under Impact Loading Conditions

P. Stelzer, Z. Major (University of Linz)

The increasing use of thermoplastic polymers in structural applications exposed to impact loading conditions drives the need of accurate and reliable FEM simulations. Classical elasto-plastic formulations are based on the von Mises yield criterion and only of limited suitability for modeling polymers. Advanced material models become necessary to capture the complex mechanical deformation and damage behavior. However, the experimental effort is significantly higher to provide material input data. The objective of this study was the modeling of the deformation and damage behavior of thermoplastic homopolymers under monotonic and impact loading conditions with the commercial FEM code LS-DYNA. For this purpose, the use of the material model SAMP-1 and the damage model GISSMO for a Polypropylene and a Polycarbonate grade was investigated.

A comprehensive experimental characterization, applying full-field strain analysis using a digital image correlation technique and a high speed camera, was performed for the calibration of the models. To derive the input parameters for SAMP-1, uniaxial tensile and simple shear tests were carried out. Experimental compression data was taken from previous studies. Coupon tests were conducted to assess the stress triaxiality dependent damage and failure behavior. To describe the actual local deformation behavior and to provide a straight-forward methodology, a direct experimental approach was favored to derive the material model input data. Validation was performed on the tensile tests by comparing the numerical and experimental results of the global force-displacement curves and of the local deformations.

The elasto-viscoplastic constitutive model SAMP-1 was calibrated for ambient temperature (23 ± 1 °C) and different strain rates (0.0002, 0.1, 1, 10 and 100 s^{-1}). Until the yield point is reached, the mechanical response is linear elastic. The characteristic pressure dependent quadratic yield surface of SAMP-1 was fitted by the input of tensile, compression and shear data. Moreover, the model incorporates multiaxial hardening behavior. The non-isochoric plastic deformation could be captured by the experimentally determined apparent Poisson’s ratio.

The determination of the failure curve, in terms of failure strains as a function of the stress triaxiality factor, is essential for the calibration of ductile damage models like the formulation implemented in SAMP-1 or GISSMO. A combined experimental-numerical approach was used to identify equivalent plastic strain values and corresponding stress triaxiality factors at fracture initiation. The method of an average triaxiality factor was selected as a rather direct way to extract the failure strains from the experimental tests.

Application of Reduced Model to Estimating Nij of HYBRID3 AF05 Dummy in Sled FE Simulation

T. Yasuki (Toyota Motor)

Nij is one of dummy injury scale measured by HYBRID3 AF05%ile dummy, and there was difficulty of optimizing Nij by response surface method due to availability of the several peaks of Nij in time domain.

In this study, “reduced model” was employed to negotiate the difficulty. At first, HYBRID3 AF05%ilt FE model sitting on a sled FE model was generated, and nine parametric studies were performed by changing two parameters including air bag and seat belt. These results were utilized to establish “reduced model” of HYBRID3 AF05%ile and the sled by ODYSSEE developed by CADLM and presented at 10th European LS-DYNA Conference.

Next, the reduced model was used for estimating Nij as a function of time by ODYSSEE for sixteen cases with updated parameters including air bag and seat belt. The sixteen Nij as a function of time were reconstructed by the reduced model, and they were compared with sixteen results by the FE models. They showed good correlations with the result of the FE models.

The reduced model was also applicable to estimating time history of dummy injury scales including head G, neck tension force, chest deflection, and femur axial force.

CAE Prediction of H-Point (Occupant Positioning in the Vehicle) using LS-DYNA, ARUP-HPM Tool

C. G. Thangam, F. Eklöf, E. Mårtensson, P. Setterberg, J. Lindberg, S. Johnsson (Volvo Cars)

At Volvo Car Corporation costly and time consuming physical test loops are increasingly replaced with virtual prototyping using many CAE tools such as LS Dyna code. Computer models of seat will enable comfort analyses of prototypes in early stages of the design process. Accurate CAE prediction of h-point (occupant positioning in the vehicle) will enable ensuring Seat Static Comfort.

Recent studies at Volvo Car Corporation showed that good h-point prediction level can be achieved. And subsequent validation with physical measurements showed us that in spite of challenges in modelling complex material properties of polyurethane Foams, inherent variations in the seat manufacturing process, ARUP – HPM tool can be used to predict h-point. Further ongoing investigations and feasibilities with Seat trim assembly process simulation using J-Seat will also be touched upon.

Andreas Hirth's Contributions to the World of Occupant Safety Simulations

Longtime work companions

Forming Simulation, Meta Language and Input Decks

M. Fleischer, J. Sarvas, H. Grass, J. Meinhardt (BMW Group)

The use of finite element (FE) simulations for a virtual validation of the forming process for sheet metal parts has been introduced in the mid 1990s and is state of the art in the automotive industry today. Two challenging tasks for determining whether a tool design and its process parameters are feasible are the prediction of the material behavior during the forming process and the springback of the final part.

For improving the predictive accuracy of the forming simulation, the level of detail has increased steadily regarding many aspects of the simulation model. For example, the material behavior during drawing is influenced by the preceding trimming operation as the latter causes damage at the trimmed edge. Furthermore, during the drawing the pressure distribution between the blank and the blankholder may vary significantly due to the deflection of the tool and the press. This can result in a disadvantageous restraining behavior. Considering these effects may lead to a further improvement of the simulation's accuracy.

As a result, the increase in size and level of detail of the FE models poses a challenge for the future simulation systems and their application.

One way to deal with this complexity is a process orientation in the software workflows. The goal is to use of the best suitable program for each step in the simulation system. Therefore in the first step, pre-processor, solver and post-processor should be independent from each other. Therefore a neutral interface between the pre-processor and solver was generated with the meta language "OFPL". With this meta language it is possible to describe metal forming processes in a process- and object-oriented way with a minimum of numeric parameters. Now, the next step is to model the solver input decks with object- and process-orientation.

Experimental Validation of Detecting Surface Deflections on Sheet Metal Parts with LS-DYNA

A. Weinschenk, A. Schrepfer, W. Volk (TU München)

The appearance of surface deflections on sheet metal parts is undesirable. When surface deflections are detected on a sheet metal part during tryout, the tool geometry has to be modified. This procedure is performed at a later stage of the product development process which leads to high costs and effort. Therefore, it is useful to detect surface deflections before the actual production in the finite element simulation. Then, it is much easier to modify the tool geometry in such a way that surface deflections don't occur any more.

In this paper, the reliability of the stoning method provided by LS-DYNA is investigated in a parameter study. The influence of the moving direction, the step size, the chosen area of the part for the stoning, the stone length, and the stone width on the detected surface deflections are analyzed. The results show that the chosen size of the area for the detection has a huge effect on the detected surface deflection.

Similarly, the stone length has an influence but the stone width does not. To validate the finite element simulation, the strain distribution of the numerical part is compared to the physical part.
The stoning method is applied to a physical part and the detected surface deflections are compared to those detected in the simulation.

The study is conducted on a typically challenging part: a curved sheet metal part with a door handle depression.

Advances in IGA for Sheet Metal Forming Applications

S. Hartmann (DYNAmore);
A. P. Nagy, D. J. Benson (LSTC)

In the last years, Isogeometric Analysis (IGA) has become very popular in the scientific research community. Various types of basis functions are investigated by the researchers, while the most widely used geometry representation in the Computer-Aided-Design (CAD) community is based on NURBS. It has been shown, that NURBS are particularly well suited for finite element analysis leading to qualitatively more accurate results in comparison with standard finite elements based on Lagrange polynomials.

This work will give a short overview about the general possibilities of IGA with NURBS in LS-DYNA and focus on the recent advances for the analysis of Sheet Metal Forming Applications. Therefore the benchmark example from the Numisheet 2005 conference on *"Forming of an Automotive Underbody Cross Member"* (BM2) [1], which has been analyzed by Hartmann et al. [2] in 2011 using one of the first IGA implementations in LS-DYNA, is reanalyzed to demonstrate the progress made since then.

Literature

[1] "NUMISHEET 2005: On the Cutting Edge of Technology – The Numisheet 2005 Benchmark Study – Part B", Cao, J., Shi, M.F., Stoughton, T.B., Wang, C.-T., Zhang, L. (eds.), August 15-19, 2005, Detroit, MI, USA

[2] Hartmann, S., Benson, D.J., Lorenz, D.: "About Isogeometric Analysis and the new NURBS-based Finite Elements in LS-DYNA", 8th European LS-DYNA Users Conference, May 23-24, 2011, Strasbourg, France

A Modular Material Modeling Strategy for UD Composites and Organic Sheets using MFGenYld+CrachFEM

M. Vogler, G. Oberhofer, H. Dell (MATFEM)

The very complex material and failure behavior of organic sheets and UD composites demands particular requirements on the material modeling strategy. Highly anisotropic material behavior, very ductile and very brittle behavior depending on the loading direction, and complex failure mechanisms and their interaction are just some of the challenges when modeling endless fiber reinforced polymers. In the first part of the presentation, already implemented extensions of the modular material model MFGenYld+CrachFEM for organic sheets and UD composites are presented and discussed. In the second part, an outlook to new modeling strategies is given.

The extensions of the modular material model MFGenYld+CrachFEM with respect to fiber reinforced composites follow the vision of strict modularity of the different features. The basic idea is to have a unified modeling platform and a unified material card independent from the respective solver environment, where different modules can be activated or deactivated depending on the current problem and on the current material to be modeled. This concept has been proven to be successful and widely accepted in industrial applications.

When modeling organic sheets and UD composites, both ductile and brittle failure can be observed depending on the loading directions and on the stress state (triaxiality). Hence, ductile and brittle failure are considered to be concurring failure mechanisms. This means that there is some kind of interplay between ductile and brittle failure. Ductile failure (inter-fiber failure in UD composites, off-axis loading in organic sheets) can be described using strain based criteria, whereby the fracture strain is a function of stress state and strain rate. This is already available in MFGenYld+CrachFEM when activating the Ductile Normal Fracture Model (DNF model, based on stress state parameter β) and Ductile Shear Fracture Model (DSF model, based on stress state parameter θ). In order to model brittle failure (fiber breakage, fiber kinking), stress based failure criteria have been implemented in MFGenYld+CrachFEM. The fracture stress can be a function of the strain rate and a filtering of the stress value is used to avoid early failure due to stress oscillations. Compared to unreinforced polymers, the yielding behavior itself is not very pronounced, but the post failure behavior becomes more and more important. This means that considerable deformation and energy absorption after the first crack is still possible due to successive fiber breakage, fiber matrix pullout and matrix yielding. This is especially important under bending load cases where successive failure of different layers is observed. In order to approximate this behavior in MFGenYld+CrachFEM successive failure of shell integration points over the thickness and a controlled softening after initiation of failure are implemented. That is, the failure risk is evaluated separately for each integration point over the shell thickness. Once failure is detected in one integration point, softening starts via controlled thinning of the respective shell element. With this approach, crack initiation and crack growth can be approximated under bending load cases.

In the second part of the presentation, new modeling strategies for endless fiber reinforced polymers are briefly presented. The first one developed by H. Dell is a superposition of 2 transversely isotropic models for a better description of the anisotropic behavior of organic sheets. The second one is based on a structural tensors approach acc. to the Ph.D. thesis of M. Vogler. With this general anisotropic elastic-viscoplastic model, an exact tracking of the fiber directions during the deformation process can be done. Furthermore, initial fiber missalignments (draping process, forming process) can be regarded easily if these information are provided via mapping. When modeling organic sheets, the evolution of anisotropy due to finite fiber rotations as it is observed in +/-45° load cases can be regarded. Next steps are the integration of further process parameters (press moulding, forming) like porosity, fiber directions and fiber density.

Holistic Approach for Simulation Driven Design Process for Fiber Reinforced Plastics

C. Hinse, S. Kaul (SimpaTec)

The future years have become more and more of importance for the automobile industry. The EU has, in the meantime, committed to reduce greenhouse gas emissions by a considerable percentage compared to 1990 levels. The fact that traffic generates a good portion of the total CO2 emissions creates a special dimension of responsibility for this area. Almost half of the emissions are attributable to passenger car traffic. This aspect presents a considerable challenge for the automotive industry. Possible solutions lie in the development and use of new, more efficient technologies and designs.

The new development provides several innovative enhancements, which can essentially improve the speed, robustness and reliability of simulation. These allow enterprises to get the most out of virtual simulation for injection molding and thus create core competiveness and add a substantial amount of value to their products.
The use of different materials as well as different strategies of molding can often lead to quality problems of a wide variety of different kinds. The attempt to recognize and avoid problems by means of 'Trial & Error' is a thing of the past. Today, developers and designers have a reliable and effective tool in form of 3D process simulation.

The combination of established processes is one of the most common methods to make a product more efficient. In general the target is to transfer a former multi-step process into a one-step process. This trend is also the reason to combine already established simulation tools with each other to simulate those "new" processes. Within this presentation a bi-directional interface between Moldex3D and LS-DYNA will be demonstrated to simulate the draping process by the compression molding process with a one-step process. By using this method it is secured that the interaction of those former tow processes which occur by the integration will be fully taken into account. The final result is the simulation of the process as well as the simulation of the deformation with the option to export the final material properties for further FEA.

The presentation gives an overview of the current development status by means of specific examples, and shows the possibilities of 3D simulation regarding the "new" one-step process.

Multi-Scale Modeling Technics Applied to a Multi-Material Design Context: SFRP, CFRP, Additive Manufacturing

S. Calmels (e-Xstream)

Short fiber reinforced plastics are already widely used in the automotive industry for various fields of applications, from under the hood to the interior or exterior components, such material choice is now commonly done for lightweighting purpose. Since the early 2000 years, the industry's interest is growing also for the usage of CFRP to extend metal replacement opportunities to more structural components. Nowadays, a 3rd category of material and manufacturing process offers new interests to this industry: the additive manufacturing. Hence, the choice has never been so large to reach lightweighting targets. However, all these materials brings with them a challenge when it comes to design component : all of them are showing mechanical behavior highly dependent on the manufacturing process which requires the usage of dedicated technics to be able to predict correctly the component's performance and design them in an optimal way.

Multi-phases materials in general are well known to provide heterogeneous, non linear and anisotropic mechanical behaviors through a given component. All these 3 aspects are driven by the local state of microstructure resulting from the various manufacturing processes required. Moreover, depending on the type of phases, short or long chopped fibers, continuous fibers, woven, the type of process, injection, compression, draping, FDM, each of them can show very specific behaviors regarding damping, failure, post failure, possible delamination. All these specificities requires to develop dedicated material models for each type of multi-phase material in order to be able to predict the component's behavior and then design it efficiently. Last of them, materials for additive manufacturing applications are highly interesting but also highly challenging to model. The process induces local material behaviors depending on the printing direction of course, but also on the resulting interlayer interface, the intra and interbeads porosity. A multi-scale material modelling technic is required to simulate accurately the effect of the additive manufacturing process on a such made component's behavior

This paper will address the application of e-Xstream multi-scale material modeling strategy to predict SFRP, CFRP and AM components. A special focus will be made on the latest development concerning the prediction of 3D printed components. This will demonstrate how simulation can be improved, for safety design simulations in particular, in the automotive industry, helping to reduce design delay, cost and weight of the structures.

Air Blast Reflection Ratios and Angle of Incidence

L. Schwer (SE&CS)

The Unified Facilities Criteria 3-340-02 provides blast wave reflection ratios as a function of angle-of-incidence as Figure 2-193. However no text is provided indicating the source of this information. A knowledgeable source provided that the figure combines experimental and analytic results. In an effort to assess the UFC information, comparisons are made two different sets of small scale experiments and LS-DYNA Eulerian simulation results. These comparisons are agree fairly well with UFC except in the region of the critical angle-of-incidence where there are large differences.

Of interest to LS-DYNA users, the comparison also includes the simple analytical expression for reflection ratios as a function of angle-of-incident proposed by Randers-Pehrson and Banister, and used in the LS-DYNA algorithm Load Blast Enhanced. It is shown that the Randers-Pehrson and Banister representation of reflection ratios is a lower bound on the above mentioned experiments and LS-DYNA simulations.

Comparison of MM-ALE and SPH Methods for Modeling Blast Wave Reflections of Flat and Shaped Surfaces

J. Trajkovski, R. Kunc, I. Prebil (University of Ljubljana)

The Multi-Material Arbitrary-Language-Euler (MM-ALE) and Smooth-Particle-Hydrodynamics (SPH) are widely used methods for numerical examination of structural response under blast loading. On the other hand, the methods are quite demanding for use. They have their own advantages and disadvantages depending on the structural geometry and its relative position with regard to the blast wave source location. This paper presents comparison results of detailed numerical examination using both the MM-ALE and SPH method. A series of simulation tests were performed using flat and shaped armor steel plates in LS-DYNA.

The comparison results showed that the SPH method offers better results and efficiency compared to the MM-ALE method. This is especially evident in cases involving curved structural geometry.

The results of this paper will be of great value in the decision making process when choosing an appropriate method for blast response analyses of structures.

Simulating Reinforced Concrete Beam-Column against Close-In Detonation using S-ALE

S. K. Tay, R. Chan, J. K. Poon (Ministry of Home Affairs)

A 3-stage loading on a reinforced concrete beam-column involving pre-load, blast and post-blast compression to failure was analyzed with the S-ALE solver. This paper presents the findings from the simulation and the results were compared to full-scale blast trials of reinforced concrete beam-column test specimens.

Processor Count Independent Results: Challenges and Progress

B. Wainscott, Z. Han (LSTC)

When a different number of cores is used to run the same model in MPPDYNA, different results are generally produced. We discuss a couple of different reasons for this, and why it is to be expected in a parallel program. Work is in progress to offer options which work around this issue, and results of some test problems are presented.

Maximizing Cluster Scalability for LS-DYNA

P. Lui, D. Cho, G. Lotto, G. Shainer (Mellanox Technologies)

High performance network interconnect is an integral component that enables all compute resources to work together. It is the key to scaling the LS-DYNA simulation in a cluster environment for both network compute and network storage to accelerate CAE simulations. The latest Mellanox InfiniBand adapters have introduced a novel high-performance and scalable architecture for high-performance clusters. This architecture was enhanced to provide higher performance and scalability for the largest supercomputers in the world, today and in the future.

In this paper, we demonstrate the new features and technologies that are driven by the latest InfiniBand adapter. Hardware capabilities featuring CPU offloads, MPI tag matching, MPI collective operations acceleration and message transport services make LS-DYNA perform at scale. In this study, we will review the novel architecture used in the HPCX MPI library and explore some of the features in HPC-X that can maximize LS-DYNA performance by exploiting the underlying InfiniBand hardware architecture.

The newly debuted Mellanox ConnectX-5 HCA, which supports up to 100Gb/s EDR InfiniBand will be analyzed as well. For comparison purposes, we will also contrast the performance and scalability advantages of EDR InfiniBand, which is based on an CPU offload architecture, over Intel Omni-Path interconnect, which is based on an onload architecture, on the LS-DYNA simulations.

On Demand Licensing with LS-DYNA

Prof. U. Göhner (DYNAmore)

The LS-DYNA simulation software is used on different High-Performance-Computing platforms since many years. Most of the users have access to in-house Cluster systems running LS-DYNA in the MPP-version on up to several 100s of Cores and an appropriate number of licenses to use their hardware effectively [1]. In recent years there was increasing demand inside the LS-DYNA user base for a flexible and efficient offering of software licenses both for on-premise hardware and for Cloud-based HPC-platforms. Since 2016 DYNAmore and LSTC offer a flexible and cost-efficient pay-per use licensing scheme. This licensing scheme is being implemented and can be accessed through LSTC and DYNAmore, who are responsible for the development and distribution of the LS-DYNA simulation software.

In this paper the different licensing schemes are being presented and practical experiences with the new LS-DYNA pay per use licensing concept are compared to other licensing possibilities. In addition the usage of LS-DYNA on Cloud-based infrastructure is shown. Advantages and disadvantages are discussed for different configurations and scenarios involving both SMEs and large organizations from automotive and other industries.

Tuesday Afternoon Sessions

17:30 – 18:45

Simulation of Short Fiber Reinforced Plastics with LS-DYNA Considering Anisotropy, Rate Dependency and Rupture

B. Lauterbach, M. Erzgraeber (Adam Opel);
C. Liebold, A. Haufe, M. Helbig (DYNAmore)

Stochastic Approach to Rupture Probability of Short Fiber Reinforced Polypropylene for Automotive Crash Applications

N. Sygusch, B. Lauterbach (Adam Opel);
N. Ruesch (Hochschule Darmstadt);
S. Kolling (THM Gießen); J. Schneider
(TU Darmstadt)

Due to the ongoing advancements for safety regulations parts made out of fiber reinforced polymers (FRP) gain more and more importance in the area of pedestrian and occupant safety. When studying the rupture behavior of FRP one can observe it’s stochastically distributed nature.

Furthermore, these rupture distributions are dependent of fiber orientation angle and strain rate. However, these effects are usually not covered in today’s simulations and simplified rupture criteria are chosen. In crash simulations, for example, rupture of structural parts is assumed to occur when a critical stress or strain threshold is exceeded.

Experiments have shown, that the state of rupture is more complex. Rupture behavior of FRP is not restrained to a particular value but is rather described mathematically as a distribution function. Utilizing a spectrum of rupture values instead of a fixed one by incorporating the stochastically distributed rupture behavior into the material model could improve rupture prediction in CAE.

Numerical-Experimental Correlation of Mechanical Tests on Fiber-Reinforced Polyamide Composites

A. Molaro, M. Lanzillo, F. Uimbardi, A. Causa, B. Villacci (SAPA)

This work (supported by the APPS4SAFETY financed project) presents an experimental and numerical study of the mechanical behavior of polyamide 66 (PA66) filled with short glass fibers (GF) and short carbon fibers (CF), which are appealing materials for the development of active automobile safety devices. As the latter need to be validated through numerical simulations of crash tests, the study described herein is aimed at the determination of the elements that define the LS-DYNA cards for the PA66-GF and PA66-CF composites.

Firstly, such fiber-reinforced materials have been thoroughly characterized by performing tensile tests on specimens cut from injection-molded panels at different orientations relative to the preferential fiber direction (0°, 45° and 90°).

Secondly, representative simulations of the experimental mechanical tests have been performed by developing in LS-DYNA two- and three-dimensional models of the specimens and subjecting them to quasi-static tensile loads. The material model used to describe the behavior of short fiber-reinforced thermoplastics is *MAT_NONLINEAR_ORTHOTROPIC (*MAT_040).

Finally, the simulated stress-strain curves have been calibrated with the experimental ones; namely, the parameters of the numerical curves have been optimized to obtain a good interpolation of the experimental results. For both the PA66-GF and PA66-CF composites, two-dimensional modeling has provided a better correlation between numerical and experimental data in comparison with the three-dimensional one.

Study of Occupant Lower Leg Injury Value Using Interface New Function

T. Ishihara, H. Sugaya, K. Maehara, H. Mae (Honda R&D)

Background

In recent years, development at CAE in automotive development is an indispensable technology to shorten the development period and reduce the prototype cost. The crash safety CAE has been used for development as well, but the occupant injury value (especially Lower Leg injury value) is predicted to be greatly affected by deformation of the vehicle frame in addition to modeling of Restraint device.

In addition, occupant dummy models contain a large number of soft tissues in order to raise the biofidelity, so there are cases in which the results differ greatly even with a slight difference in the amount of deformation. From the above, in order to predict the influence on lower Leg injury value, a tool is required to examine the influence of body deformation.

Objective

In order to examine the lower leg injury value due to the frame deformation difference, we developed a function to confirm the sensitivity of the deformation amount difference easily by scaling the deformation amount calculated by the frame CAE, in collaboration with LSTC and JSOL .

This function is named Interface_local and will be released in the future In this presentation, we report on the results of verification of the implemented functions and the effects of occupant lower Leg injury values due to differences in frame deformation.

THOR 5th Dummy FE Model Development

A. Lakshminarayana, C. Shah (Humanetics)

Alongside the hardware development of the THOR (Test Device for Human Occupant Restraint) 5th percentile female crash test dummy in cooperation with NHTSA, the development of the THOR 5th FE model is also well underway. The THOR 5th is an advanced small female crash test dummy and referred hereafter as THOR-5F. The main objective of the current paper is to describe detailed FE model development and biofidelity evaluation of the THOR-5F FE model. The paper also presents design support using the FE modeling for the THOR-5F ankle hardware design.

As with all FE models, geometry, material modeling and structural connectivity are of focus when developing the THOR-5F FE model. All the instrumentation and sensors are accurately realized in the model. Furthermore, it has first-hand access to the physical counterpart that allows the model to incorporate the latest hardware design features. The THOR-5F FE model is then evaluated against over 20 biofidelity tests from head to toe for functionality and performance evaluation. The current paper presents selected biofidelity test validations.

The THOR-5F FE model showed promising functionality and performance for the evaluated biofidelity validation cases. Concurrent development of the FE model enabled the hardware team a possibility to evaluate the concept design aimed to meet the biofidelity requirements, reduce design cycle and reduce production cost by performing the FE analysis prior to manufacturing hardware parts. This paper illustrates in particular the ankle design improvement in the THOR-5F hardware supported by the FE analysis. Such synergy between hardware and FE complements each other for the improved hardware design and better model predictability and performance.

The first FE model of the THOR-5F will be released in the 3rd quarter of 2017. This highly anticipated model will consist of a state-of-the-art mesh and appropriate connectivity, instrumentation, basic material models and preliminary validations to all available biofidelity tests to ensure the model functionality. The model will be validated rigorously against additional test load cases as they become available.

Physical Appearance Evaluation of Automotive Seat Structure with J-SEATdesigner

N. Ichinose, H. Yagi (JSOL)

Seat styling is one of the important factors for automotive interior design. Automotive OEM company specifies 3D design to the seat supplier or their seat design section and the seat supplier designs 2d pattern to achieve desired 3D design as possible. To design the pattern from 3D seat design require a lot of experience, in many company, limited number of specialist can design the pattern based on their experience. These companies strongly desire the simulation system to design the pattern from 3D seat design.

We had released J-SEATdesigner v2.0, in last June, which has basic features to set up seat manufacturing process like sewing/hooking/covering for LS-DYNA. v2.0 provides residual stress evaluation on seat fabric from sewing process simulation in LS-DYNA but the process simulation is not enough to evaluate physical appearance because of simplification of fabric 2d pattern. To evaluate physical appearance in LS-DYNA, the feature of J-SEATdesigner has been improved based on collaboration work with Japanese seat supplier.

In this paper, new features for advanced sewing set-up in J-SEATdesigner will be introduced and new quantitative evaluation of wrinkle on manufactured seat surface will be shown. As I mentioned above, manufactured seat surface is one of the important factors for interior design, this means that it is important how the user feel the wrinkle on seat surface. This feeling is highly depending on fabric texture and color which cannot be considered in sewing process simulation. To resolve this issue, the sensory evaluation based on computer graphics technology is implemented in J-SEATdesigner. This sensory evaluation will also be introduced.

Forming Simulations of Niobium Sheets – Upgrade of the Numerical Model and Outcome for Novel Productions

A. Amorim Carvalho, M Garlaschè, A. Dallocchio, O. Capatina,
L. Prever-Loiri, M. Narduzzi, J. Brachet, B. Bulat (CERN);
L. Peroni, M. Scapin (Politecnico di Torino)

The installation of superconducting Radio Frequency (RF) Crab Cavities is one of the key upgrades in the framework of the High Luminosity Large Hadron Collider (HL-LHC) at CERN. These devices – built out of niobium sheets – are shaped into a complex geometry entailing very tight tolerances, in order to comply with strict RF requirements.

Numerical simulations via LS-DYNA proved to be a useful tool to optimize the fabrication process of these RF cavities. Simulations are performed for predicting the capability of different processes and tools to yield shaped parts complying with geometry and surface requirements.

Having gained experience from the production of a first crab cavity design, the numerical model was enhanced with implementation of material models stemming from a comprehensive testing campaign of niobium rolled sheets.

The frame of upgraded simulations is currently contributing to the production campaign for the second crab cavity design, currently being carried out at CERN.

This contribution will present the procedures followed for the upgrade of the numerical model and present the obtained results.

New Features for Metal Forming in LS-DYNA

X. Zhu, L. Zhang (LSTC);
B. Hochholdinger (DYNAmore Swiss)

Forming of Ultra-High-Strength Sheet Metals with Alternating Blank Draw-In

R. Radonjic, M. Liewald (University of Stuttgart)

Reduction of the vehicle weight and improvement of the passenger safety are permanently defined requirements for design and manufacturing of the dedicated car body components. One possibility to fulfil before mentioned requirements is use of thin walled ultra-high-strength steel sheets for manufacturing of the car body structural parts.

However, when forming such kind of sheet metal materials, severe problems may result from the large amount of springback, which occurs after release of formed part. In order to reduce part shape deviations from nominal, forming of a hat channel shaped part geometry with the alternating blank draw-in was modelled and simulated in this study. In this investigation an ultra-high-strength steel of DP 980 grade was used. Performed simulations were calculated by using the FE-Code LS-DYNA. In order to detect advantages of this kind of forming process, conventional deep drawing of the same part geometry was simulated as well.

Simulation results showed that the part shape deviations after forming with the alternating blank draw-in were significantly reduced when comparing to part shape deviations occurring after conventional deep drawing with this symmetrical flange draw-in. Evaluation of simulation results before and after release of the part was carried out along three different cross sections to understand influence of complex stress state on springback occurrence of component. Finally, the successful process management which delivers negligible part shape deviations is presented in this paper.

Computational Material Models for TSCP Plastics Comparison of the Deformation Behavior with MAT 24 and MAT SAMP-1 with DIEM

M. Dobes, J. Navratil (Robert Bosch)

The subject of this article is comparison of the deformation and stress response for two computational material models suitable for POM material. Both material models use erosion of finite elements, but in the first case *MAT_24 is used together with MAT_ADD_EROSION and with parameters combination and in the second case *MAT_SAMP-1 is used together with DIEM (Damage Initiation and Evolution Model). The new computation material model SAMP-1 was built based on the original experimental data used for *MAT_24 and additional tests (3-point bending test, puncture test, etc.). The strain rate effect is considered based on the experiments in defined range and other strain rate values were calculated based on the analogue with Johnson-Cook constitutive material model. The attention is focused on tensile/pressure definition of computational material model in plasticity. This is very important effect for good prediction of the cracks with using damage material model DIEM.

The next characteristic is an accumulation of the damage which leads to rupture and finite elements deletion. This property is depending on the stress triaxiality. The results of deformation and stress response are very different for both approaches and SAMP-1 gives closer results to the experimental reality. The two types of the practical tasks are used for comparison of the stress and deformation behavior of these two computational material models. The first comparison checks deformation and stress response and second case tests difference between damage models. This work directly connects to previous articles for FSM (Fuel Supply Modules) introduced in preceding LS-DYNA conferences.

We find some non-physical behavior of the *MAT_SAMP-1 for specific configuration. So, this article is appeal for LS-DYNA users and developers to propose of solving of this non-physical behavior *MAT_SAMP-1 in specific situation, like as contact, compression loading area.

Typical Semi-Crystal Polymer

Failure Models of Plastics – Material Characterization for *MAT_ADD_EROSION (DIEM)

A. Fertschej, B. Hirschmann, M. Rollant, P. Reithofer (4a engineering)

The tensile test is a standard testing method for many different materials to determine elasticity, plasticity and failure. Due to DIC (digital image correlation) these tests are time consuming and in the case of dynamic testing also cost intensive. To characterize the dynamic deformation behavior dynamic bending tests on 4a impetus are a cost-efficient alternative.

In the last years further test methods for 4a impetus were developed to characterize failure. These test methods are easy and fast to perform and failure at different triaxialities can be specifically investigated. Also a high-speed-camera (newest accessory kit) can be implemented in 4a impetus. This allows the visualization of dynamic behavior of the material during test (crack initiation and propagation in detail) [1].

Using the aforementioned tests the generation of a material card including failure describing the material behavior near to reality can be performed. The material characterization is done by reverse engineering using the 4a impetus process. The material parameters are adapted iteratively until simulation and test fit with a minimum of deviation. Detailed information can be found in [3].
4 Failure modeling

LS-Dyna offers many material models for plastics that have an implemented damage/failure modeling. The most commonly used failure models are implemented in 4a impetus. This includes simple ("constant" plastic failure strain) up to highly complex models (plastic failure strain in dependency of strain rate and triaxiality) with access to different failure models (Johnson Cook, Xue-Wierzbicki, Mohr-Coloumb, etc). The significant inputs for the chosen failure model are accessible over the design variables. So an easy failure modeling and optimizing within 4a impetus software solution is possible.

Thermoplastics mostly have a ductile failure behavior (no failure under compression and shear), so failure criteria can be modeled especially for the triaxiality above 0.33. For failure lower than the triaxiality of 0.33 just assumptions can be made [4]. The Gurson and GISSMO model derived from metal models consider this fact also by assuming high plastic failure strain at negative triaxialities [5].

To model damage/failure the keyword *MAT_ADD_EROSION with the Damage Initiation and Evolution Model DIEM [7] was used and adapted to the test curves. A final validation of this damage/failure material was performed for a dynamic puncture test on 4a impetus.

Literature

[1] A. Fertschej, P. Reithofer, M. Rollant, "4a impetus (PART 2): innovations – test methods, MAT_SAMP-1, anisotropy, composites and more", 14th German LS-DYNA Conference Bamberg, 2016.
[2] P. Reithofer, M. Rollant, M. Fritz, A. Haufe, V. Effinger, "Validation and Material Modeling of Plastics", Europäisches Dynaforum Straßburg, 2011.
[3] https://www.carhs.de/en/companion-poster/product/caecompanion-pdf.html, pp. 50-52.
[4] H. Staack, D. Seibert, H. Baier, „Application-oriented Failure Modeling and Characterization for Polymers in Automotive Pedestrian Protection", Complas XIII, Barcelona, 2015.
[5] J. Effelsberg, A. Haufe, M. Feucht, F. Neukamm, P. DuBois, „On Parameter Identification for the GISSMO Damage Model", LS-Dyna-Forum, Ulm, 2012.
[6] A. Fertschej, P. Reithofer, M. Rollant, „Failure of Thermoplastics - Part 2 Material Modeling and Simulation", 10th European LS-DYNA Conference Würzburg, 2015.
[7] N.N., „LS-Dyna Keyword User's Manual – Volume II", r.5695 from 25. Sept. 2014.

Creep Modeling of Plastic Components in Sealed Connectors

H. E. Miled (Delphi Connecting Systems)

This work address the sealed connectors for engine compartment in particular high pin count housings and pin header. Silicon components, such as grommets and interfacial seals, are used and designed to ensure good contact pressure on plastic components and thus good sealing performance.

On several products, we noticed a deformation of plastic housings, in contact with seals, that leaded to a loss of sealing performance. This deformation is the sum of an instantaneous deformation, and a creep deformation. The instantaneous deformation results from elasto-plastic behavior during assembly and thermal change (e. g at 125°C) that leads to seals expansion and weaker plastic mechanical properties. Whereas the creep deformation results from the visco-plastic behavior and evolves with time under a constant loading and a constant temperature.

This paper describes and illustrates the numerical approach, used to identify the ratio of instantaneous and creep deformations on a given connectors. Simulation results on LS-DYNA, with sweetable models, are compared to experimental testing

A Review of S-ALE Solver for Blast Simulations

I. Kurtoglu, B. Balaban (FNSS Savunma Sistemleri)

Blast modeling and simulation is a very important field in the military land vehicle industry. Increasing demands for higher protection levels leads the engineers to more challenging design and simulation cases. In most situations, Arbitrary Lagrange Euler (ALE) method is the most well-known method for blast simulations and also for determining the effects of blast loads on structures. Various studies are performed for the effect of mesh size and the domain shape for traditional ALE solver of LS-DYNA. The newly implemented S-ALE solver is stated to give shorter simulation times and also less memory requirements using the advantage of structured mesh.

In this work, the S-ALE solver is compared to the traditional ALE solver for mine blast in steel pod. Different mesh sizes and advection methods are used for comparison. In addition to the displacement, momentum and deformation pattern, the solution times and memory requirements are also examined. Fluid-structure interaction (FSI) performance for solid interfaces is reviewed, as well.

A Comparison between Three Air Blast Simulation Techniques in LS-DYNA

H. Bento Rebelo, C. Cismasiu (Universidade NOVA de Lisboa)

When simulating structures subjected to the effects of blast loading, one might resort to three different methods of simulation. These methods are the empirical blast method, also known as Load Blast Enhanced (LBE), the Arbitrary Lagrangian Eulerian (ALE) method, and a coupling method that allows the application of empirical blast loads on air domain simulated with the ALE formulation.

Furthermore, for the ALE method, both a mapping technique, that allows the mapping of data from 2D ALE simulations to 2D and 3D ALE meshes, and a complete 3D ALE simulation could be performed. In order to verify and compare the efficiency and accuracy of these air blast methods, an air blast loading on a reinforced concrete slab is modelled. Additionally, mesh convergence studies of 2D and 3D ALE simulations are performed.

Secondary Shocks and Afterburning: Some Observations

L. Schwer (SE&CS); S. Rigby (University of Sheffield)

Air blast tests that included normally reflected pressure measurements by the University of Sheffield, provided evidence of the so called ‘secondary shock.’ This repeat test data provided an opportunity to explore the effect of the LS-DYNA afterburning model parameters on the time of arrival and magnitude of the secondary shock.

While the measured pressure histories alone are insufficient to uniquely calibrate the afterburn model, this manuscripts attempts to illustrated the effect on the secondary shock of changing the four afterburn model parameters:

1. Start time for adding energy
2. End time for adding energy
3. Amount of energy to be added
4. Rate at which the energy is added, i.e. either linearly increasing or constant.

Leveraging Rescale's Cloud HPC Simulation Platform to Run LS-DYNA Models and Accelerate Design Exploration: Examples and Case Studies

W. Dreyer, T. Smith (Rescale)

Reliance on simulation results is on the rise, resulting in larger model sizes, more runs, and increasingly sophisticated multidisciplinary simulation workflows. Meanwhile, chip manufacturers are coming up with increasingly specialized hardware (memory-intensive chips, SSD drives, low-latency networks, GPUs, specialty co-processors, etc.) that is being leveraged by LS-DYNA simulation software to deliver better performance. The days of the one-size-fits-all CPU are over. In the age of multidisciplinary simulations and highly-specialized compute hardware, the simulation environment must adapt by scaling up on new, powerful hardware and scaling out to larger compute clusters.

Luckily, the cloud is perfectly suited for that. On Rescale, engineers have instant access to LS-DYNA (customized version or out-of-the-box) and HPC hardware. They can also bring their own version of LS-DYNA, meaning they have the flexibility to customize their IT environment to solve any given engineering problem.

In this presentation, we will explore the benefits of simulation in a cloud-based environment like Rescale. Through concrete engineering use-cases and benchmark results, we will illustrate how Rescale can be used to run and interrogate LS-DYNA models efficiently and to share results without transferring large data sets. We will also highlight how Rescale's DOE framework can be leveraged to automate job submission for multiple model configuration and to easily compare alternatives.

HPC in the Cloud: Gompute Support for LS-DYNA Simulations

I. Fernandez (Gompute)

Gompute delivers comprehensive solutions for High Performance Computing, in-house, in the cloud or both. With over 10 years' of experience providing solutions and services to the Engineering communities, Gompute delivers a collaborative and productive infrastructure to either manage your HPC environment or burst out into the Gompute data center, ranging from 1 > 1000's of cores.

HPC in the Cloud – An Alternative to Cover "Just" Capacity Issues? Challenges & Outlook for Dynamic Scaling with LS-DYNA

A. Heine, J. Tamm (CPU 24/7)

HPC is accelerating the research and development efforts of large companies and academic organizations. A lot of engineers from smaller companies or smaller CAX departments are still working with limited compute resources (e.g. desktop PC or workstations), which are not scalable. The simulation driven product development in the SME world needs therefore alternative cloud based cluster solutions to be competitive.

The presentation will give you an insight about the typical HPC cloud requirements for CAE, the challenges and what needs to be considered before using cloud offerings for CAE applications. "Hot topics" like licensing models, security, processes and virtualized vs. bare metal servers will be covered.

A summary about the HPC cloud experiences from an operational point – roadblocks of deploying own dedicated physical clusters and complex software of HPC applications – will complete this session.

A Unified Environment for Processing Test Videos and Simulation Models

S. Kleidarias, V. Pavlidis (BETA CAE Systems)

Throughout the development process of a complex structure such as a vehicle, simulation and testing are both present in almost all phases. Being able to use them efficiently and concurrently whenever this is needed, it maximizes the benefit to the user, as this provides valuable information to identify defects in either the simulation or the test and therefore, avoid errors and delays in the development chain. One common type of test result is the videos, very common in crash domain. Enabling the processing of a video, which is a demanding task on its own, within the same environment where the simulation models are being processed, allows for the direct and easy comparison/correlation of the two and the deduction of useful conclusions.

This presentation showcases an LS-DYNA model and relevant real test videos both being processed in META, the post-processor of BETA CAE Systems. META forms a single unified environment for detailed model post-processing as well as for complete video processing. First of all the model view can be easily matched to that of the video either in one plane or in perspective view. Synchronisation of video frames and states of the model can be achieved in a simple way. Valuable information can be extracted from the video. Points as well as distances and angles can be tracked on the video and respective results can be readily plotted on the video as well as in a 2D plot. Tracking functionality for user defined points can also overcome cases where the tracked point is hidden for a number of frames and it also allows for manual tracking for certain frames so as to by-pass tracking discontinuity wherever automatic fix is not possible. Measurements on the video are facilitated by overlaying a grid as well as by defining user defined coordinate systems, either static ones or dynamic. The capability for eliminating rigid body motion from video frames, based on anchoring one or two tracked points on the video, can further assist the correlation of the video with a crash simulation model and thus lead to better conclusions.

Systems Engineering with Status.E and CAViT – Comparison and Assessment of CAT & CAE Data

G. Geißler, M. Liebscher, R. Hausdorf (SCALE);
M. van der Hove (AUDI)

d3VIEW – Data to Decision Platform Development Update

S. Bala (LSTC)

Workshop on Welding Simulation

T. Loose (DynaWeld)

In this workshop, two simulation models will be presented addressing the Gas Metal Arc Welding of a T-joint as well as a Laser Welding overlap joint with a tension test.

Herein, the most important features of a welding simulation are discussed:

- preparation of material data
- single- and multi-phase materials
- aluminum and steel
- weldpath and weld sequence
- heat source and heat input control
- heat input simulation with SimWeld
- contact, clamps and loads

The workshop is closed with a short demonstration of the welding preprocessor DynaWeld.

Wednesday Morning Sessions

8:30 – 10:10

Plastic Instability of Rate-Dependent Materials – A Theoretical Approach in Comparison to FE Analyses

C. Keller, U. Herbrich (Bundesanstalt für Materialforschung und –prüfung)

In FE simulations of dynamic events, the accuracy of numerical results strongly depends on the quality of the material models and parameters used. Material characteristics identified in static tests are not suitable or are of limited suitability for the description of the deformation behavior of components and structures under dynamic loading. On this account, it is necessary to determine material properties in dynamic tests and to provide reliable input data for numerical simulations, e.g. by means of rate-dependent material models or flow curves.

This can be achieved, for instance, by carrying out tensile tests with different loading rates, which, – statically or dynamically – are characterized by a material dependent plastic instability and necking of the specimens. In certain applications, the evaluation of the tensile tests in the range between initial plastic deformation and uniform elongation is sufficient. However, for the consideration of large deformation problems, it is essential to evaluate and establish material properties for the plastic deformation behavior beyond the uniform elongation.

Comparisons of results from experimental studies and FE simulations of dynamic tensile tests with ductile materials show significant differences in terms of the necking strain as well as the post-critical deformation behavior. In order to conduct research regarding the cause of determined disagreements, the present contribution displays a theoretical approach describing the instability in rate-dependent elastoplastic materials in comparison to results of numerical analyses. It is well known that under dynamic conditions, in addition to the dependence on the strain, the strain rate and inertia, both thermal softening and damage evolution in the material affect real deformation processes. In contrast, analytical and numerical approaches allow for a specific selection and separate evaluation of the afore-mentioned influencing factors.

This paper focuses on the plastic instability of rate-dependent, ductile materials. For this purpose, an analytical instability criterion under isothermal conditions is derived and applied. In conclusion, the theoretical findings are compared to simulation results from numerical studies and differences are discussed.

Short Introduction of a New Generalized Damage Model

T. Erhart, F. Andrade (DYNAmore);
P. Du Bois (Consultant)

In LS-DYNA, several constitutive models exhibit properties that are based on Continuum Damage Mechanics (CDM) [1]. Associated isotropic or anisotropic stress degradation represents softening and failure behavior of metals (e.g. *MAT_104), composites (e.g. *MAT_221), polymers (e.g. *MAT_187), and other material types. As an alternative, sophisticated (e.g. stress state dependent) damage models such as GISSMO [2, 3] or DIEM [4] can be used as add-ons to a wide range of standard material models via *MAT_ADD_EROSION.

With the release of LS-DYNA version R9, a new keyword *MAT_ADD_GENERALIZED_DAMAGE (MAGD) was added as a further step towards an even more versatile tool in this area. The primary idea was to provide non-isotropic (tensor type) damage with multiple independent damage variables. Users can define the entries of the damage tensor by functional input (*DEFINE_FUNCTION) to achieve maximum flexibility. The evolution of damage variables is driven by strain based values that can be determined in a number of ways. They can either be arbitrary history variables of the accompanying material model, or quantities derived from the plastic strain tensor. Therefore transformations of that tensor to the principal strain system or to the local system of material directions can be selected. Besides the possibility of using individual plastic strain tensor components as damage driving quantities, functional combinations of them are also allowed through the keyword *DEFINE_FUNCTION. For the damage evolution laws itself, the GISSMO approach can be used where corresponding input variables (such as stress state dependent failure strain, regularization, etc.) can be defined separately for each damage parameter. As a matter of fact, MAGD is a very flexible tool to incorporate non-isotropic damage into existing material models. The first successful application was done for aluminum extrusions, which show marked directional dependencies of failure strain [5].

This presentation will describe the underlying theory of MAGD, explain the various input options, and demonstrate its main functionalities using simple numerical examples.

Literature

[1] Lemaitre J., Desmorat R., "Engineering Damage Mechanics", Springer, 2010.
[2] Neukamm F., Feucht M., Haufe A., Roll K.: "On closing the constitutive gap between forming and crash simulation", 10th International LS-DYNA Users Conference, Detroit, 2008.
[3] Andrade F.X.C, Feucht, M., Haufe A., Neukamm F.: "An incremental stress state dependent damage model for ductile failure prediction", International Journal of Fracture, 200:127, 2016.
[4] Borrvall T., et al.: "A General Damage Initiation and Evolution Model (DIEM) in LS-DYNA", 9th European LS-DYNA Conference, Manchester, 2013.
[5] Du Bois P., et al.: "A new versatile tool for simulation of failure in LS-DYNA and the application to aluminium extrusions", 14th International LS-DYNA Conference, Detroit, 2016.

Ductile Failure in Large-Scale Analyses of Aluminum Structures

D. Morin, T. Berstad, O.S. Hopperstad, M. Langseth (NTNU)

Modelling of ductile failure in aluminum alloys is of high importance during the design of lightweight structures subjected to impact loading. Such designs are today based on numerical analyses and their credibility is crucial to reduce the development time and costs of such products. Under impact conditions, lightweight structures are most likely exposed to severe loadings and failure is a phenomenon which cannot be ignored. To accurately capture ductile failure, an analyst should employ advanced constitutive models to describe properly the local behavior of the materials as well as proper discretization to predict the correct deformation mode of the structure. This approach would lead to the use of advanced anisotropic yield surfaces to capture accurately the anisotropic yielding and plastic flow of aluminum alloys as well as fine solid element meshes to predict correctly the strain localization process [1]. Unfortunately, these approaches require large investments from the user both in terms of calibration of the constitutive models and CPU time when running the numerical analyses. These large costs prevent the use of these techniques into large-scale analyses and simplified approaches are still required within an engineering environment.

The aim of this work is then to propose a constitutive and failure criterion suitable within an engineering context both in terms of calibration cost and applicability for shell element analyses. The yielding and plastic flow of the investigated aluminum alloy is then reduced to an isotropic non-quadratic model using *MAT_36 and ductile failure is predicted using the Cockcroft-Latham failure criterion added to the model through the *MAT_ADD_EROSION keyword. A simple calibration procedure based on a single tension test is applied to calibrate the constitutive model and the failure criterion. Based on the tensile test results, a simple but cost-effective regularization scheme is proposed to handle the mesh sensitivity of the failure model for shell elements of various sizes.

The calibration and validation of the proposed approach is presented based on quasi-static and low-speed impact tests carried out on large aluminum stiffened panels recently published by Morin et al. [2]. A satisfactory agreement is found between the proposed modelling approach and the experiments.

Literature

[1] D. Morin, O.S Hopperstad, O-G Lademo, M. Langseth, Multiscale Modelling of Aluminium Components for Crash Loadings, 10th European LSDYNA conference, Manchester, UK, 2013

[2] D. Morin, B. L. Kaarstad, B. Skajaa, O. S. Hopperstad, M. Langseth, Testing and modelling of stiffened aluminium panels subjected to quasi-static and low-velocity impact loading, accepted for publication in International Journal of Impact Engineering.

Characterization and Modeling of Anisotropic Behavior of Aluminum Profile

F. Andrieux, D. Sun (Fraunhofer IWM)

Weight reduction is an important step to limit energy consumption. As a consequence extruded aluminum profiles are increasingly used for lightweight vehicle construction. Since fracture strains of aluminum profiles are relatively low, damage modeling is crucial for reliable crash simulation.

For aluminum profiles not only the stress state but also the orientation has an influence on both the deformation and damage behavior and needs to be characterized. For this purpose smooth tensile, notched tensile, shear-tension and punch specimens were extracted from a profile of EN AW 6106 T7 in three directions and tested. For instance it is shown that the force vs. displacement curves obtained from shear tests with specimen in the diagonal direction strongly differ from those with specimens in longitudinal or transverse direction, although tensile tests deliver very similar behavior for the three directions. Digital image correlation (DIC) analyses were performed to determine local strain values such as critical values at failure. The local values from DIC analysis and model predictions were compared to evaluate the accuracy of the deformation model.

The numerical investigations were conducted with LS-DYNA FEM software, using shell elements. The strain based damage model GISSMO, which is well suited and widely applied for crash simulation, was retained. For materials with complex deformation behavior e.g. anisotropy it is still open, which available material model is suitable for the calculation of local strains which directly rule a strain based failure model. In this work deformation models with increasing complexity were investigated. The isotropic von Mises model and two anisotropic material models according to Barlat were compared. For the first anisotropic model, Barlat 3-parameter, two variants were used. In addition to the original model which needs the specification of three parameters, an extended variant (option HR=7), which requires the definition of three flow curves in three directions was investigated.

The best compromise over all tests was found with the second investigated anisotropic model, YLD2000, which requires the definition of eight parameters. The extended Barlat 3-parameter-model can predict the orientation dependence of flow stress under uniaxial tension over the whole deformation range very well, but this good accuracy cannot be transferred to all stress states. For instance it does not well predict the anisotropic behavior under shear loading. There is still a great requirement on development of a material model which describes the orientation dependence of the hardening behavior over all stress states.

The anisotropic model YLD2000 and the isotropic model were used to simulate the crushing and bending of profiles. For these loading cases both models led to similar numerical results.

Lightweighting and Cost Reduction Using Optimization-Led Design Software

A. Farahani, M. Kiani, D. Mittal (ETA);
A. Kaloudis (BETA CAE)

Lightweighting is a major focus currently in the product design and development of structures, such as ground transportation vehicles (passenger cars, trucks and buses) and other consumer products. Lightweighting is a multipurpose economical solution for many industries including environmental, construction, and transportation. Product cost reduction and reduction of Co2 (34.1 MPG 2016 to 54.5 MPG by 2025), reduction in the shipping costs and a reduction in transportation costs are just a few examples of how lightweighting affects product design and development.

Lightweighting, for OEMs and suppliers, is an uphill battle, in which they must design and produce products that require a balance between a variety of competing factors such as cost, mass, multidisciplinary performance, multimaterial function, joining and manufacturability. Furthermore, they must achieve this balance while meeting high government and consumer standards.

Multimaterial products, for lightweighting, are the future of product design and devlopment for Electric Vehicles (EV) and Battery Electric Vehicles (BEV). One of the most difficult tasks for manufacturers is the proper use of of materials, the choice of advanced materials (Advanced High Strength Steel, Aluminum and Carbon Fiber Plastic), placed in the proper location, with the optimal geometry, optimal grades and optimal gauges. To assist OEM's and suppliers in addressing these challenges, in 2015 ETA and BETA CAE joined forces to create a software product called "ACP OpDesign". It offers a systematic process for product design and development and lightweighting, based on ETA's patented ACP Process technology.

ACP OpDesign is an optimization-led design tool, used from the concept level to the manufacturing stage based on the Accelerated Concept to Product (ACP) Process®. The process orchestrates and drives the maturity of a product from the concept phase, to its near production stage for manufacturing. Allowing multidisciplinary loading (linear and nonlinear), it searches for the optimal material types and best geometry, grade and gauges. ACP OpDesign is designed based on ANSA and META (developed by BETA CAE Systems) and acts as a gateway to drive the design using LSTC multi-function solvers of LS-TaSc (Topology Optimization) LS-Opt (Parametric Optimization) and LS-DYNA (Solver) for the multidisciplinary solution.

The presentation will give an overview of the FutureSteelVehicle body in white (Developed by WorldAutoSteel) for a battery electric vehicle, resulting in a 39% mass reduction. The overview of the FSV design from the concept stage (packaging and styling) to the Low Fidelity Design Concept (LFDC) will be presented using ACP OpDesign. New technology will be explored to show how it enhances and simplifies the process. The technology reduces the resource requirements and reduces product development time to market, by leveraging the many processing tools built within its software platform ANSA.

Using this systematic method, ACP OpDesign creates a balance between structure and strength, synchronizing the individual facets of the product development process. The advantages include design efficiency, shorter product design and engineering time, lower project costs and provides a unique database for product design and development Life Cycle Management.

MDO Collision/NV/Stiffness Optimization with LS-OPT

R. Ishii, M. Takeda (JSOL);
Y. Tanaka (Toyota Auto Body);
M. Nishi (Nihon Emsco)

LS-DYNA is heavily used to analysis transient phenomenon like car crash and makes a huge achievement about physical simulation in a wide variety of industry. For the goal of LS-DYNA, one-model, one-code as solution, it give you a wide variety of function has been developed at each section.

Nowadays, LS-DYNA has been developed further and become possible to evaluate Frequency domain analysis and Acoustic analysis as FRF/SSD/AcousticBEM/FEM etc.

This paper is intended for MDO (Multidisciplinary Design Optimization) with LS-DYNA and LS-OPT. The object is automotive which has many complicated parts. It is so hard to meet the demand for couple of standard for the safety/NV/strength. LS-DYNA can calculate for not only the crash but strength and NV (noise, vibration) evaluation.

The MDO evaluating some linear analyses simultaneously is the common case, but optimization with combination of both linear and non-linear analysis like car crash would be not so common case.
It would be possible for LS-DYNA and LS-OPT to consider this case. So, the purpose of this paper is challenge to this case, which mean the confirmation to benefit and effect to car design process.
When MDO with collision consideration is regarded useful when car design and estimation of performance, the usage of this type MDO will become widely used.

Efficient Global Optimization using LS-OPT and its Parallelization

A. Basudhar, N. Stander, I. Gandikota (LSTC);
K. Witowski, A. Svedin (DYNAmore Nordic)

Simulation-based design has evolved significantly in the past few decades. While the computing resources have advanced, model complexity has also increased considerably to capture the physics in greater detail. Therefore, reducing the number of “expensive” samples needed to obtain an optimal design is very important. Also, parallel computing has now become widely available, making the simultaneous selection of multiple samples desirable for any optimization algorithm.

Efficient global optimization (EGO) has become a very popular method since the late 1990s. In this work, a constrained EGO implementation in LS-OPT v6.0 will be presented that can be used to select samples either in serial. One of the limitations of EGO, in its original form, is that it selects one sample per iteration and is unsuitable for parallelization. Different approaches to overcome this issue have been proposed, such as the Kriging-Believer, Constant Lier, as well as Pareto-based methods. However, there is scant literature pertaining to parallel sampling for constrained EGO, especially in the context of reliability-based design optimization (RBDO).

The current LS-OPT implementation of EGO parallelizes the algorithm by selecting one sample per iteration using expected improvement (EI) maximization (basic idea of EGO), while the rest are selected as space filling samples. Evaluation of additional space filling samples helps in reducing the variance of the Kriging prediction globally, thereby updating the improvement function for the next iteration.

In addition to the current LS-OPT v6.0 implementation, this work will also present a new Pareto-based parallel constrained EGO method for deterministic optimization and for RBDO that aims to increase the efficiency of EGO as well as to widen its scope of application, e.g. RBDO. A guidance function based on proximity to the limit state will be used in the case of RBDO. A classification method will provide the total probability of feasibility ($P_{feasible}$) of all constraints. $P_{feasible}$ will be used as one additional objective (irrespective of the number of constraints) to define a multi-objective problem for locating the samples. One of the infill criteria in sequential constrained EGO is to maximize the product of EI and $P_{feasible}$. However, either of the two terms may dominate the other. The proposed approach will mitigate this issue by considering the Pareto front with trade-off between the two criteria. Also, instead of treating EI as an objective, it will be decomposed in two functions (prediction mean and variance), as EI may not always balance exploration and exploitation. Examples validating the proposed method will be presented.

Numerical Simulation of High-Speed Joining

M. Gerkens, Prof. G. Meschut (University of Paderborn)

The increasing trend of multi-material and space-frame design in automotive car body construction consolidates the need for mechanical joining technologies with one-sided accessibility. The high-speed joining (also called “RIVTAC®” or “Impact”) is an innovative and flexible technology. A tack with a profiled shank and an ogival shaped tip is pushed into the joining partners with a speed of 20 to 40 m/s without pre-punching. The plastic and friction work is converted into heat which causes an abrupt rise of temperature in the joining zone. This improves the flowability of the material which leads to filling the annular grooves on the shank of the tack. A high form fit is achieved. Simultaneously, a non-positive connection due to pressing and compression of the material is obtained, especially for high-strength steels.

Furthermore, effects of inertia are used, so that thin-walled structures can be joined without any additional tools (e.g. a die). But the local stiffness is an important aspect. The axial force during the joining process and the radial strains effects a deformation of the joined structure. The numerical simulation is a powerful tool to predict the deformation and the joinability of a complex structure as well as the mechanical properties of the connection.

The lecture is about detailed process- and loading-simulation of high-speed joining in LS-DYNA. Experimental and simulative investigations reveal the necessity of a correct material description.

The simulations are using the elasto-viscoplastic material-model by Johnson-Cook (MAT224), whereby plastic heating is causing adiabatic temperature increase and material softening. It is possible to define flow curves depending on plastic strain rate and temperature by a multiplicative approach. Furthermore, the friction parameters are very important for the physical consideration of the non-positive connection. Static and dynamic coefficients of friction (parameter FS and FD) have to be defined as well as coefficients for viscous friction (parameter VC), which are used to limit the friction force to a maximum.

The coupled structural and thermal simulations are done in an explicit and implicit code as axisymmetric 2D-model.

Continuous Simulations from Resistance Spot Welding Process to Joint Strength

S. Yagishita, T. Kawashima, N. Ma (JSOL)

Resistance Spot Welding (RSW) is a low cost and efficient method compared with rivet and adhesive. RSW is also easily automatized and can make robust joints. Therefore, it has been widely used in assembling vehicle bodies of thin metal plates.

These days, the use of Ultra High Strength steel and Aluminum plates increases in vehicle structures, so there is necessity to study stable welding conditions and predict the welding effect, especially the effect of Heat Affected Zone (HAZ) on these materials.

Despite of the simple principle of the RSW process, the complicated physical phenomena occur sequentially. Plates are pressed with spot welding electrodes and melted by the Joule heat generation of the electric current resistance. Finally material property gradually changes due to phase transformation depending on their temperature history, especially on cooling speed.

The temperature distribution over than melting temperature or the phase transformation temperature can be used to evaluate the nugget size and phase transformation zone. So it is important to take into account these sequential and multiple physical results in order to evaluate joint stiffness and strength.

With the aid of enhanced multi-physics features of LS-DYNA, we performed the continuous simulations from resistance spot welding process to the joint strength considering the effects of phase transformation occurred during cooling.

Resistive Spot Welding Simulations Using LS-DYNA

P. L'Eplattenier, I. Çaldichoury (LSTC);
T. Loose (DynaWeld)
U. Reisgen (RWTH Aachen)

Resistance Spot Welding (RSW) is a very important welding process for thin sheet metals with many applications, in particular in the automotive industry. In this method, the contacting metal surfaces are joined by the heat obtained by Joule heating of an electrical current flowing through resistances. These resistances are composed of the bulk resistance of the parts being welded, and of the contact resistances at the interfaces between the electrodes and the sheets, and between the sheets.

The amount of Joule heating energy delivered to the spot is determined by the magnitude and duration of the current and the values of the resistances. The resistances usually depend on the geometry, material properties and temperature: the bulk resistance of the metals is temperature dependent, and the contact resistances depend on the local contact pressures and temperature. The electrical conductivities of the materials and the yield stress are also used in some models to predict the contact resistance.

The current flow then depends on the local values of the bulk and contact resistances. It is thus very important to have a model which captures all these phenomena.

Recently, the contact resistance model in the EM solver of LS-DYNA has been extended to allow RSW simulations, where the user can define a local contact resistance as a function of different local contact parameters, using a *DEFINE_FUNCTION. The EM solver has also been adapted so that the current flow takes into account the local contact resistances. The local Joule heating is added to the thermal solver, and temperature dependent electrical conductivities can be taken into account via EM EOS's.

The model will be presented, then we will show how to set up a RSW case, and will present some results and benchmarks with experiments.

Modeling of Curing Adhesives between Jointed Steel and Aluminum Plates using *MAT_277 in LS-DYNA

S. Dong (Ohio State University);
A. Smith, A. Sheldon (Honda R&D Americas)

In this paper, a sandwich structure of two jointed plates with an adhesive layer in between is simulated. A top plate of aluminum, and bottom plate of steel are jointed with rivets. Then the plates go through the oven for paint baking, which lasts for approximately 60 minutes with oven temperature increasing in the beginning, then keeping constant, and decreasing towards the end.

The difference in thermal properties between the aluminum and steel causes different expansion rates between the top and bottom plates, which leads to plastic deformation at, and around the mechanical fasteners. A layer of adhesives is therefore employed between the two plates, aiming to increase the bonding and also to reduce the plastic deformation. However, both the chemical and mechanical properties of the adhesives change nonlinearly with temperature. Therefore, it is critical to model the properties of the adhesives properly so that the deformation of the sandwich structure can be predicted, and the stress distribution can be analyzed. A new material card in LS-DYNA, Mat_277 is used in this project for the adhesives.

This material card relates the chemical and mechanical properties of the adhesives not to temperature, but to an intermediate parameter named degree of cure. The degree of cure is decided by both the time and temperature of the baking process. The entire process is simulated using the LS-DYNA implicit solver. Instead of imposing the temperature at all nodes, a thermal-structure coupled analysis is conducted by way of using the *BOUNDARY_CONVECTION card. This allows the different thermal conductivities of aluminum and steel to be taken into consideration.

It was found that the adhesive strength prior to full cure is critical in preventing the plastic deformation at the joints. With certain levels of pre-cure strength, the adhesive layer can increase the bonding between the two plates without delaminating, which in turn could allow less mechanical fasteners to be employed. The relationship between the properties of the adhesives and the number of mechanical fasteners needed is therefore studied. Simulation results are validated qualitatively with experimental data and good correlation is found.

Numerical Evaluation of Low-Speed Impact Behavior of a Fabric Layered Composite Plate in an Industrial Context

S. Treutenaere, F. Lauro, B. Bennani, G. Haugou, W. Xu (University of Valenciennes);
E. Mottola, T. Matsumoto (Toyota Motor)

The use of Layered Fabric Reinforced Polymers (LFabRP) in the automotive industry is growing significantly. In order to ensure the safety and design imperatives, a new material model was developed for the LFabRP to improve the predictability of low-speed impact simulations by means of Finite Element Analysis (FEA).

The delamination prediction with FEA requires costly computation time methods such as the use of cohesive elements at ply interfaces. The proposed method includes the delamination evaluation directly within the material model and operates as a plugin for the constitutive intralaminar model. It allows to describe in an accurate manner the behavior of a layered material by using only one shell element through-the-thickness.

The material model recomputes a realistic strain field by means of a high-order zigzag theory. It takes into account the delamination effects on the continuity of the strain field but remains based on five degrees of freedom. By ensuring the internal energy equivalence between both element and material model theories, a realistic strain field for layered material is provided to the intralaminar material model.

The intralaminar material model is based on a pre-existing continuum damage model (Onera Damage Model). To improve the efficiency and the precision for the modelling of LFabRP, friction mechanisms, a rheological viscoelastic model and a smeared crack approach for the fibre failure were introduced.

The validation of the present model was carried out by means of controlled impact tests on a hydraulic high-speed jack. The LFabRP taken as reference is made up with three different fabric preforms. The material parameters are exclusively determined thanks to standard tests on the preforms taken individually on order to evaluate the model ability to predict low-speed impact behavior. Moreover, experimental 3D damage reconstruction by means of ultrasonic inspection is compared to simulation predictions.

Recent Developments and Application of *MAT_REINFORCED_THERMOPLASTIC

T. Klöppel (DYNAmore)

Material Characterization of a 3D-Woven Carbon Fiber Preform at Macro-Scale Level for Manufacturing Process Modelling

G. Scarlat, R. Ramgulam, P. Martinsson, H. Bayraktar (Albany Engineered Composites)

The latest generation of CFM LEAP aero engines that power the Boeing 737-MAX and Airbus A320neo have their fan blades and fan-cases made out of 3D woven carbon fiber and epoxy composites. These composites are manufactured using preforms that are 3D woven and undergo a complex forming process during manufacturing prior to being injected with resin during the RTM process. The present paper showcases the work done at Albany Engineered Composites (AEC) to characterize and model the mechanical behavior of the 3D woven carbon-fiber preform at the macroscopic level with the purpose of simulating manufacturing processes.

A variety of factors during such forming process can have a significant effect on the quality of the preform as it arrives at the RTM injection stage. A “good” preform quality usually means the absence of any wrinkles, high-shear areas and other similar local flaws at the end of the forming process, which in turn will have an effect on the performance of the molded part. The use of FE simulation tools can help predict such local defects, and allow (relatively cheap, in the virtual world realm) the exploration of some different combination of system and process parameters which will avoid such issues.

The first task was to select the most appropriate material model that can adequately represent the 3D woven carbon fiber preform at the macro-scale level, in an FE model using shell elements. This approach was decided due to the fact that it is computationally impractical to model each tow at the meso-scale level due to the overall dimensions of the parts that are typically manufactured. Since there is no “universal” material model developed for 3D woven textiles, several built-in dry-fabric material models available in LS-DYNA were tested.

The calibration of the material model parameters was done based on the fidelity of the model in matching the experimentally measured response of 3D woven preform coupons. For this purpose, an extensive set of coupon tests were conducted for in-plane behavior characterization. The main types of loading that the preform is subjected to during the forming operations are uni-directional tensile and shear. The draping behavior of material models considered is another important criteria and was evaluated as well.

The current paper will detail all the above steps undertaken within the material characterization workflow and will present comparisons of LS-DYNA simulation against experimental results for representative tests of the 3D woven preform coupons.

Numerical Investigations of Adhesive CFRP-Joints and Determination of Transverse Properties of the Adherends

T. Behling, M. Holzapfel (DLR)

Within the DLR (German Aerospace Center) project "Next Generation Car" adhesive joints of carbon fiber reinforced polymers (CFRP) are investigated. The focus was set on a numerical model to predict the failure mode (surface ply failure, delamination or adhesive failure) depending on the stacking sequence of the laminate. In a first step, the 'five-point bending' test was evaluated and chosen to measure the out-of-plane shear strength of a woven fabric and unidirectional CFRPs with various fiber angles.

The results from the Digital Image Correlation (DIC) were compared to analytical and numerical models. An evaluation routine was derived to assess the out-of-plane shear properties of the CFRP. Finally, fine discretized numerical models of single-lap joint (SLJ) specimens were discussed and compared to tests.

2D Modeling of Blast Induced Rock Damage around a Single Blasthole

A. Saadatmand Hashemi, T. Katsabanis (Queen‘s University)

Rock fragmentation by blasting is one of the most challenging activities in mining industry. The selection of proper explosive material for site specific conditions, correct size of burden and spacing of the blastholes along with an appropriate delay time of detonations have always been of concern for mining engineers in order to get the desired results from blasting. The precision required for pit wall damage control in order to mine with steeper angles in open pit mining activities or to control the blast induced damage around the underground excavation boundaries are the most important aspects which require comprehensive understanding of blasting phenomena and its impact to the surrounding rock medium.

In this paper a 2D model representing a cross section of a blast has been developed using the LS-DYNA finite element code. The failure of the rock as well as the damage sustained as a result of the detonation of one hole or a series of holes, detonated sequential, has been analysed. Coupling of charge to the borehole, air, timing and geometry of the blast are discussed in view of practical experience.

Numerical Modeling of Concrete Response to High Strain Rate Loadings

R. Sharath, D. Arumugam, B. Dhanasekaran, T. Subash (Larsen & Toubro)

The dynamic characterization of concrete is fundamental to understand the material behavior in case of earthquakes and extreme dynamic events like Blast and impact. Extensive research is available on the study of quasi-static or nearly static behavior of concrete, but limited investigations/research exists on the prediction of dynamic response, especially under high strain loadings. Numerous material models are available for modeling the dynamic behavior of concrete, this research focusses mainly on numerical simulations of the quasi static and dynamic behavior of the concrete including the strain rate effects.

For this research, popular material models MAT072R3 (KCC), MAT084 (Winfrith), MAT 272 (RHT) and MAT159 (CSCM) were implemented, that are available in the explicit dynamic software LS-DYNA. Single element tests verification subjected to varying strain rates in tension as well as in compression were the starting point of validation/comparison of different material models. The single element tests on different strain rates confirms the experimental behavior. Followed by, to study the quasi static behavior of concrete, numerical simulations of cylinder test (tension/compression) and three-point loading tests were carried out. Quasi static response of the concrete for the different material models were analyzed and the best suitable material model for the scenario is suggested.

Backed up with the above numerical analysis results, reinforced and unreinforced concrete slabs subjected to blast loadings which usually involves high strain rates were analyzed. The damage behavior, crater formation and dynamic response of the slabs were compared with data extracted from the reference papers and relevant codes. Finally, the analysis results using different material models were summarized and discussed.

Numerical Prediction of the Dynamic Response of Prestressed and Reinforced Concrete Hollow Core Slabs Under Blast Loading

A. Maazoun, S. Matthys (Ghent University);
J. Vantomme (Royal Military Academy)

This paper investigates the use and accuracy of finite element simulations by means of LS-DYNA of reinforced concrete hollow core slabs with a compression layer, simply supported and subjected to blast loading. The aim of this paper is to explain how to develop a numerical model, in order to predict the maximum deflection and crack distribution. Parametric studies related to hourglass energy, erosion value and the material model of the concrete are performed in order to evaluate the effects on the dynamic response of the hollow core slabs. This numerical analysis is completed by experimental tests for validation purposes.

Simulating Dynamic Loads on Concrete Components using the MM-ALE (Eulerian) Solver

S. K. Tay, R. Chan, J. K. Poon (Ministry of Home Affairs)

Concrete components can either be modeled as Lagrangian or MM-ALE solids. This paper provides an overview of the various concrete material models available in LS-DYNA for the use with the MM-ALE solver under the simple load case of uniaxial compression. The manuscript concludes with a case study of the behavior of concrete panels subjected to air blast.

Simulation of Fluid-Structure Interaction between Injection Medium and Balloon Catheter using ICFD

L. Wiesent, Prof. M. Wagner (OTH Regensburg)

Arteriosclerosis is a major health issue worldwide. While it is commonly treated by the implantation of an balloon-expandable stent, micro injuries may occur during stent deployment, and induce in-stent restenosis, whose consequence can be fatal. Studying this undesirable phenomenon is usually limited as experimental data is hard to obtain on ethical ground. Numerical simulation are performed to better understand this problem.

To construct a more realistic simulation of a balloon-expandable stent, a partitioned strongly-coupled FSI simulation of the balloon deployment was set up using the ICFD solver of LS-DYNA, - a quite innovative approach. The complex balloon configuration as well as the interaction of the injection medium and the balloon structure was considered. The balloon structure consisting of shell elements was obtained from preliminary balloon folding and pleating simulations. The balloon consists of a flexible thin walled polyamide. The injection fluid is implemented using volume elements. Balloon deployment was initiated by a pressure boundary condition inducing a volume flow into the balloon.

The initial feasibility analysis showed promising result including a continuous balloon deployment and a reasonable development of the fluid pressure and velocity field. However, applying this FSI approach to a more complex balloon structure led to a non-convergent solution. The non-convergence could be mainly reduced to mechanical factors including the low wall thickness of the balloon (< 0.05 mm) and the flexibility of the polyamide. Further, the ICFD solver shows less accuracy concerning the FSI conditions when dealing with thin flexible structures as well as enclosed volumes. A shell thickness of 0.06 mm is believed to result in a convergent solution. Based on these findings, a more detailed examination of the convergence issues and their possible solutions can be explored as future works. The work presented in this thesis is believed to innovative, and provides a promising approach to a realistic FSI simulation of a balloon-expandable stent.

Generalized Porous Media Flow in ICFD-LS-DYNA: FSI, Free-Surface, RTM and Parachute Modeling

R. Paz, F. Del Pin, I. Çaldichoury (LSTC);
H. Castro (Conicet)

In recent years industries like aerospace, automotive and those related to oil production have increased their trustfulness on numerical models and codes for the design, research, production and verification of highly critical parts and production processes. Most of these industries have adopted manufacturing procedures involving composites materials in liquid state, like the Liquid Composite Molding (LCM) and the High Pressure Resin Transfer Molding (HPRTM) methods, where a Newtonian (or Non-Newtonian) fluid flows through highly anistropic matrices filling an initially empty container. Hydraulic fracture (or Fracking), where the rock containing the oil is fractured by a hydraulically pressurized liquid, is another example of problems where a free surface flow develops over a porous material.

In this article the numerical modeling of the free-surface fluid flow through general anistropic porous media is introduced. Also, Fluid/Structure Interaction modeling involving the dynamics of a porous fabric (like parachute deployment) is presented. A generalization of the Navier-Stokes equations that will allow the definition of sub-domains with different permeability/porosity was developed. A level-set technique is used to track and correctly represent the moving interfaces in free-surface problems. The SUPG|OSS stabilizing Finite Element Method for the spatial approximation and the second-order Fractional Step Method for the time integration were adopted.

Effect of Porous Components on the Aerodynamics of a Bluff Body

S. Szyniszewski, M. Pelacci, J. Aguero, D. Birch (University of Surrey);
Y. Liu (Southwestern University)

The objective of this work was to measure and simulate the flow around bluff bodies with porous components experimentally. Flow via a car radiator can affect the overall aerodynamics of a vehicle. Resin injection in woven reinforced composites is another example of the flow through porous media. From the scientific perspective, the coupling of the external flow with the flow via a porous body requires is needed to simulate such problems.

We carried out wind tunnel tests on cylinders with porous components in various location to validate the current computational tools. Our samples included fluid, porous and solid domains. Force balance measurements of the drag, wake profiles and flow velocity fields were measured in the wind tunnel. The experimental results were compared with LS-DYNA simulations.

We found that the interaction between the flow in the porous domain and the external flow produced surprising flow patterns that differ significantly from the flow around a solid body. Also, the simulations showed the need for the inclusion of the turbulence model to capture the interface flow at the porous boundary adequately.

Future effort is required on the incorporation of the turbulence model in LS-DYNA porous media flow solver. Also, the future optimization of the porous media permeability and geometry could reduce the overall drag of cars and aeroplanes.

Investigating the Post Processing of LS-DYNA in a Fully Immersive Environment

E. Helwig, F. Del Pin (LSTC)

The use of virtual reality (VR) in engineering applications has been expanding for the last decade. Immersive technology is quickly becoming a tool for pre and postproduction decision-making and analysis. Virtual reality can assist in reducing the number of physical prototypes, build collaboration between various engineering disciplines, and speed up manufacturing time and hence reduce design cycles.

We examined the integration of LS-DYNA into a workflow using results from fluid-structure interaction problem. The solution demonstrates the complexity problems faced by the users of our software. The expected result was to generate a “life like” simulation model, while maintaining a high degree of engineering data in the analysis for output. The simulation data was placed in a virtual environment using a passive visualization solution, and eliminating the requirement for an active VR headset. The investigation identified key hardware and software considerations while optimizing the workflow process. Scalability, computation time, component costs and functionality were variables considered during development. It is our firm belief that seamlessly integrated visualization tools and state of the art physics solvers are in the core of future design and manufacturing pipelines.

Recent Developments in LoCo – Instant Collaboration in Simulation Data Management

R. Bitsche, M. Thiele, T. Landschoff (SCALE);
M. Koch (Dr. Ing. h.c. F. Porsche)

While computer simulation has become a standard business practice for many companies, the number of simulations performed, the size and complexity of the models, the computational costs, and the predictive capabilities of the models keep increasing. As a consequence, the effective management of simulation data and related process information has become paramount.

In the past 10 years SCALE has developed a comprehensive simulation and test data framework (SCALE.sdm) in close collaboration with the German automotive industry. Several Apps cover the entire CAE design process. This presentation focuses on recent developments in LoCo, SCALE's system for simulation data management. LoCo applies several new approaches to simulation data management, such as strict offline capabilities, automatic synchronization of relevant data, comprehensive version management of all involved objects, and novel approaches for the assembly of models.

Version control - the management of changes to data - is an important aspect of simulation data management. The version control functions of a simulation data management system are usually either based on the lock-modify-unlock or the copy-modify-merge approach. LoCo is based on the latter approach, as it is the natural choice for an environment in which individual engineers must be able to observe the effects of their own changes in isolation from the work of others.

Vehicle crash simulation is a prime example of a field where large teams of engineers simultaneously work on complex models and perform large numbers of computationally expensive simulation runs. During a vehicle development process there are several phases where a finite element model is set up based on CAD data. During such a phase a team of multiple simulation engineers creates finite element models of the various components of the vehicle in close cooperation and must make sure that these sub-models can be joined together correctly to assemble a full car model. This setting is unusual in the sense that there is no need for the individual engineers to observe the effects of their own changes in isolation. Ideally, the model setup team should be able to collaborate in a way such that each team member can instantly see and use the changes made by other team members. In the end of 2016 a new feature designated "LiveMode" was introduced in LoCo to accommodate this approach.

The new LiveMode feature was implemented in such a way that the user can seamlessly switch between normal mode (copy-modify-merge) and LiveMode (lock-modify-unlock). In LiveMode all users can access and work on the same version of the model. Components opened by one user are automatically locked for all other users and automatically released once they are closed. As data upload and download is handled by a synchronization mechanism in the background, there is no need for the user to actively "check out" or "check in" components.

The LiveMode was first introduced at selected costumers in January 2017. First experiences show that, besides the model setup use case described above, a second use case has developed. The LiveMode is also used as an "instant collaboration tool". Typically two users, who would like to collaborate on a specific task, switch to LiveMode, work on the same set of data simultaneously and then return to normal mode.

Data Management and Loadcase Composition in ANSA

T. Fokilidis, L. Rorris, T. Loiras (BETA CAE Systems)

It is common knowledge that simulations of virtual models hold a key role during the design process of a vehicle. Considering the continuously growing number of regulations, but also the different variants that a vehicle can have, one concludes to a plethora of similar or completely different simulations. As far as a simulation process is concerned one of the most demanding issues is its build up. The tools that a CAE analyst should have at ones disposal must be characterized by robustness and automation to deal with the numerous and really complex numerical simulations.

The most important case during the model build up is a comprehensive model organization. BETA CAE Systems has come up with new data types that facilitate the set-up for the LS-DYNA simulation run. These new data types come to fully support data management in both, ANSA and LS-DYNA Include, files but furthermore in SPDRM implementations. Moreover, advanced capabilities of version control and the storage of all attributes that a file needs to be followed during a simulation build up, ensure a productive process starting from file input, moving to the assembly, passing to the load-case set up and finalizing with a bulk LS-DYNA file output.

The current paper introduces the new ANSA techniques for a successful LS-DYNA simulation run set up that decreases the production time to its minimum and avoids errors that human interaction produces inevitably.

The Benefits of Scripting for CAE Engineers – How a Little Can Go a Long Way

G. Newlands, M. Thornton (ARUP)

The pressure on a CAE engineer can be great at times, with tight schedules and demands for results. Sometimes the exact tools you require are not available – but what if you could create them yourself? Writing a script or tool to speed up a process may seem daunting, or not worth the effort. Why would you spend a day writing a script to do something when it will take you a couple of hours to actually do the task? The answer is often it will not be the only time you will do the "something". It might be once, or it may be 10's or 100's of times.

This is where scripting can really help you. But it is not confined to "speeding things up". Scripting can be used in a number of areas to aid the CAE engineer:

- Custom tools to do exactly what you want.
- Custom checks specific to company guidelines to make sure your model is up to scratch.
- Model manipulation tools.
- Specify a process for others to follow.
- And many more…

This paper looks at how scripting can have a part to play in the everyday work of a CAE engineer. Using the Oasys LS-DYNA Environment software we will demonstrate the benefits of scripting through a number of worked and real-life examples. We will also show that it is actually very quick and easy to learn and to write useful scripts.

Introduction to scripting
We will have a brief introduction into the world of scripting, and how you don't need to spend days and weeks training to gain knowledge and skills that are useful.

Preprocessing
Here we will look at examples of using scripts within the preprocessing environment. This will relate to:

- Model assembly.
- Model modification.
- Building multiple models.
- Reading and manipulating external data.
- Following a process.
- Custom model checking.
- Automation.

Post processing
Here we will look at examples of using scripts within the postprocessing environment. This will relate to:
- Interrogating and combining LS-DYNA output to produce user defined components that are applicable to your situation.
- Automatic post processing of results.

Conclusions
We will conclude with the benefits of scripting, and how they apply to real world projects.

Workshop on MPIP - Material Parameter Identification Process with 4a impetus

A. Fertschej, B. Jilka (4a engineering)

The material card generation using 4a impetus solution is shown exemplary for thermoplastic materials. The latest software features of 4a impetus are presented, especially the “Autofit” workflow and implementation of anisotropic material laws are main topics. New test methods focusing on failure estimation and component validation complete the workshop topics.

Besides the presentation there will be time for interactions between the presenters and the audience.

Wednesday Morning Sessions

10:40 – 12:20

Modeling of Deformation and Damage Behavior of High Strength Steels under Multiaxial Crash Loading

D. Sun, A. Trondl. S. Klitschke (Fraunhofer IWM)

The predictive capability of crash simulation concerning material failure is still in need of improvement due to the coupled complex influences of triaxiality, strain rate and temperature. Because of their lower ductility the use of high strength and ultrahigh strength steels requires a more accurate prediction of failure. Until now experimental results about the influences of triaxiality on failure are available mainly for quasi-static loading and systematic investigations which take into account the influences of triaxiality as well as strain rate on deformation and failure are missing. Effects like adiabatic softening and influence of strain rate on strain localization in different stress states have not yet been determined systematically and quantitatively. It is also open, which kind of numerical methods can be used in crash simulations to model the strain-rate dependences of deformation and damage behavior under multiaxial loading in reliable and efficient manner.

Therefore, different specimen tests on high strength steels were performed under tension, shear, biaxial tension and bending at different strain rates. Strain fields in the specimens were measured with two high-speed video cameras for 3D strain field analysis and the local failure strains were determined. Adiabatic temperature rise especially at the damage locations was measured with an infrared high-speed camera. The micro-mechanisms of damage under different strain rates and triaxialities were analyzed by fractography and metallography. A material model which considers the influences of strain rates and adiabatic effects was suggested and applied to simulate different loading scenarios. To avoid the time-consuming fully coupled thermal-mechanical approach, a strain-rate dependent Taylor-Quinney coefficient was introduced to control local heating arising from plastic work. It leads to variable softening effects at different strain rates.

Based on these results the material model 224 in LS-DYNA was extended at DYNAmore for the capability of a strain-rate dependent Taylor-Quinney coefficient. This model was used to simulate the strain-rate and temperature dependence of the flow behavior. For damage modeling the GISSMO model taking into account the influences of triaxiality and strain rate on failure strain was used. The parameters for the material model and the damage model were determined by simulating shear tension, holed tension, smooth tension, notched tension and Nakajima punch tests. The measured force vs. displacement curves and strain and temperature distributions can be well calculated using the determined model parameters. The influence of strain rate on failure strain is quite different for the different steels and the different triaxialities. Component tests on a rectangular profile and simulations were performed to validate the used material models.

A Status Review of Failure Simulation at the Federal Aviation Administration

D. Cordasco, W. Emmerling (Federal Aviation Administration);
P. Du Bois (Consultant)

The Federal Aviation Administration (FAA) is developing models and methods for simulating fan blade off impact for engine containment. Accurately predicting deformation and failure in such an event is essential for advancing the industry and FAA goal of certification by analysis. Furthermore, industry and government engineers require publicly available tools to standardize the analysis during the engine design, development and certification phase. In addition to the high strain rates and steep temperature gradients typically realized in the impact event, a complex three-dimensional state of stress develops which is dependent on the impactor and target material properties, geometry, relative orientation and impact velocity. The resulting failure surfaces which characterize the plastic failure strain by the state of stress fully defined by the triaxiality and Lode parameter are highly nonlinear. Often, crucial stress state data points are not easily realized by typical specimen level standard tests necessitating the development of new experiments to fully characterize the surface for a given material.

The FAA Aircraft Catastrophic Failure Prevention Program (ACFPP) working in conjunction with government, academia, and industry has made considerable progress in developing a family of tabulated plasticity models based on the Johnson-Cook approach available in LS-DYNA (MAT_224, MAT_224_GYS, MAT_264). These models have been created to meet the particular need for simulations that can predict the transition from petaling to plugging failure modes as well as the intermediate mixed modes which develop with changing target plate thickness and impact speed. In addition, the material constitutive and failure response for several typical aerospace metals including Ti-6Al-4V, Al-2024, Inconel-718, and stainless steel 410 is underway. During testing, it was observed that a tensile-compressive asymmetry in yield as well as an anisotropic directional dependence may arise for some materials depending on inherent material microstructure and also the manufactured plate thickness. Therefore, each model offers an increasing degree of capability in modeling these phenomena. These tasks have highlighted some of the challenges in complex failure modeling and underscore the need for a robust experimental testing program to complement computational modeling.

A Comparison of Damage and Failure Models for the Failure Prediction of Dual-Phase Steels

F. Andrade (DYNAmore);
M. Feucht (Daimler)

The aim of this contribution is the comparison of different damage failure models that are available in LS-DYNA. In particular, the focus is concentrated on the failure prediction of dual-phase steels which are largely used in the automotive industry. Typically, such alloys provide a good compromise between ductility and strength for which this kind of material is also often used in safety relevant components. Examples are parts of B-pillars, side rails and cross members, i.e., parts that may be subjected to intensive loadings in a high speed car crash scenario. In contrast to some other usual alloys, dual-phase steels are often reasonably isotropic and well described by J2-based plasticity. This allows the use of simple and very efficient material formulations (e.g., *MAT_024 in LS-DYNA [1]) without excessively losing accuracy in crash simulations.

Despite the fact that the elastoplastic behavior of such alloys can be generally well captured by simple plasticity models, the fracture behavior in practical applications still demands the consideration of several effects like stress state dependence, nonlinear paths, material instability, spurious mesh dependence, among others. Therefore, we consider three different damage/failure models available in LS-DYNA in order to calibrate the fracture behavior of a typical dual-phase steel: (a) the GISSMO damage/failure model [1–3]; (b) the Gurson-Tvergaard-Nedlemann model [4, 5] and the Cockcroft-Latham failure model as implemented in *MAT_135 in LS-DYNA [1, 6].

We will shed some light on the differences among these models and verify their ability in reproducing experimental data on the coupon level for different stress states. The goal is to understand the advantages and limitations of each model concerning the prediction of failure. A detailed discussion will then follow the results obtained with the three models.

Literature

[1] LTSC: “LS-DYNA Keyword User’s Manual – Volume II: Material Models”, Livermore, 2016.
[2] Neukamm F., Feucht M., Haufe A., Roll K.: “On closing the constitutive gap between forming and crash simulation”, 10th International LS-DYNA Users Conference, Detroit, 2008.
[3] Andrade F.X.C., Feucht, M., Haufe A., Neukamm F.: “An incremental stress state dependent damage model for ductile failure prediction”, International Journal of Fracture, 200:127, 2016.
[4] Tvergaard V., Needleman A.: “Analysis of cup-cone fracture in a round tensile bar”. Acta Metall 32:157–169, 1984.
[5] Feucht F.: “Ein gradientenabhängiges Gursonmodell zur Beschreibung duktiler Schädigung mit Entfestigung”, PhD thesis, Technische Universität Darmstadt, 1999.
[6] Lademo O.-G., Engler O., Keller S., Berstad T., Pedersen K.O., Hopperstad O.S.: “Identification and validation of constitutive model and fracture criterion for AlMgSi alloy with application to sheet forming”, Materials and Design 30, pp. 3005-3019, 2009.

Applications of Multiscale and Subcycling Methods for Press Hardened Steel Parts Failure Assessment

Y. Drouadaine (ArcelorMittal)

Press Hardened Steel (PHS) parts, like B-pillar made of Usibor® 1500, are today the best steel solutions in order to fulfil the most aggressive crash requirements with the lowest body structure mass. However, these severe crash conditions request a failure risks analysis during the vehicle design process.

The usual Usibor® 1500 failure modelling method proposed by ArcelorMittal always remains compatible with a current full car model mesh size, whether for base metal or spot-weld Heat Affected Zone (HAZ) or laser welded line. Shell elements in the mesh size range 3 mm – 5 mm are used with CrachFEM fracture criterion.

One alternative, for an improved failure risk prediction, would be to use a meso-scale model with solid element mesh size less than 0.5 mm for the most critical areas. This method leads to a heterogeneous model when used in a full car crash simulation (finest 3D elements to 2D elements mesh size ratio close to 10 times).

In this case, the time step is imposed to the whole model by the smallest elements. This may lead either to increase drastically the CPU time or to provide results affected by an excessive added mass level in case of a too ambitious imposed time step.

LSTC and DYNAmore have recently developed (2014) new features like Subcycling *CONTROL_SUBCYCLE_K_L and Multiscale *CONTROL_SUBCYCLE_MASS_SCALED_PART_{SET} in order to reduce the calculation time without adding a huge amount of mass. It seemed interesting to use these new features in the field of heterogeneous crash models without none of the two time-step issues previously mentioned.

The present paper will describe results obtained by these new features, especially the multiscale option. It is applied to different mesh size heterogeneous models, from a three points bending test on a specimen till a full car model during side impact. Energy balance stability, parts deformation, local plastic strain will be presented and compared with the ones obtained by the unique time-step reference calculation.

A CPU time performances comparison will be also presented and then, the multiscale method efficiency will be established for each tested model. The elapsed time saving reaches 34% in case of a 3D local failure analysis performed on a full vehicle during side impact, without any negative impact on the quality of results.

Current limitations of heterogeneous models, questions and expectations regarding the multiscale approach and the 3D-2D transition modelling will be also discussed to conclude the presentation.

Topology Optimization Methods based on Nonlinear and Dynamic Crash Simulations

Prof. F. Duddeck, M. Bujny, D. Zeng (TU München)

Topology optimization for crashworthiness has been investigated during the last years, starting from methods based on linear elastic and static simulations[1] or so-called equivalent static loads (ESL) obtained by a single nonlinear crash simulation with a subsequent optimization loop based on the linear stiffness matrix and the corresponding sensitivities[2]. Both methods do not consider material nonlinearities in their optimization process, which are essential for structural components designed for energy absorption, although it is well-known that plasticity and failure play an important role.

As alternative, optimization methods have been proposed, which use fully nonlinear and dynamic crash simulations. The first method, proposed for example in Patel's PhD thesis, uses a hybrid cellular automata approach (HCA) and derives optimal structures using a homogenized energy density approach where each finite element is modified until the highest degree of homogeneity is achieved[3]. Because this is not fully appropriate for thin-walled structures, Hunkeler modified the approach (HCATWS – Hybrid Cellular Automata for Thin-walled Structures) such that deformation energy is only homogenized between larger structural entities (i.e. thin walls)[4]. The most recent method, a combined level set method (LSM) and evolutionary approach, was then proposed by Bujny et al. where more appropriate objectives and constraints can be used with the drawback of higher computational costs[5].

In this paper, the latest results for HCATWS and LSM will be presented, see also[6,7]. Special focus is here the investigation of the influence of different material models for plasticity and failure. Examples are, for example, inspired by recent material model development for magnesium alloys with a characteristic anisotropy in the plasticity model[8]. As a result, it is shown that the optimal topologies depend on the correct material model and that it is necessary to use nonlinear and dynamic finite elements for crash topology optimization.

Literature

1 Duddeck F, Volz K. A new Topology Optimization Approach for Crashworthiness of Passenger Vehicles Based on Physically Defined Equivalent Static Loads. ICRASH Conference, Milano, Italy, 2012.

2 Park G.-J. Technical overview of the equivalent static loads method for non-linear static response structural optimization. Struct. Multidisc. Optim. 43(3):319-337, 2010.

3 Patel NM. Crashworthiness Design Using Topology Optimization. PhD thesis, University of Notre Dame, USA, 2007.

4 Hunkeler S. Topology Optimisation in Crashworthiness Design via Hybrid Cellular Automata for Thin-walled Structures. PhD thesis, Queen Mary University of London, UK, 2013.

5 Bujny M, Aulig N, Olhofer M, Duddeck F. Evolutionary Crashworthiness Topology Optimization of Thin-Walled Structures. 11th ASMO UK / ISSMO / NOED: Int. Conf. on Numerical Optim. Methods for Engrg Design, Munich, Germany, 2016.

6 Zeng D, Duddeck F: Improved Hybrid Cellular Automata for Crashworthiness Optimization of Thin-Walled Structures. Struct. Multidisc. Optim., online, 2017.

7 Duddeck F, Hunkeler S, Lozano P, Wehrle E, Zeng D: Topology Optimization for Crashworthiness of Thin-walled Structures under Axial Impact Using Hybrid Cellular Automata. Struct. Multidisc. Optim. 54(3): 415–428, 2016.

8 Gese H, Klamser H, Stolfig P, Gross M, Meyer LW, Abdel-Malek W. Weiterentwickeltes Materialmodell zur Bewertung des Crashverhaltens von Mg-Extrusionsprofilen. 5th Nordmetall Kolloquium, Adorf, Germany, 2009.

LS-TaSC Product Status

K. Witowski (DYNAmore);
W. Roux (LSTC)

The LS-TaSC Version 3.2 topology and shape design tool is presented. The presentation introduces the multi-point numerical derivatives scheme that allows constrained optimization using the mass fractions and load case weights as variables. This allows constrained optimization using any response or mathematical expressions as constraints or objectives.

In addition, the capabilities currently under development such as designing to maximize the fundamental frequency and new geometry definitions will also be presented.
All these capabilities will be illustrated using examples.

A Systematic Study on Topology Optimization of Crash Loaded Structures using LS-TaSC

K. Weider, A. Marschner, Prof. A. Schumacher (University of Wuppertal)

In this contribution the chances and limitations of using LS-TaSC for topology optimization of crash loaded structures will be shown. With two demonstrative examples the influence of scaled loads, intermediate densities and conflicting constraints are discussed.

As no analytical sensitivities are available in explicit finite element simulations, LS-TaSC uses the interal energy density as sensitivity, which makes it an efficient optimization strategy for stiffness applications. For the provided examples, both a maximum stiffness as well as a crashworthiness design is developed. In the latter case, the multipoint method of LS-TaSC is used to cope with the acceleration of an impactor as objective function.

To compare the resulting topology with other topology optimization methods, a thickness optimization of a shell interpretation is performed. The structural behavior is also compared with that of the optimized solid model.

Free-Form Shape Optimization on CAD Models

D. Baumgärtner, M. Breitenberger, Prof. K.-U. Bletzinger (TU München)

A novel approach for the free-form shape optimization on CAD models is presented. The optimization workflow consists of three steps:

- Creating a finite element mesh of the CAD model
- Vertex-morphing for exploring the best design (Millions of DOFs possible)
- Isogeometric B-Rep mapping for updating the CAD model

The output of the proposed optimization workflow is a complete high-quality CAD model. The presented techniques allow controlling selectively geometric continuities (up to G^2) on the edges of the mechanical optimized CAD model.

Various examples from automotive and aircraft industry confirm the quality, flexibility, and robustness of the proposed optimization workflow and thus highlight the advantages of integrating vertex morphing and CAD modeling.

Hot Rolling Simulation of Aluminum Alloys using LS-DYNA

P. Simon, G. Falkinger (AMAG);
S. Scheiblhofer (LKR Ranshofen)

Aluminum and its alloys are widely used and applications of aluminum alloys can be found in all kind of industries, from packaging to transportation, from consumer electronics to architecture and lighting. In order to obtain a better understanding of the production process of aluminum alloy plates and sheets AMAG rolling GmbH strengthens the use of coupled thermo-mechanical finite element methods in combination with microstructure based user defined material models.

The presented work shows a hot rolling simulation, which is an important step of the process chain. A detailed overview of 2D and 3D models for multi pass simulations is given and a graphical user interface for an automated simulation setup is presented. With the help of a Python based routine the virtual billet is trimmed and remeshed after each rolling pass in order to reduce the computational time. A comparison of the implicit simulation to the explicit simulation regarding the computational time is made and some simulation results are shown.

Furthermore a comparison of the obtained rolling forces and temperatures between measurements of the hot rolling mill and simulation is given for an industrial pass plan.

Tool Cooling Simulation for Hot Forming

T. Kuroiwa (JSOL)

Our solution development status on hot forming technique will be reported with a special focus on tool cooling. Hot forming is one of the most promising techniques among which contribute to saving weight of automobiles. Thermal management of blank material is especially important to ensure the hardness of it, since sufficiently rapid cooling of heated blank is needed to cause martensitic transformation.

There are two reasons to do a CAE analysis of hot forming, to our knowledge.

1. Save cost in prototyping of tools with pipes for cooling water.
2. Accurate prediction of distribution of martensitic phase.

Our goal is to develop designer's CAE tool of hot forming which predict cooling performance of tools and martensitic phase distribution of blank materials. Simulation of hot forming is a kind of multiphysics problem since it is consist of many physical processes: plastic deformation of blank materials, phase transition of blank, conjugate heat transfer between blank-tool-cooling water, and flow of cooling water. In this paper we limit our scope on tool cooling and present how recent development of LS-DYNA contributes to the performance increase of the simulations.

The Structural Conjugate Heat Transfer Solver – Recent Developments

T. Klöppel (DYNAmore)

It is the objective of this contribution to present and discuss recent developments in the structural conjugate heat transfer solver in LS-DYNA to be used mainly for the thermo-mechanical coupled simulation of complex manufacturing processes such as for example hot forming, heat treatment and welding. Beside the multi-physics nature the complexity of these processes is due to the usually challenging constitutive behavior of the employed materials.

In the next section some of the thermo-mechanical material models in LS-DYNA are briefly discussed and recent modifications and enhancements to those models are shown. Furthermore, the new material **`*MAT_GENERALIZED_PHASECHANGE`** (**`*MAT_254`**) is introduced, which features a very flexible and widely applicable modelling approach for phase change kinematics.

The paper also addresses two new options for the thermal contact. First of all, heat transfer between a shell edge and a surface (either shell or solid face) can now be considered. Second, a special welding contact formulation has been implemented. Above a certain temperature, the formulation switches from a sliding to a tied formulation and uses different parameters for the heat transfer. Although both modifications have been motivated by welding applications, they have also proven to be helpful for other applications.

The presentation of a new heat source boundary condition in the fourth section completes this work's list of developments. It is defined with the keyword **`*BOUNDARY_THERMAL_WELD_TRAJECTORY`** and allows modelling of a moving heat source for coupled and thermal-only simulations and provides a very flexible input for the shape of the heat source.

An Analysis of the Hot-forming Process with Thermal and ICFD Simulations

M. Kintsch, S. Szabo, R. Schneider (Voestalpine Automotive Components);
W. Rimkus (Hochschule Aalen)

As a result of the continuous rising requirements, such as complex component geometries and crash performances, high strength steel components are playing an increasingly important role in the area of automotive car body construction. The forming limits of high strength steels can be enhanced if deformation takes place above corresponding recrystallization temperature. Furthermore, the specific cooling rate of the sheet material is important to be able to apply a targeted hardening of the component during the forming process. For this, cooling channels within the forming tool are necessary to achieve the desired cooling rate for the hardening process.

The physical processes (heating and cooling) should be determined in the digital product design to reduce the development time and to decrease the costs for validation. For this way of simulation there are different boundary conditions and models to be considered that strongly influence the complexity of such calculations.

In this study a comparison of different thermal simulation models, considering the active cooling by an incompressible computational fluid dynamic (ICFD) calculation, is carried out. Hereby, two different assumptions for the cooling channel boundaries are compared. In the first conjecture the heat transfer coefficient between cooling fluid and the steel die is set constant. In the second assumption the heat transmission is calculated by an ICFD simulation and allocated to corresponding tool surface areas. Further, the obtained heat transfer parameters are used to calculate the thermal load of the tool after multiple Hot-forming cycles.

The overall aim of this study is an assessment between the accuracy of the different applied calculation models and the required effort of the simulations. Thereby, not only the computation time is considered but also the additional work for pre- and post-processing is taken into account to make a conclusion about an eventual benefit of a CFD-Simulation for hot forming processes.

Investigating the Influence of Local Fiber Architecture in Textile Composites by the Help of a Mapping Tool

M. Vinot, M. Holzapfel (DLR);
C. Liebold (DYNAmore)

Standard approaches for the modelling and simulation of composite structures rely on the homogenization of material properties on unidirectional plies. Doing so, simulations lose the ability to precisely describe local phenomena and complex failure mechanisms. In the research campus ARENA2036, the project DigitPro (Digital Prototype) develops a method based on a closed simulation process chain to take into account potential production effects by braided or woven composite structures. Starting from the process simulation, crucial information like fiber orientation and waviness is mapped on a target mesh for structural analysis. The resulting model is then investigated and potential needs to change the component's geometry or the manufacturing process are detected.

The mapping tool ENVYO®, developed at DYNAmore, offers various possibilities for the transfer of information generated from process simulation. Thus, it is necessary to investigate the impact of mapping algorithms and respective parameters on the structure simulation. The present paper details the mapping procedure for textile composites. The influence of local fiber architecture is finally investigated on a generic structure and compared to a standard approach.

The Digital Prototype as Part of ENVYO – Development History and Applications within the ARENA2036 Environment

C. Liebold, A. Haufe (DYNAmore);
M. Vinot (DLR); J. Dittmann, P. Böhler (University of Stuttgart);
H. Finckh, F. Fritz (ITV Denkendorf)

DigitPro, a sub-project of the government funded research project Active Research Environment for the Next generation of Automobile (ARENA2036) deals with the development of a Digital Protoype, a closed simulation process chain which not only covers different simulation disciplines such as crushing or process analysis, but also various material modeling approaches on the micro-, meso-, and macro level. Various software tools are being used by the project partners, namely the German Aerospace Center (DLR), the Institute of Textile Technology and Process Engineering (ITV), and the Institute of Aircraft Design (IFB) at the University of Stuttgart.

Within recent years, the capabilities needed to close the simulation process chain have been implemented into the mapping tool ENVYO which was officially introduced at last year’s German LS-DYNA users meeting. This paper will give an overview on recently implemented features and will point out the significance of ENVYO for a closed simulation process chain and data management during the component’s design and verification lifetime. Within this context, an overview on future topics which will be realized within the succeeding project “Digital Fingerprint” will be given.

Acknowledgements
This research and development project is funded by the Federal Ministry of Education and Research (BMBF), Germany under the supervision of Project Management Agency (PTKA) in Karlsruhe. The author is responsible for the contents of this presentation.

Manufacturing Simulation as Part of the Digital Prototype

P. Böhler, J. Dittmann, D. Michaelis, P. Middendorf (University of Stuttgart);
C. Liebold (DYNAmore)

The research project Active Research Environment for the Next generation of Automobile (ARENA2036) is a long term project funded by the Federal Ministry of Education and Research Germany. Within this project four sub-projects are located. DigitPro, one of those sub-projects, deals with the development of a Digital Protoype. A closed simulation process chain is built which not only covers different simulation disciplines such as crushing or process analysis, but also various software solutions and material models. The main goal is to use the digital prototype to decrease the weight if an automotive structure by 10% and the development time by 50%.

In this project one of the focused manufacturing processes for composite structures is the braiding technology followed by an infusion process. A complete numerical prediction is necessary for the braiding as well as for the infiltration process to decrease the development time and to increase the mechanical performance of braided structures. Within this work an overview of the newest developments in braiding and infiltration simulation and especially in the transfer of the necessary data from one process to the next is given.

An overview on the succeeding project “Digital Fingerprint” will be given as the results of the project DigitPro will be used there.

Acknowledgements
This research and development project is funded by the Federal Ministry of Education and Research (BMBF), Germany under the supervision of Project Management Agency (PTKA) in Karlsruhe. The author is responsible for the contents of this presentation.

Textile Process Simulation as Part of Process Chain

H. Finckh, F. Fritz, G. Gresser (ITV Denkendorf)

By simulation of the whole textile generation process at a micro-/mesoscopic level, the interaction between machine parts and the textile as well as the threads among each other can be studied in detail. The result is a highly detailed virtual textile which can be either used for further process simulations like draping or virtual testing. In the latter the computation of mechanical properties for macroscopic draping or structural applications can be determined.

Enhancements are presented referring braiding simulations using LS-DYNA. It is shown how the braid attaches to a complex change of contour dependent on chosen process and material parameter.

Another emphasis is put on a new kind of multiaxial woven fabric called Open Reed Weaving (ORW). Multiaxial weaving technology allows for the addition of two independent thread systems under a nearly arbitrary angle on to the conventional 0°/90° woven fabric. These additional warp fibers can be added partially or covering the whole width of the textile and therefore allowing a wide range of textiles with various designs and textile constructions and also varying materials. The resulting mechanical properties of production parameters shall be investigated by the developed fabric-models.

This presentation will give an overview on the newly developed simulation of the multiaxial weaving process and the braiding simulations developed in the BMBF-project DigitPro, a sub-project of the government funded research project Active Research Environment for the Next generation of Automobile (ARENA2036).

Assessment of the Capacity of a Reinforced Concrete Structure for Impact with Military Jet Aircraft

M. Miloshev, M. Kostov (Risk Engineering)

The current paper presents analysis of a reinforced concrete structure for impact with Phantom F-4 military aircraft. The analysis is performed by the missile-target interaction method. The finite element (FE) model of the aircraft is validated by comparison of the load-time function, obtained by numerical impact simulation into rigid planar target to the one, obtained from full-scale impact test. The latter was performed in 1988 at the SANDIA national laboratory in the US and the results were made public. Two models of the jet fuel are investigated – rigid fuel model and Smooth Particle Hydrodynamics (SPH). The rigid fuel model refers to the case where the mass of the fuel is lumped to the structural elements of the fuel tanks. Very good match between the numerical and experimental results is achieved which implies that the FE model captures well the mass and stiffness distribution of the real aircraft and is suitable for performing missile-target interaction analyses.

The target is a reinforced concrete structure with an asphalt layer which is laid on top of it. It is assumed that this asphalt increases the capacity of the structure and acts as an energy dissipating layer in case of high velocity impact. A number of analyses is performed assuming asphalt material properties, defined for different temperatures and strain rates. The parameter which is compared as a result from the analyses is the area of perforation of the structure. It is related to the amount of debris which could penetrate, as well as to the assessment of the consequences of possible fire inside.

Evaluation of Debris Modeling Technique on Failure Simulation of Concrete Structures

S. Tokura (Tokura Simulation Research); K. Niwa (Terrabyte)

It is a crucial issue to protect important buildings, especially nuclear power plants against large natural disaster or terrorist attack. Nuclear power plant is a concrete structure and highly accurate prediction of the failure modes, e.g., crack propagation, penetration and spalling is required at the impact of projectile. In reality, when concrete material is failed, the failed material is not deleted and piles up as debris instead. In the case of second impact or subsequent seismic load, existence of debris cannot be ignored since the impact force is loaded on the structure through the debris.

Although many sophisticated failure models are implemented in LS-DYNA, the generation of the debris cannot be simulated when the failure models working with erosion capability are used since the failed elements are deleted. In contrast, *DEFINE_ADAPTIVE_SOLID_TO_SPH/DES keyword provides the simulation technique considering the effect of the debris since the failed solid elements are replaced with the particles. In this presentation, more realistic post-failure behavior of the concrete structure using *DEFINE_ADAPTIVE_SOLID_TO_SPH/DES is shown and the application of this capability to the real problem on concrete structure is discussed.

Comparison of the RHT Concrete Material Model in LS-DYNA and Ansys Autodyn

C. Heckötter, J. Sievers (GRS)

The goal of this paper is to compare the implementations of the RHT concrete model developed by Riedel, Hiermaier and Thoma in the commercial hydrocodes ANSYS AUTODYN and LS-DYNA for different stress conditions. The considered stress conditions include hydrostatic compaction related to the Mie-Grüneisen equation of state in conjunction with a p-α model, uniaxial tension, uniaxial compression and tri-axial compression. Parameters of the failure surface and residual strength surface were fitted to data of several tri-axial compression tests. Major differences are observed regarding the behavior under uniaxial tension. It is pointed out that the current model implementations do not consider fracture energy. Hence, the simulation results depend inherently on mesh size in cases where tensile properties are relevant.

Furthermore, comparative validation studies based on impact tests dealing with punching failure of reinforced concrete slabs subjected to hard missile impact have been performed. The sensitivity of the ballistic limit velocity as well as residual missile velocities on model parameters is discussed.

The work of GRS was carried out in the framework of the German reactor safety research program sponsored by the German Federal Ministry of Economic Affairs and Energy (BMWi).

The Use of LS-DYNA for Body NVH "The Success so far"

T. Zeguer (Jaguar Land Rover);
Y. Huang, M. Souli (LSTC)

For many years LS_DYNA is mainly used for crashworthiness impact analysis at the same time the software has many new features added to it and is progressing toward a multi physics software that have the potential for many applications tailored to many field including NVH (Noise, Vibration and Harshness). To the customer NVH is defined in terms of how the car feels with regards to vibration levels and how the car sounds with regards to the perceived loudness and quality of the interior noise. To the development engineers, NVH is defined in terms of measurable tactile and acoustic responses.

This presentation will show examples on how LS_DYNA NVH compare to one of Today's NVH mainly used software (Nastran, Abaqus, Optistruct and equivalent software). Comparison will be done on full BIW NVH model for NTF (Noise transfer Function), Dynamic stiffness, Global modes (Torsion, Bending), Normal modes and Equivalent Static stiffness.

Recent Developments for Frequency Domain Analysis in LS-DYNA

Y. Huang, Z. Cui (LSTC)

Since the 971 R6 version, a series of frequency domain features have been implemented to LS-DYNA. They include FRF, SSD, random vibration, acoustics, response spectrum analysis and fatigue analysis. With the implementation of these features, the application of LS-DYNA is extended to the new area of frequency domain analysis.

Some new developments and updates have been made for the frequency domain features, since the last European LS-DYNA Users' Conference in 2015. The main developments and updates include

- ERP (equivalent radiated power) computation based on SSD
- Direct SSD solver
- Incorporation of phase difference in cross PSD in random vibration analysis
- new ASCII and binary databases for post-processing of the frequency domain analysis results
- Incorporation of phase difference in cross PSD in random vibration analysis
- Incorporation of rotational dof in excitation and response in FRF and SSD ...
- And others

These new developments and updates were made to meet the requirements from users, and to enhance the capabilities of frequency domain analysis in LS-DYNA.

This paper provides a review on these new developments and updates.

Acoustic Analysis for Impact Sound with LS-DYNA

R. Ishii (JSOL);
T. Yamamoto (Nihon Emsco);
Z. Cui, Y. Huang (LSTC)

LS-DYNA is heavily used to analysis transient phenomenon like car crash and makes a huge achievement about physical simulation in a wide variety of industry.

For the goal of LS-DYNA, one-model , one-code as solution it give you, a wide variety of function has been developed at each section.

Nowadays, LS-DYNA has been developed further and become possible to evaluate Frequency domain analysis and Acoustic analysis as FRF/SSD/AcousticBEM/FEM etc.

It will be important to consider how it works as actual experiment or how to define and use in order to make it possible to evaluate multi-physics extensively.

This study is intend for the evaluation to impact sound when a crash occurs between one object and another. To compare the result with experiment and check the both sound by ear would be fulfilled in this paper.

The conventional other software executing acoustic analysis executes with frequency domain or also do a transient analysis with the assumption that steady state continues in appropriate period.

It defines the source of sound on the surface of object vibrating and put it as input to BEM/FEM.

As it is well known LS-DYNA can do a transient analysis, it can calculate the source of sound itself. The object vibrates due to collision and the sound propagates to the air. This is the mechanism for sound. LS-DYNA further is able to calculate acoustic analysis taking over previous collision simulation.

The comparison between LS-DYNA analysis and experiment is theme of this paper.

In the future, In order that LS-DYNA has evaluated physically many kinds of simulation in one-model and it will be alternative to actual experiment.

The consideration at this time is first step to confirm the validity and helpfulness of this solution.

Dr. Yun and Dr. Zhe have made an extensive contribution to this comparison. I would like to express heartfelt thanks to them.

A New Eigensolver for High Performance NVH Analysis: MCMS (Multi-Level Component Mode Synthesis)

Prof. K. Chang-Wan (Konkuk University);
R. Grimes (LSTC)

NVH (Noise, Vibration, and Harshness) problem is one of the most important targets for comfortable and quiet product design. Especially, in automobile industry, as the degrees of freedom of NVH automobile FE model increases, from millions of degrees of freedom to tens or hundreds millions of degrees freedom, the computational time for NVH analysis becomes serious bottle neck in automobile design and analysis process. Therefore, it has been strongly desired to reduce the computational time of eigenvalue problem analysis, which is the most important part of NVH analysis.

In early 2000, the Automated Multi-Level Substructuring (AMLS) method was developed to reduce the computational costs of NVH analysis by the group of University of Texas at Austin, in which author of this paper is also involved in the development of AMLS method as well as Fast Frequency Response Analysis(FFRA) method. As the AMLS method provide significant contributions on automobile NVH analysis, many CAE softwares such as NASTRAN, ABAQUS, Hyperworks released new feature which is similar to AMLS. LSTC has developed a new eigensolver, MCMS(Multi-level Component Mode Synthesis). MCMS generally produces approximate eigensolutions that are less accurate than those computed by Lanczos method, but the error can be tolerated in automobile NVH applications.

Workshop on SDM and CAE-Processes with SCALE Solutions

R. Bitsche, G. Geißler (SCALE)

The workshop gives an overview of the SCALE SDM products such as LoCo, CAViT and Status.E.

There will be a discussion on how to benefit from SCALE solutions as a user or project manager. The application of selected uses cases will be presented within live demos. Examples of typical CAE workflows and process automation using SCALE SDM applications are introduced.

A lively discussion at the end of the workshop is very welcome to investigate a potential integration of SDM software in your environment.

Workshop on the New LS-PrePost Interface for ICFD Preprocessing

I. Çaldichoury (LSTC)

A new interface for model setup is under development in LS-PrePost where the user can easily assign physical properties and options for his model. LS-PrePost will then automatically translate the definitions into keyword format.

The current capabilities focus on the ICFD solver but will be extended in the future.

Wednesday Afternoon Sessions

15:40 – 16:55

Enhancing Fracture Prediction by Local Material Property Distribution – Feasibility Study

D. Riemensperger (Adam Opel);
A. Haufe (DYNAmore)

FE Implementation of AA6xxx Series Aluminum Pre-Strain Dependent Strengthening Response During Paint Bake

S. Jurendic, Z. Liang, R. Burrows (Novelis); S. Saha (RWTH Aachen)

As the drive for light-weight materials in the automotive industry increases, so does the use of heat-treatable AA6xxx series aluminum alloys in both body panels and structural components. These alloys have a distinct advantage in that they can be formed in a soft state of temper and later on heat treated in the paint bake process to achieve a higher overall component strength. The material maintains good formability throughout the forming process and achieves high strength in the final heat treated component.

With this, a new set of challenges has arisen in finite element modelling of components made from these materials, which now has to account for the material being in different metallurgical states during production versus final application (e.g. crash analysis). To further complicate things, the strengthening response of the material during final heat treatment is highly dependent on the amount of pre-strain that locally arises in the material during forming.

In this work, a microstructural model is applied to predict the yield strength of the material in a deformed and paint baked condition. The model considers influences of pre-strain, time and temperature on the microstructural transformations taking place in the material during heat treatment. These results are then implemented in a finite element model of a deformation process. Both the microstructural model as well as the component level finite element model are validated for the case of tensile test measurements at different levels of pre-strain and heat treatment parameters typical of an automotive paint bake process. To consider more complex deformation, a further validation case of a cylindrical deep drawing process is presented. Finally, simulation of a static axial tube crush test is carried out to illustrate the influence that inclusion of the pre-strain dependent paint bake response has on the simulation results.

Experimental Investigation and Numerical Characterization of the Bake-Hardening Effect of a Two-Phase Steel

D. Koch, A. Haufe (DYNAmore);
M. Feucht (Daimler)

Topology Optimization of the Bogie Structure of a Tracked Military Vehicle

K. Akcengiz, B. Balaban (FNSS Savunma Sistemleri)

Military vehicles are operated under harsh conditions on battle field and required to be reliable for successful combat operations. On the other hand, mass reduction of the structural parts is a critical issue to increase the vehicle performance in terms of mobility. Structural optimization methods are commonly utilized to get lighter and stiffer designs to increase the mobility and survivability performance without any strength reduction in structural integrity.

In this study, topology optimization is performed on a bogie structure, which is a suspension system mounting provision part, of a tracked military vehicle. Multiple loading scenarios are considered that correspond to different operating conditions (velocity and terrain profile) of the vehicle. The objective of the optimization study is to reduce the mass of bogie part without a loss in stiffness. As a first step, topology optimization is carried out with LS-TaSC software using Hybrid Cellular Automata (HCA) method which is a heuristic, gradient based approach. Next, member size control, draw direction and mass distribution parameters are considered and the optimized results are compared with the original design.

Improvement of Response Surface Quality for Full Car Frontal Crash Simulations by Suppressing Bifurcation using Statistical Approach

M. Okamura (JSOL)

In recent years, it has become increasingly important to take into account dispersion of product quality in automotive industry. Liability and performance has been guaranteed by adding safety margin to its target in the past. However, needs in cost reduction and trade-off of conflicting requirements do not allow manufacturers enough amount of safety margin anymore.

A common way to assess robustness of the structure in full car crash is to build response surface and study sensitivity of input scatter to output, and there are many papers available on the issues. However, most of these papers handle linear problems or problems where bending modes are dominant such as side crash simulations.

When it comes to frontal or rear crash where buckling mode is dominant, quality of response surfaces tend to be poor, since bifurcations in behavior bring high non-linearity to response surfaces. One measure is to increase the number of simulation runs in order to improve the accuracy of response surface, but as the size of full car simulation models becomes bigger, it is not affordable to run over 100 runs most of times.

The fundamental problem is that the response surface is with gaps due to bifurcations so that trying to fit highly non-linear response surface by adding points is not the absolute solution, but to reduce non-linearity of the surface in order to make it easy to fit.

In this study, scatter propagation mechanism is visualized based on statistical calculations, and structural design of front structure of an automobile is enhanced in order to suppress bifurcations with help from a statistical analysis software DIFFCRASH. Triggers of bifurcation are located and mechanisms of the bifurcations are studied, and design modifications are made to stabilize the deformation modes. As a result, the roughness of response surface has been reduced, and accuracy of the response surface has been improved.

Combined Analysis of LS-DYNA Crash-Simulations and Crash-Test Scans

L. Jansen, D. Borsotto, C. Thole (Sidact)

In robustness campaigns and optimization processes metamodels are created out of a set of crash-simulations. With the help of such analyses the models used for the simulations can be improved. For example instabilities can be found and explained or the needed material can be minimized under certain safety restrictions.

An important question in this context is: How good can these metamodels represent the reality? To answer this question, one can compare the crash-simulations to the real crash-tests, which were recorded by camera systems after the crash. To be able to compare the test-data with the LS-DYNA crash-simulations, we first need to convert the test-data by matching the geometries and transferring the part information from the simulation to the crash-test. Afterwards one can calculate the combination of the simulations, which approximates geometry and deformation behavior of the test-data as close as possible. The distance and difference in behavior between this calculated best fit and the actual crash-test can be used to measure the quality of the simulation model. Once the evaluation of the model is finished, the test-data can also be added to a robustness campaign as an additional simulation and used for further analysis.

This allows us to answer questions such as: How does the test fit into the simulation subspace? Which simulation runs are similar to the test for a certain crash event? Which of the dominating crash events found in the simulation can also be found in the test?

Thus, the described matching procedure combined with exemplary further analysis methods on the one hand allow for a quick and automated matching between test and simulation and on the other hand a more detailed validation of the simulation model in comparison to the actual test. Due to the conversion of the test-data, the same post-processors can be used for both the simulations and the test-data, resulting in a smoother workflow.

Equivalent Energy Method for Welding Structure Analysis

T. Loose, J. Rohbrecht (DynaWeld)

Simulation means real physical processes are calculated on computers with numerical methods. The real world is substituted by a virtual world. The benefits on welding are:

- predict the state before manufacturing
- complex high costly physical tests are replaced by low costly virtual tests,
- visualization of states of work pieces, which are not or hardly able to be measured,
- automation of analysis and evaluation, which cannot be realized by physical tests,
- explanation of formation processes as basis for the design of optimization tasks,
- training and education.

Simulation for welding starts before more than 20 years [1, 2, 3] and is meanwhile in use for all welding processes. Experience has to be elaborated for new materials, new processes or new design. Welding simulation helps to achieve this experience rapidly [4, 5].

Two topics are required for industrial applied welding simulations:
- The simulation has to be performed economically.
- The setup of the simulation has to be founded on process data.

The welding procedure specification (WPS) contains all data relevant for the welding process. This paper explains how to use this information as input for a welding

Large simulation models as well as thick parts with multilayered welds leads to a special challenge. Welding simulation for this kind of specimens becomes economic only under use of simplified approaches: the metatransient method for 3D or 2D models. The authors developed the equivalent-energy-method (EEM) which is based on the data from the WPS. This method postulates that the energy input per unit length in the simplified model is the same than in the detailed model. Additionally a temperature field calibration guarantees for the simplified method, that the heating of the area next to the weld, which is mainly driving the distortion, is estimated correctly.

Literature

[1] Schweiger, W.: Simulation von Schweiß- und Härte-Verfahren auf Basis des FE-Programmes SYSWELD, Bd. 936. In: VDI (Hrsg.): VDI-Berichte 936, 1992, S. 197–205

[2] Dilthey, U. ; Sudnik, W. ; Mokrov, O. ; Habedank, G.: MAGSIM for Windows: GMA pulse welding of fillet welds. TWI, 1998

[3] Dilthey, U. ; Dikshev, I. ; Mokrov, O. ; Pavlyk, V.: Software package Simweld for simulation of gasmetal-arc-welding prosesses of stells and aluminium alloys. In: Mathematical modeling of weld phenomena 7, pp 1057 .. 1080, TU Graz, 2005

[4] Brand, M. ; Loose, T.: Anwendungsgebiete und Chancen der Schweißsimulation. In: Schweiß- und Prüftechnik (2014) ÖGS Österreichische Gesellschaft für Schweißtechnik (Hrsg.), Nr.05-06, pp 138-142, Wien

[5] Loose, T. ; Boese, B.: Leistungsfähigkeit der Schweißstruktursimulation im Schienenfahrzeugbau. In: Vortragsband 10. Fachtagung Fügen und Konstruieren im Schienenfahrzeugbau, 2013, pp 61 - 67, Halle

Prediction of Residual Deformation from a Forming and Welding Procedure in Alloy 718 using LS-DYNA

E. Odenberger, L. Peréz Caro (Swerea);
M. Schill (DYNAmore Nordic)

Accurate prediction of the final geometry of a component is of major importance to the manufacturing industry. This is a complex task, especially if the manufacturing involves several process steps. In order to succeed, the complete manufacturing process has to be included in the simulation. Sheet metal forming procedures have been modeled for quite some time where forming, trimming and springback are included in the simulation. However, different manufacturing processes steps that affect the geometry are often involved in producing the final component.

In this paper, a forming and subsequent welding process is studied. The forming and welding simulation is done using shell elements with the novel material model *MAT_CWM.

The aim of the paper is to investigate how the different process stages affect the final geometry and compare simulation results from LS-DYNA with experimental observations.

Preliminary Study on Modeling of the Deformation and Thermal Behavior of FSW using SPH Approach

S. Patil, H. Lankarani (Wichita State University);
F. Baratzadehl (National Institute for Aviation Research)

Material flow in the solid-state Friction Stir Welding (FSW) is quite a complex process. Investigation of material flow can be carried out either by experimentation or by numerical simulation. However, compared to experimentation, numerical simulation is inexpensive, efficient and convenient, but quite challenging to model. The challenging issue in modeling FSW is to deal with the large deformations of the workpiece material. The Lagrangian simulations of FSW show that the severely distorted finite elements are caused due to the large deformation of the workpiece material, which makes the Lagrangian approach inappropriate for modeling FSW.

A good alternative is to study it in a SPH environment. SPH formulations are used to overcome the shortcoming of Lagrangian formulations due to its continuous regimes. The basic idea of the SPH approach is that the mesh is obliged to follow material flow. Thereby the excessively distorted elements can be avoided as in Lagrangian formulations. In this paper, we fulfill this aim by using a SPH method. We also intend to do some preliminary experiments about weld strength .An important consideration in applying the SPH approach is an advection method which determines the mesh motion in every step of the analysis.

Based on the simulation results, it is concluded that the material motion characteristics on the top surface and through the depth (volume) of friction stir welds have been made for the advancing and retreating sides. The motion trends are consistent with the reported published experimental evidence.

The present paper organized as following. First SPH modelling performed .After that thermal history validated with FE model. Temperature history data is in good agreement with FE model.

New Generation Modeler for LS-DYNA Material Parameter Conversion

H. Lobo, E. Strong, A. Beckwith (Matereality)

We describe a new software component that takes into consideration the unique multi-variate nature of LS-DYNA material models. Rate-dependent models require adjustment and tuning of many material parameters to fit the rate-dependent tensile properties. Drawing upon a robust back-end data model, a graphical user interface provides drag and drop capability to allow the user to perform tasks such as model extrapolation beyond tested data, modulus change, rate dependency tuning and failure criteria adjustment while assuring self-consistency of the underlying material model. Unit system conversions are also facilitated, eliminating error and ensuring that material inputs to simulation correctly reflect the intent of the CAE analyst. The utility of the Matereality CAE modelers is illustrated with examples for LS-DYNA material models MAT_019, MAT_024 and MAT_089 LCSR.

Experimental Investigation on the Damage Behavior of a Rubber-Toughened Polymer

M. Helbig (DYNAmore)

Explicit and Implicit FE Simulations of Material Tests for Subsequent Durability Analyses

P. Thumann, Prof. M. Wagner (OTH Regensburg);
B. Suck (BMW Group);
S. Marburg (TU München)

To improve the efficiency in sheet metal manufacturing, more and more press systems with increased stroke rates are in use. Due to the increased stroke rates, the structural-dynamic loads too increase. Thereby, not only the press systems are highly loaded, but also the tools for manufacturing the blanks. This has to be accounted for in the early design phase of the tools in the virtual process chain.

Because of safety requirements, high safety factors are used during the design of the tools, leading to high manufacturing costs. Furthermore, stroke rates are limited due to the movement of high masses of the tools, reducing the productivity of the press system.

To reach smaller allowable safety factors, resulting eventually in fewer costs, and to improve the stroke-limitation due to the moved masses, a simulation methodology for the evaluation of the durability of selected tool components is developed. Therefore, special material tests were designed and carried out. The tests have to be investigated with FE simulations for subsequent durability analyses.

Investigation on the Dynamic Behavior of AlgoTuf 400F Steel

G. Toussaint (Defence Research and Development Canada)

In order to improve finite element simulation predictions of a dynamic event such as a blast or a ballistic impact on a structure, the dynamic behavior of the materials involved has to be investigated. The information gathered from this investigation can then be further used to choose the constitutive material model as well as identified its parameters. In this paper, the main objective is to share the findings from this investigation for the AlgoTuf 400F steel.

The first section of the paper presents the quasi-static test that were performed by Defense Research and Development Canada (DRDC) as well as the split Hopkinson pressure bar (SHPB) tests that were performed at a strain rate between 103 and 104 s-1. These experimental data showed that at low strain rates, the material did not exhibit exactly the same behavior in the rolling direction (longitudinal direction) compared to the transverse one. It was also found that at higher strain rates, the effect of the manufacturing method on the properties through a 25.4 mm (one inch) thick plate could be neglected. Nevertheless, the material has showed sensitiveness to the strain rate and this was taken into consideration in the constitutive material model.

In the second section, the plasticity parameters identified for the simplified Johnson-Cook constitutive strength model obtained using these experimental data are presented.

The third section describes the 2D axisymmetric finite element (FE) model of the SHPB test and shows good agreement between numerical and experimental results. It is therefore possible in the last section to perform a parametric analysis to study the deformation response of an AlgoTuf 400F plate loaded by a spherical air blast, using the particle blast method.

The next step of this investigation will be to identify a constitutive damage/failure model and get its parameters to be able to predict accurately the deformation and damage/failure response of an AlgoTuf 400F steel plate subjected to a blast event.

Absorbing Materials – Tests Versus Simulations

R. Ridky, M. Popovic (SVS FEM);
M. Drdlova (Výzkumný ústav stavebních hmot);
O. Koutny (Bogges)

An absorption capacity of soft, viscoelastic materials at high strain rates is included in wide range of practical applications. One of the critical questions in any similar analysis is setup of material properties including all physical constants. There are many kinds of tests analyzing dynamic properties of absorbing elements.

In many cases testing and also numerical simulations are influenced by many types of measurement and simulation conditions. This investigation was focused on relations between particular testing results in their complexity and related numerical simulations as well. The goal of this investigation is to find a numerical material model of absorbing material suitable for explicit numerical simulation especially focused on blast load. Number of variations in real testing is usually strictly limited because of experiment costs. On the contrary numerical simulations have no limits in number of analyzed variants but they have handicap in correlation with reality. This should be solved by combined development of requested products. This analysis used four kinds of real tests with different strain rates as consistent and verified points that numerical models should confirm and fill in unknown gaps between them.

Numerical Modelling of the Plastic Deformation of Ti-6Al-4V Sheets Under Explosive Loading

D. Kakogiannis, F. Coghe, L. Rabet (Royal Military Academy)

The subject discussed in the present paper is the numerical modelling conducted during the design process of an experimental setup that is used to investigate the response of circular Ti6Al4V sheets under impulsive loading. A multiscale numerical modelling method to predict the plastic deformation of the material is presented focusing on the definition of the parameters of the Cazacu-Barlat model (*MAT_233/*MAT_CAZACU_BARLAT) as implemented in LS-DYNA. The definition of the parameters is linked to the microstructure of the material through the viscoplastic self-consistent polycrystal model(VPSC7) of Lebensohn and Tomé using the optimization method of Simulated Annealing. A comparison is made between the direct input of the parameters and the parameter fit, provided by LS-DYNA.
The method is validated with experiments where sheet specimens of two thicknesses are tested using an experimental setup with which a planar blast load is applied and with high speed cameras and the digital image correlation (DIC) technique, strains are measured in real time on the specimens.

Review and Advances of Coupling Methods for the ICFD Solver in LS-DYNA

F. Del Pin, I. Çaldichoury, R. Paz (LSTC)

The ICFD solver in LS-DYNA specializes in the solution of Incompressible fluid flows. The main goal is to accurately predict the values of pressure and velocity subject to the constraint that Div(v)=0 where v is the fluid velocity. One area of applications is the prediction of lift and drag in aerodynamics of external flows, detachment or recirculation for internal flows, etc. These are what could be regarded as typical CFD applications. The real potential of the ICFD solver arises when it is coupled to other solvers within LS-DYNA.

In this paper the numerous existing coupling methods available for the ICFD solver will be reviewed. Also in this paper the latest advances regarding to coupling methods will be presented. In terms of coupling ideas the following will be discussed: Fluid Structure Interaction (FSI) weak and strong, Conjugate heat transfer weak and strong and the addition of electromagnetism as part of the coupled solution. Also the coupling with the Discrete Element Method (DEM) will be introduced enumerating the different features available in the coupling process. Finally a new variation that incorporates the steady state solver coupled to a static linear or non-linear structural solution will be presented.

Applications of ICFD / SPH Solvers by LS-DYNA to Solve Water Splashing Impact to Automobile Body

G. Wang, E. DeHoff (Honda R&D Americas);
F. Del Pin, I. Çaldichoury, E. Yreux (LSTC);
K. Gardner (Ohio State University)

When a vehicle runs at high speed on a watery or muddy road, the high speed splashing generated by tire rotation often causes damage to the underbody panels. This is a challenging dynamic FSI (fluid-structure interaction) problem for Honda to consider. ICFD and SPH are two powerful solutions used in FSI applications, especially for high speed flow and with dynamic free surface evolution. This paper presents some FSI solutions of water splashing of automotive components by using ICFD and SPH solvers from LS-DYNA:

(1) Aluminum underbody panel water impact. The panel is impacted by a water balloon shot out of a water cannon that simulates a large water splash contacting the panel. Results with ICFD and SPH solvers are discussed and compared with the corresponding test data from the Ohio State University in Columbus, OH, USA. In brief, the ICFD results of peak displacement/force on the panel caused by the impact agree well with the tests while the SPH method over predicts the load/displacement.

(2) Water splash impact on vehicle. In this study, a model of a half CBU runs at 45 mph into a 0.475 m high water layer of water. The vehicle bottom surface is set up as a deformable structure while the other parts are rigid. It is shown that the SPH method can easily simulate the realistic water splashing caused by the moving tires and vehicle body, whereas the ICFD solution has difficult time arriving at a solution and also takes much longer to solve.

Hydrodynamic Drag Force Predictions for Amphibious Military Vehicles

I. Kurtoglu (FNSS Savunma Sistemleri)

Amphibious military vehicles are very important in the battle field due to their flexible operating environment. In general, amphibious military vehicles are designed for water operations in still lakes and rivers with low current speed. There are also some examples of amphibious military vehicles operating in open water environment with harsh sea state conditions. Having this operational flexibility makes the amphibious military vehicles strategically important for armies.

However, making a military vehicle amphibious brings challenging problems especially in the design stage. Drag force prediction is obligatory for the thrust requirement determination. High drag forces mean high thrust requirements which may also affect the selected thrust system. Due to high thrust demand, the system may be bigger, and hence heavier, and this weight increase may affect the amphibious performance of the vehicle. Therefore, estimating the drag forces for defined amphibious operation conditions is very important.

Also, optimizing geometrical form of the vehicle according to the amphibious capability of the vehicle becomes more important. In this study, the drag forces on an amphibious military vehicle at different operation velocities are predicted using Incompressible Computational Fluid Dynamics (ICFD) solver implemented in LS-DYNA.

In the simulations, the vehicle model is scaled according to the Froude number used in the verification tests. The test scale is limited by the pool dimensions. The drag tests are performed in Istanbul Technical University Naval Architecture and Design Faculty laboratories. Drag force predicted from the simulations for a predefined speed is compared with the one obtained from the pool tests. The accuracy of the drag force estimated from simulations is reviewed for different solution parameters of LS-DYNA ICFD solver.

An Investigation into Modeling Approaches for the Dynamic Response of a Shipping Container Cart and Suspended Automotive Parts under Random Base Excitation using LS-DYNA

Prof. S. Noll, A. Ramanathan (Ohio State University);
E. DeHoff, R. Rittenhouse (Honda R&D Americas)

Shipping containers are exposed to complex dynamic loading conditions during transport via truck, rail, and air. The loading conditions are further complicated by contact nonlinearities between the cart and ground, between cart holders and suspended parts, and between neighboring suspended parts. This study focuses on developing a modeling strategy to simulate a container cart with parts undergoing ASTM D4728 truck random vibration tests.

Initially, the linear frequency-domain properties and response to a random excitation profile of the cart structure were examined and correlated with experimental measurements. The linear response predictions lacked the required modeling fidelity to capture the nonlinear dynamic behavior observed during the testing as the input power spectral density profile was increased to higher levels. Thus, the nonlinear response was then simulated in the time domain using the explicit integration method in LS-DYNA. The use of LS-DYNA also allowed the simulation of the complex contact-driven boundary conditions. The total run time was determined to be prohibitively long for the second approach as the time step size was 0.3 μsec for the required simulation duration of more than 5 sec.

Finally, a dynamic substructuring strategy was employed through the use of super elements. This final approach captured the dynamic amplification of the cart, maintained the contact nonlinearities, and reduced the total run times. This modeling approach in LS-DYNA appears promising to capture the complex dynamic loading conditions affecting shipping containers in a computationally efficient manner.

Application of Model Order Reduction Techniques in LS-DYNA

P. Friedrich, M. Thiele (SCALE);
D. Weigert (Audi);
U. Reuter (TU Dresden)

Hierarchical Multi-Level-Optimization of Crashworthy Structures using Automatic Generated Submodels

H. Singh, Prof. A. Schumacher (Bergische Universität Wuppertal); C. Falconi, A. Walser (Automotive Simulation Center Stuttgart); S. Trentmann, L. Benito (Iges. Für numerische Simulation); H. Müllerschön (SCALE); C. Foussette, P. Krause (divis intelligent solutions)

The optimization of large crashworthy systems like a vehicle body in a crash loaded case is a time consuming and costly process. Extensive computer simulations are required to improve the crashworthiness of such large systems using an optimization technique. The simulation time can be reduced by dividing the large system into small subsystems also known as submodels. The submodels are used in the optimization to shorten the response time. But the generation of the submodels by hand is a challenging process and it requires a huge effort to validate them. This paper deals with the structural optimization of large crashworthy systems with a hierarchical Multi-Level-Optimization method using automatically generated and validated submodels. The process of automatic generation of submodels is developed using the so called connecting island algorithm [1]. The algorithm is developed in Tcl language using the software Generator [2] and Animator [3]. The two important parameters in this process are the threshold ratio and the connecting island value. These are based on an evaluation function which is a structural response of the large system with time averaging and space averaging method. The size of the submodel depends on these two parameters. Evolutionary algorithms are applied to optimize the time required to generate an appropriate submodel size with help of ClearVu Analytics [4] software. The process of automatic validation of submodels is based on three methods, so called local, global and response validation method which are discussed in detail. The Multi-Level-Optimization process is carried out in two hierarchical levels. The level 1 is a large system and level 2 is a submodel of the large system. Each level has different design variables, constraints and structural responses. The levels are coupled together using interface boundary conditions in form of nodal displacements. These boundary conditions are extracted from the large system and mapped onto the subsystem in an automatic process. If the boundary conditions are exact, the submodel region will deform identical to the deformation of this region in the large system [5]. The update of these boundary conditions and the correlation between the two levels is studied and discussed in detail. The research is demonstrated on an academic example of a cantilever frame impacted by a rigid sphere and on an industrial application of a Toyota Yaris front crash [6]. The future possibilities of the Multi-Level-Optimization method and the submodeling technique are shown in the paper.

Acknowledgement

This research was supported by the "Bundesministerium für Bildung und Forschung (BMBF)" within the scope of the KMU-innovativ research project "Entwicklung von Softwaremethoden zur effizienten Ersatzmodell gestützten Optimierung für die Crashauslegung im Fahrzeugentwicklungsprozess (eEgO)". Beside the University of Wuppertal, Automotive Simulation Center Stuttgart e.V. , SCALE GmbH, GNS mbH, divis intelligent solutions GmbH and the Technical University of Munich are involved in the project.

Literature

[1] Falconi D., Carlos J.; Walser, Alexander F.; Singh, H.; Schumacher, A. (2017): "Automatic generation, validation and correlation of the submodels for the use in the optimization of crashworthy structures", submitted to WCSMO12 (2017)

[2] The multipurpose Pre-Processor for FEM Analysis, GNS-mbH Gesellschaft für numerische Simulation (n.d.), retrieved from http://gns-mbh.com/generator.html

[3] The Trendsetting Post-Processor for FEM Analysis, GNS-mbH Gesellschaft für numerische Simulation. (n.d.), retrieved from http://gns-mbh.com/animator.html

[4] ClearVu Analytics, divis intelligent solutions GmbH, http://www.divis-gmbh.de, (2010-2017)

[5] Link, S; Singh, H.; Schumacher, A. (2016): "Influence of submodel size and evaluated functions on the optimization process of crashworthiness structures", LSDYNA Forum (2016)

[6] LS-DYNA-Model TOYOTA YARIS MODEL (NCAC V01): "National Crash Analysis Center (NCAC)", http://www.ncac.gwu.edu/vml/models.html (2010

Workshop on the Introduction to the New Optimization Tools for Forming Simulation with eta/DYNAFORM

M. Merten (DYNAmore)

Recently, DYNAFORM version 5.9.3 was published, which contains several new features. Particular highlights are the new automatic springback compensation process or the reworked blank & trim line development, which improve the workflow of a toolmaker in daily life.

The goal of this workshop is to present these new features and how they work. How they can be an applied will be shown with the aid of a live demonstration on a simple geometry. Attention is drawn on a quick and proper setup for these processes.

Wednesday Afternoon Sessions

17:25 – 18:40

Development of a 2015 Mid-Size Sedan Vehicle Model

R. Reichert, S. Kan (George Mason University)

A detailed finite element model of a 2015 mid-size sedan vehicle was developed using a reverse engineering process. The model consists of about 1000 parts and 2.25 million elements representing geometry, connections, and material characteristics of relevant structural and interior components. This paper describes the level of detail of the simulation model, the validation process, and how it performs in various impact configurations when compared to full-scale crash test data. Members of the Center for Collision Safety and Analysis (CCSA) at George Mason University, formerly known as the National Crash Analysis Center (NCAC), have been developing a fleet of finite element vehicle models over the last 20 years.

The updated mid-size sedan presented in this paper is the latest model with a high level of detail using state-of-the-art modeling techniques. A thorough validation process, using test results from frontal, side, and roof crush impact configurations, ensures a high level of correlation for a variety of load cases. Special focus has been placed on occupant compartment intrusion and vehicle pulse evaluation for frontal impact scenarios. Realistic wheel kinematics in the Insurance Institute for Highway Safety (IIHS) small overlap load case was achieved through adequate failure modeling. An objective correlation analysis tool was used to evaluate how well simulation results match respective test results.

The model is currently being used for several research studies, including the development of structural countermeasures to significantly reduce occupant compartment intrusion for the National Highway Traffic Safety Administration's (NHTSA) left and right frontal oblique offset configurations.

Small Electric Car Front Cross-Member Assembly Low Speed Impact Simulation

Prof. G. Lampeas, I. Diamantakos, K. Fotopoulos (University of Patras);
I. Lopez Benito (Batz S.)

In the frame of EVolution project a small electric car has been developed. One of the demonstrators of car structure design is the front cross-member assembly. In the present work the numerical simulation of the front cross-member assembly low speed impact is presented. The crash is considered according to Allianz test protocol.

The basic features of the front cross-member assembly structure comprise the application of foam type materials (more specifically low density poly-urethane foams) and crash boxes design, which are assembled behind the transversal beam; both features aim at the maximum possible impact energy absorption during a crash event.

FE models of all assembly parts are built and the FE model of the whole front cross-member assembly is constructed applying proper contact definitions and initial conditions. The developed FE model is solved using LS-DYNA explicit FE code. Strain-rate depended material properties are utilized for the foam materials. Numerical simulation results concerning structural deformations, absorbed kinetic energy and force applied on the vehicle during impact evolution are drawn and presented.

For the validation of the numerical model a physical test according to Allianz test protocol has been performed. Numerical simulation results compare well with experimental test measurements, leading to successful validation of the developed FE model.

Airbag Folding with JFOLD – Latest Developments and Case Studies

R. Taylor (ARUP); S. Hayashi (JSOL)

JFOLD is a software tool that helps users perform simulation based airbag folding in LS-DYNA. Today's airbag deployment analysis demands accurate folding of complex designs, but this is often a very time consuming process requiring expert input. JFOLD's continuous development focuses on making the process simpler and quicker and to give the non-expert access to complex folding techniques. This is achieved through three core elements: intuitive graphical interface, built-in customisable tool libraries and realistic, state of the art examples and tutorials.

Our presentation will describe a driver's airbag case study of two folding patterns with deployment validation using CPM. Two methods of folding a passenger airbag will also be shown, including and a novel way to quickly flatten the bag from 3D to 2D.

To further reduce airbag development cycle times and cost, a new airbag morphing application is under development to help the user optimise the design in a virtual environment. This and other new features will be presented.

Curve Comparison using esiCORA

M. Sommer, M. Seshadri (ESI)

CAE simulation is essential for the development of new vehicles and safety systems. The difficulty is to assure that simulation results corresponds to the real testing results. The common practice is to compare the time history results graphically. Therefore, this comparison should be accurate and credible.

The CORA rating developed by PDB calculates the correlation between two unique curves. The result is a number between zero and one, whereas the larger the value the better is the correlation.

CORA combines two different methods to assess the correlation of the curves. Corridor method and Cross correlation method are used within a specified time range interval in order to compensate for the disadvantages of each of these methods alone.

Visual-Viewer is the Visual-Environment integrated post-processing tool for CAE applications, providing dedicated plotting solutions where CORA rating is embedded. In Visual-Viewer, the CORA rating can be computed using the following options:

1) esiCORA computation for the selected curves by right click method:
   a) "esiCORA Value", will compute the CORA value and attach it to the selected curve
   b) "esiCORA Report" will generate the Corridor, Cross Correlation rating, Intervals limits for the selected simulation and reference curves

2) esiCORA rating computation for several curves:
   This method allows to use the global CORA computation parameters and create a full CORA report. For repeatable tasks the Load Case Table allows to compute multiple load cases which will be added for CORA rating report computation.

Numerical Modelling of the Fluid Structure Interaction using ALE and SPH: The Hydrodynamic Ram Phenomenon

D. Varas, J. A. Artero-Guerrero, J. Pernas-Sánchez, J. López-Puente (University of Madrid)

Vulnerability against high-velocity impact loads is a critical issue for the design of aerospace structures due to the fact that aircrafts can be subjected to different types of loads during their service life which may cause a catastrophic failure. Bird strikes, hailstones, runway debris or even the ice released from the edge of a propeller blade may impact different parts of the fuselage. Wings represent the largest exposed area to impact threads of all the vulnerable components of an aircraft, therefore impacts onto a fuel tank inside the wings are considered of special relevance in aircraft vulnerability.

Hydrodynamic Ram (HRAM) is a phenomenon that occurs when a high-energy object penetrates a fluid-filled container. The projectile transfers its momentum and kinetic energy through the fluid to the surrounding structure increasing the risk of excessive structural damage leading to a catastrophic failure. HRAM consists of four principal stages: shock, drag, cavitation and exit. Each stage contributes to structural damage in a different way and extent. The study of the HRAM phenomenon is not only important for the aircraft industry. High velocity impacts on fluid filled containers are of great interest for different industrial fields such as safety of industrial facilities or road haulage. In those cases, an impact in the vessel may produce the failure of the tank and serious consequences on the environment or even toxic and flammability effects.

This work shows the numerical modelling developed to reproduce the effects of the HRAM phenomenon in different fluid filled square tubes (aluminum and CFRP) when impacted by steel spherical projectiles at different velocities and filling levels. The simulations are performed with the finite element code LS-DYNA using two different techniques for the fluid phase: the ALE and SPH formulations. Experimental tests providing the pressure in different points of the fluid, deformation of the walls and cavity evolution are compared with the numerical results in order to assess the validity and accuracy of both ALE and SPH techniques in reproducing such a complex phenomenon. In addition, the numerical results revealed that ALE is the most appropriate technique to simulate HRAM attending to its computational cost.

The numerical model validated has contributed to a better understanding of the phenomenon as well as to study some possibilities to attenuate the effects of the HRAM on the affected structures. With the aim of reducing both the pressure wave generated by the impact and the cavity growing, thin sandwich structures with two metallic skins and a core of air have been located in different positions inside the fluid filled tube. The results show improvements regarding the vulnerability of the fluid filled impacted tubes.

Novel Simulation of Composite Material behavior Subjected to Hyper-Velocity Impact (HVI) and Produced Secondary Debris by using Smoothed Particle Hydrodynamics Code (SPH) Methodology in LS-DYNA

E. Giannaros, Prof. A. Kotzakolios, S. Tsantzalis, V. Kostopoulos (University of Patras);
G. Campoli (ESA /ESTEC)

Present paper deals with the simulation of hypervelocity impact response of composite fiber reinforced polymer material (CFRP) and produced secondary debris using SPH methodology in LS-DYNA. Ballistic limit, crater diameter and secondary debris distribution are evaluated.

Isogeometric Models for Impact Analysis with LS-DYNA

M. Montanari, N. Petrinic (University of Oxford);
L. Li (LSTC)

The advent of isogeometric analysis (IGA) opened new horizons for reducing design and optimization costs. By employing the same mathematical formulation to describe CAD and simulation models, IGA integrates design and analysis into a new paradigm. Our work is part of a technology validation effort that aims to assess IGA for the analysis of impacts.

Case studies of (i) wave propagation, (ii) model calibration and (iii) ballistic impact are presented to compare the finite element analysis (FEA) against IGA.

The IGA element formulation results more expensive than traditional FEA; however, it captures travelling stress waves more accurately. This improves significantly our ability to predict the dynamic response of systems, for example, undergoing high strain-rate loadings. Unfortunately, the mathematical formulation which makes IGA more accurate does not allow discontinuities in the displacement field. This prevents, for example, a projectile to penetrate a target plate. It is argued that the physical failure can be predicted by using material models combining strain-rate-sensitive failure criteria. From an impact engineering perspective, these case studies underpin the analysis of turbine fan blades and their containment casing. In order to reduce the costs of analysis, this work exploits IGA analysis-ready models that cut down the meshing costs, and LS-DYNA parallel computing capabilities.

Testing in Support of the Development of Accurate Numerical Simulations of Plastic Deformation and Failure

A. Gilat, J. Seidt (The Ohio State University)

Testing in support of the development of materials models in numerical simulations consists of material characterization tests and validation tests. Testing at various strain rates, temperatures, and loading conditions is used for characterizing plastic deformation and failure of materials. The data from the tests is used for developing constitutive equations (material model) that are utilized in numerical codes that are used for simulations of practical applications.

Emphasis in the present paper is on the significance of the Digital Image Correlation (DIC) method for measuring full-field deformations and the development of new tests. It includes the use of DIC in Split Hopkinson (Kolsky) Bar (SHB) tests (compression, tension and torsion), a special apparatus, consisting of a hydraulic actuator and a very long transmitter bar, for tests at intermediate strain rates (50 s-1 –200 s-1), tensile tests with DIC at elevated temperatures (up to 850°C), and simultanious full-field deformation and temperature measurement in tensile tests at low and high strain rates.

Many of these tests have been used during in the development of the deformation and failure model (MAT224) in LS-DYNA. The model is based on experimental determination of a failure surface that gives the equivalent plastic strain to failure as a function of stress triaxiality and the Lode parameter. In validation tests material specimens or components are subjected stress states that are different than the ones used for determining the material models. The tests are simulated and predictions of loads, deformation and failure are compared with measurements. Two examples of validation tests, the punch test and the spot weld test, are presented.

Application of Digital Image Correlation to Material Parameter Identification using LS-OPT

N. Stander (LSTC);
K. Witowski, A. Haufe, M. Helbig, D. Koch, C. Ilg (DYNAmore)

Digital Imaging Correlation (DIC) is an optical method which provides full-field displacement measurements for mechanical tests of materials and structures. It can be used to obtain temporal displacement, deformation or strain fields from an experimental coupon and can be combined with Finite Element Analysis to identify the constitutive properties of a material [1].

Because DIC-based parameter identification is an inverse process, optimization is used to obtain the parameters which will minimize the discrepancy between the measured and the computed fields. The Mean Squared Error functional typically used [1] is: $f(x)=\sum_{j=1}^{n} \|\varphi_j(x)-\tilde{\varphi}_j\|^2$ where $\varphi_j(x)$ is a vector of nodal displacements/strains (for computation and experiment) at a number of observation points and n is the number of observation states. The functional can be augmented to incorporate global force-displacement measurements or any other functional resulting in parameter identification based on DIC.

As part of this research, the DIC methodology was implemented in the LS-OPT code utilizing the following main features: (i) multi-point responses and histories, (ii) the alignment and automatic mapping of points to an FE model. A graphical interface is provided for importing ARAMIS/GOM® (a popular commercial optical test package) test results.

As a first test example, a parameter identification problem was constructed using a flat tensile bar with a hole in which the deformation field history was used as "test" input to recover the original parameters. Further parameter identification examples will be presented at the conference.

Literature

[1] Mahnken, R., Stein, E. Parameter Identification for Finite Deformation Elasto-Plasticity in Principal Directions, Comput. Methods Appl. Mech. Engrg, 147, 17-39 (1997).

Numerical Investigations on Ricochet of a Spin-Stabilised Projectile on Armour Steel Plates

M. Seidl, T. Wolf, R. Nuesing (ISL)

Expanding urbanisation poses new challenges. Developing countries are facing a rapid Urbanisation with an estimated 4.3 billion urban inhabitants by 2035 [1]. History has shown that conflict and large-scale disasters are likely to occur where large populations reside [2]. During peace-keeping missions, soldiers move in closed urban locations surrounded by house walls and close to their armored vehicles. In event of combat, the likelihood of projectiles ricocheting in a closed environment—than on the open battlefield—arises.

This study examines projectile ricochet, which is the deflection off a surface of a projectile from its original trajectory after striking the target at a low angle [3], with focus on the influence of the angle of attack αt on the projectile behavior after ricocheting. Numerical simulations—using the explicit Lagrangian solver—are used for a qualitative investigation, as measurement precision limits the determination of the influence of αt.

Literature

[1] S. Collins, "Challenges of Urban Cities in 2035 - NATO Needs to Act Today to Prepare the Alliance for the Future," Allied Commander Transformation, 30 09 2015. [Online]. Available: http://www.act.nato.int/volume-4-urbanization-the-future-challenge-for-nato. [Accessed 04 01 2017].

[2] P. Piper, "A Concept for Future Military Operations on Urbanized Terrain," Department of the Navy Marine Corps Combat Development Command, Quantico, VA, 1997.

[3] M. Jauhari, "Bullet Ricochet from Metal Plates," J. of Criminal law and criminology, pp. 60(3):387-94, 1970.

Numerical and Experimental Investigation of Asymmetrical Contact between a Steel Plate and Armor-Piercing Projectiles

T. Fras (French-German Research Institute of Saint-Louis);
P. Pawlowski (Polish Academy of Sciences)

The study presents the discussion on modeling of interactions between armor-piercing (AP) projectiles and add-on perforated plates realized by the experimental investigation, numerical finite-elements simulation and the simplified modeling based on the integration of the equations of motion of a rigid projectile.

It is known that relatively thin steel plates with an array of holes, i.e. perforated plate, are efficient against impacts of small-caliber projectiles, which makes them applicable for light-weight armored vehicles as passive add-on armors. A number of holes in such plates increases the probability of asymmetrical contact between the plate and a small-caliber projectile, which causes its destabilization or fragmentation. Depending on the hit-point, AP projectiles behave differently; they may be strongly rotated if hit inside a hole, while hitting an area between holes, the damaged projectile core deviates from the initial trajectory. Impact on a hole-edge is the most efficient case of reducing the projectile's perforation capacity, as its core may be broken or shattered. Due to the extensive, experimental ballistic testing performed on the bainitic, slotted plates impacted by 7.62 x 51 .308 Winchester AP projectiles, the failure modes of bullets were analyzed and constituted a basis for the numerical model validation. The reference numerical, Lagrangian model was implemented in LS-DYNA by 8-node constant-stress solid elements with one integration point and stiffness-based hourglass control. The modeling of impacts confirmed dependence between the projectile failure and the hit point.

The localized-interaction model based on the integration of the equations of motion of a 6 DOF rigid projectile may be considered as a simplified method in the design of perforated pre-armors, in which obtaining the largest projectile distraction is the optimization aim. In such a robust model, the projectile-plate interactions are simplified and based on the integration of stress components, which are normal and tangent to the projectile's contact surface. The target plate and projectile are discretized by a spatial grid of points, at which interaction forces are calculated.

The performed numerical and experimental modeling proves a high protective efficiency of perforated steel plates against small-caliber AP projectiles. The asymmetrical contact between them results in a significant reduction of the bullet energy. The discussed methodology leads to improvements in the light-weight armors design.

Numerical Study of High Velocity Impact Response of Vehicle Armor Combination Using LS DYNA

G. Başaran (FNSS Savunma Sistemleri);
Prof. E. Gürses (Orta Doğu Teknik Üniversitesi)

The aim of this work is to perceive if the outcome from a ballistic impact can be predicted beforehand with the help of material testing and finite element simulations. Use of refined numerical simulation is gaining more importance especially in extreme load cases. A numerical investigation of the ballistic performance of monolithic, double layered metallic plates made of either steel or aluminum or combinations, were impacted by a 7.62-mm APM2 projectile at a velocity of 820m/s. The numerical models were developed using the explicit finite element code LSDYNA®. The effect of different metallic parts- thickness on the residual velocity of APM2 projectiles is examined. Three configurations of plate arrangements with different total thicknesses were used. Both aluminum target and projectile have been modelled as deformable bodies with Modified Johnson-Cook material model based on input parameters from literature. The predicted values of residual velocities were compared with the literature and a good correlation was found between the two.

Applications of ICFD Solver by LS-DYNA in Automotive Fields to Solve Fluid-Solid-Interaction (FSI) Problems

G. Wang, P. Rodriguez, J. Tippie, S. Smith (Honda R&D Americas);
F. Del Pin, I. Çaldichoury (LSTC)

Fluid-solid interaction (FSI) becomes a more and more important and wider application field in the automotive industry. It is challenging to CAE engineers to predict performance of complicated systems under high speed flow and with dynamic free surface evolution. This paper presents the three following FSI accomplishments in the power sports field by using ICFD solver from LS-DYNA:

(1) Gear box oil flow driven by high spinning speed shafts (Fig. 1). The goal of this work was to develop a CAE method to predict if oil in the gear box leaks out of the fuel hole at three gear box positions (00,150,400) and two rotation speeds (1060 and 3320 rpm). The CAE results completely comply with the test data.

(2) Thermal flow in muffler (Fig. 2). This CAE work was to investigate how the exhaust heat flow from the engine causes a temperature transfer along the muffler surface. Anisotropic porous modeling is used to model the flow through the catalytic converter and the CAE results are validated, respectively, by (i) the test flow rate data under constant temperature and (ii) the exhaust conjugate heat transfer test data. The results show good correlation.

(3) Windshield buffeting (Fig. 3). This project was done to explore if the window and roof of a side-by-side MUV would fail by wind force.

Simulation of Flow Induced Vibrations in Pipes using the LS-DYNA ICFD Solver

M. Timgren (DYNAmore Nordic)

Flow induced vibrations (FIV) is an important phenomenon to understand for m-shaped jumpers which are used in offshore applications. To understand how the FIV affects the life time of a m-shaped jumper FSI simulations have been used. Many papers have investigated two-phase flow in m-shaped jumpers and performed FSI simulations to understand how the flow affects the movement of the pipe. No or little information is available on how big impact the two-phase flow has compared to the case with one fluid.

This paper will perform FSI simulation of a m-shaped jumper with one fluid in LS-DYNA, water, and compare the results with other studies with two-phase flows.

Free Fall Movement Decomposition of a Payload Released by Aircraft: Study of the Aerodynamic Coefficients using the LS-DYNA ICFD Solver

E. Grippon, M. Seulin, V. Lapoujade, T. Maillot, C. Michel (DynaS+)

Through its multiphysics aspect, the airdrop domain is chosen to demonstrate the new FSI capacities recently developed within the LS-DYNA software (more specifically using the ICFD solver). To do this, DynaS+, French, Spanish and Portuguese distributor of LS-DYNA and associated services got a research and innovative subvention from the French Government, RAPID financing.

The main goal of this project, PARAFLU, is to succeed in modeling the complete sequence of an airdrop, which includes three main phases:

i. the payload freefall,
ii. the deployment of the hemispherical parachute,
iii. and the gliding phase when the payload is supported by the parachute sail.

Each step includes aerodynamic and fluid-structure interactions problems more or less complex. This paper focuses on the payload freefall phase modelling.

The falling movement of the payload released by the aircraft has been decomposed into simple motions. The package undergoes translation, rotation and combination of both, for Reynolds number ranging from 104 to 106. Similar cases from the literature have been reproduced in order to validate the ICFD solver capabilities. The aim is to evaluate its accuracy in reproducing aerodynamic phenomena such as the aeronautical coefficients observed in these configurations.

Particular attention has been paid in order to optimize the associated calculation times. Different models of turbulence has been studied.

TB11 Test for Short W-Beam Road Barrier

K. Wilde, S. Burzyński, D. Bruski, J. Chróścielewski, W. Witkowski
(Gdańsk University of Technology)

In this report we investigate numerical crashtests performed on short section of w-beam road barrier. In typical in-situ test, 60 m section of barrier is tested, while in real life application, shorter sections are often employed. In our numerical analysis, 20 m long section is simulated, hit by 900 kg car at 20°. angle. Total number of 31 simulations were performed, giving insight into cars behavior and ASI parameter value with variation to initial place of contact along the barrier.

Simulation of Wire Rope Road Barriers and Vehicle Collision: Experiment and LS-DYNA Correlation

I. Karpov, I. Demiyanushko, B. Tavshavadze
(Moscow Automobile and Road Construction State Technical University (MADI)

Wire rope road barriers are relatively new type of road barriers for Russia. This paper describes computational modelling of wire rope barrier parts (wire ropes, posts with soil) and behavior of the entire barrier under vehicle impact conditions according to Russian regulations. Different cable constructions and models were examined and compared between each other. For validation of all model parts, tests with usage of special equipment were conducted. Additionally, full-scale crash test was performed for the entire barrier, and experimental data were compared with computational results. All models showed good coincidence with the experimental data.

On the Influence of Shell Element Properties on the Response of Car Model in Crash Test

S. Burzyński, K. Wilde, D. Bruski, J. Chróścielewski, W. Witkowski
(Gdańsk University of Technology)

In this report we address the issue of selection of control card values and its influence on the behavior of the LS-DYNA car model in collision test. It is shown that the shear factor coefficient plays substantial role on the overall course of the simulated event.

Workshop on Mapping with ENVYO

C. Liebold (DYNAmore)

ENVYO is a multi-purpose mapping tool which was introduced to the public in 2016 during the German LS-DYNA Forum

The goal of this workshop, is to present the already implemented mapping capabilities and to demonstrate their usage. The general need to map simulation results is shown with the aid of dedicated examples.

The workshop is closed with an open discussion where you can place your own ideas for future mapping developments.

Thursday Morning Sessions

8:30 – 10:10

Mechanical Modeling of Li-Ion Cell Crush Experiments using LS-DYNA

M. Seulin, C. Michel, V. Lapoujade (DynaS+);
J. Marcicki (Ford Research and Innovation Center);
P. L'Eplattenier (LSTC)

Electric vehicles (EVs) and hybrid vehicles (HEVs) sales have grown spectacularly in the last few years. Safety has become an increasingly pressing issue in large-format, energy dense Li-Ion batteries used in electric and hybrid cars. The anticipation of the response to abuse conditions becomes a critical factor in designing optimized systems. Internal short circuit is one of the most dangerous scenario and has been the root cause of several catastrophic accidents in recent years.

A 3D electromagnetic (EM) model has been developed in LS-DYNA which, coupled with the mechanical and thermal solvers, allows to simulate battery cells in normal use as well as during abusive scenarios when the structure of the battery is damaged (as a result of a car crash for example). The calibration of the electromagnetic model and the multi-physics coupling first assume the mechanical part of the battery crash to be properly represented. As a consequence, the first step of the corresponding on-going project, presented in this paper, only focus on the mechanical part.

Taking into account the very low thicknesses of the different layers, the challenging part and modeling difficulty mainly comes from the computing resources necessary to run the calculation. Thereby it is fundamental to make appropriate and smart modeling choices and use available options in LS-DYNA enabling to reduce the total computational time while ensuring a correct cell behavior.

A typical cell unit has been studied both in bending and indentation behaviors so to make appropriate modeling choice (element formulation, aspect ratio…) and then minimize the computation time.

Battery Abuse Simulations Using LS-DYNA

P. L'Eplattenier, S. Bateau-Meyer, I. Çaldichoury (LSTC)

Safety is an important functional requirement in the development of large-format, energy-dense, lithium-ion (Li-ion) batteries used in electrified vehicles. Computer aided engineering (CAE) tools that predict the response of a Li-ion battery pack to various abusive conditions can support analysis during the design phase and reduce the need for physical testing. In particular, simulations of the multiphysics response of external or internal short circuits can lead to optimized system designs for automotive crash scenarios.

Recently, a so called "distributed randles circuit" model was introduced in LS-DYNA in order to mimic the complex electrochemistry happening in the electrodes and separator of lithium ion batteries. This model is based on electrical circuits linking the positive and negative current collectors reproducing the voltage jump, internal resistance and dumping effects occurring in the active materials. These circuits are coupled with the Electromagnetics (EM) resistive solver to solve for the potentials and current flow in the current collectors and the rest of the conductors (connectors, busses, and so forth). The EM is coupled with the thermal solver to which the joule heating is sent as an extra heating source, and from which the EM gets back the temperature to adapt the electrical conductivity of the conductors as well as the parameters of the Randles circuits. One of the advantages of the Randles circuit model is the relative easiness to introduce internal short circuits by just replacing the Randles circuits in the affected area by a short resistance. The Randles circuit model also is coupled with the mechanical solver of LS-DYNA where the deformations due to a battery crush allow the definition of criteria to initiate internal shorts.

Up to now, the Randles circuit model was only available on solid elements. So the user had to create a mesh with all the layers of a cell (positive current collector, positive electrode, separator, negative electrode, negative current collector, negative electrode, separator, and so forth), and define segment sets of each of the current collectors to connect them by Randle circuits in a distributive way. The construction of the case was cumbersome and the mechanical solver often presented limitations on solid elements with one very small dimension compared to the other ones due to the extreme thinness of the different layers. The mechanical deformation of a succession of stiff layers (current collectors) and very soft ones (electrodes and separators) also was a challenge.

Recently, the Randles circuit model has been extended to composite Tshells. These new elements promise a better and faster mechanical resolution, and make the setting of the simulations much easier for the user.

Finally, a new battery packaging environment is being developed in LS-PREPOST helping with the setup of cell or multi-cell cases.
In section 2, we will present the new battery cell model on composite Tshell and give some first example of internal short simulations, and in section 3, we will present the new battery packaging environment in LS-PREPOST

Modeling of a Cast Aluminum Wheel for Crash Application

Y. Leost (Fraunhofer EMI)

The modeling of wheels might have a strong influence on the vehicle response in crash simulation. The present work is about the development of a predictive cast aluminum wheel model for crash application. Coupons were directly extracted from the rim, spokes and the hub regions and experimental tensile and pressure tests were carried out to determine the material properties. The damage model GISSMO (Generalized Incremental Stress-State dependent damage Model), has been used in the material modeling to take into account the accumulation of damage, the influence of triaxiality and the mesh size dependency. Finally, the finite element model of the wheel has been validated thanks to component crash test. After some considerations about the appropriate element formulation for solids, the final simulation results are good correlated to experimental dynamic test in terms of load-displacement and failure modes

Tire Model Development Update

S. Bala (LSTC)

Using LS-DYNA for Detailed Biomechanical Impact Simulation

W. Lietz, O. Siegemund (Cadfem);
H. Ottersbach (IFA)

In this paper a workflow is presented to use real CT data of a human head in an impact simulation of a three years old child on kindergarten furniture. This paper is derived from a simulation project done by IFA and CADFEM.

Investigating impact or crash scenario is a widely needed requirement in automotive industry. For the tests and as well for the simulation, well defined dummies are used. These dummies are created in a way that specific questions can be investigated. Because of the comparability, there are commonly used criteria like the HIC value. Due to their statistical assurance they can describe the influence of an impact on humans refer to specific organ injuries or skeletal loads very well. But how can we take local values like pain level or load distribution inside of a human body's tissues into account? In testing we are restricted but not in our simulation. So this paper will show how to use realistic head models obtained from CT scans in impact simulation using LS-DYNA.

At first we will talk about the motivation of this study. What are the advantages and disadvantages of common dummies? Why should we use more realistic head models in our simulation and which results can be achieved? Further on, it will be discussed what are the main drawbacks, like material modeling and statistical assurance, of such modeling technique.

A continuous workflow from CT scan to simulation model is presented. We will show some details on how to convert real CT data into a geometry, how to mesh it, how to realize a parametric model setup and how to get the results in an automatic manner?

fter that, the model setup will be discussed in detail. Which material models and especially which material parameter are chosen. Also it is shown how to use special functionality of LS-DYNA, like *DEFORMABLE_TO_RIGID, to shorten the simulation time.

In addition of showing the feasibility of such simulation, another aim of the presented project was a sensitivity analysis of the impact. Parameters like initial velocity, table edge radius or impact points should be varied. So it will be described how an automatic preprocessing using LS-PrePost could look like, as well as which results can be used to investigate the impact.

At the end we will point out that the established workflow is feasible to investigate impact scenario with detailed modeled humans. Also an outlook of the next steps will be given.

Head Impact Analysis Correlation for Aluminum Bonnet

O. Çolpan, F. Aras (Tofas)

In recent years, vehicle manufacturers are making improvements to find more reliable solutions to pedestrian accidents. The manufacturers have to confirm these studies on some tests to carry out the requirements of international regulations. one of important these tests is head impact test.

In the head impact test, one of the most important factors that affect the injury or fatality is bonnet. The subject of this article; in recent years, light weight vehicle is more important for fuel consumption and cost of vehicle. One of these studies will be examined by virtually analyzing how the aluminum bonnet impacted the head impact test,al and whether or not the pedestrians are damaged. In addition, the analysis will be correlated with the test to improve accuracy for future studies

Validation of Thums Human Model Throw Distance in Pedestrian Accident Scenarios

M. Orlowski, C. Bastien, M. Bhagwani (Coventry University)

Increasing number of hit and run pedestrian accidents highlight the importance of accident reconstruction tools used in forensic investigations. The tools used nowadays are based on simplified assumption of particle – particle interactions (Searle's model), or real life accidents (Happer's model) which enable for prediction of the collision velocity based on pedestrian throw distance evidence obtained at the scene of the accident. Unfortunately, vehicle impact speeds can only be estimated as a range of velocities, as the Searle's model forms a velocity corridor which widens with the increase of measured throw distance giving not accurate predictions.

Development of computing architecture together with the advancement in computer human modelling opens the opportunity for bringing accident reconstruction studies to the next level and reducing the predicted velocities range. Nevertheless, to achieve this, the computer human models need to be reliable and robust. In this study, the Total Human Model for Safety (THUMS) was validated against analytical pedestrian throw distance models.

The validation studies were performed with THUMS 4.0 at three different model stances and four different impact velocities (20, 30, 40 and 50 km/h) as well as three different stances, namely: standing, walking and running pedestrian. Analyses results were validated against Searle's and Happer's throw distance models.

THUMS kinematics agreed well with the current accident reconstruction tools in terms of model behavior and predicted throw distance. The behavior of the THUMS model is different for low and high velocity impacts showing good agreement to the field data in terms of body kinematics. In particular, low impact velocities cause forward projection of the human body, while high impact velocities are characterised by the wrap trajectory of the THUMS model.

Define_Pressure_Tube: Simulating Pressure Tube Sensors in Pedestrian Crash

J. Karlsson (DYNAmore Nordic)

This paper presents the new keyword **`*DEFINE_PRESSURE_TUBE`**, designed to efficiently simulate pressure waves in a thin air-filled tube. The main application in mind is a crash detection system for pedestrian safety, where an air-filled tube is embedded in the front bumper and fitted with pressure sensors at the ends. In the case of an impact, the tube is compressed and a pressure wave travels to the sensors, enabling localization and extent of the impact. In recent years, such systems have gained popularity in the automotive industry, posing a challenging task in efficient and accurate simulations.

The keyword defines a closed gas filled tube using (hollow) beam elements and the pressure is calculated from area changes, given by contact penetration from surrounding elements. Pressure propagation is governed by a 1D model based on the compressible Euler equations, resulting in a very efficient method compared to 3D CFD or particle methods. Pressure, density, velocity and tube area are output through the keyword **`*DATABASE_PRTUBE`**, and can be visualized in LS-PREPOST.

The aim of this paper is to give an overview of the theory and usage of the keyword, as well as show comparisons with experiments and existing methods in LS-DYNA.

Impact of Soft Body Materials, an Experimental and Numerical Approach using a Hopkinson Tube: Application to Substitute Bird

J. Pernas-Sánchez, R. del Caurillo, J. A. Artero-Guerrero, D. Varas, J. López-Puente
(University of Madrid)

Aeronautic and Aerospace industries permanently seek to optimize structural components due to the high safety and reliability requirements. These improvements in the structures should withstand severe case of loads to accomplish the certification processes; traditionally the designers use experimental tests to validate it. The use of virtual testing could reduce the development times and costs but this needs reliable material models to produce accurate results.

Load cases for aeronautic structures are extensive, but among others, impact is one of the most concern loads for the structures. Literally from an EASA 2011 report[1] “A critical safety issue for the design of primary aircraft structures is vulnerability and damage tolerance due to foreign object impact from bird strike, hail, tyre rubber …”, highlighting the impact threat as a key factor in structure design. Thus, for reliable damage prediction in virtual testing, it is necessary to obtain appropriate data and develop modeling techniques for this kind of soft impactors.

In this work, a combined experimental-numerical methodology is presented to validate material models for deformable impactors under impact; applying it to a real case of study: the bird impact. In the experimental campaign substitute birds (SB), made from gelatin, are launched against a Hopkinson tube to measure the stress pulse generated in the tube [2], capturing the kinematics of the impactor using high speed cameras. The results obtained have been compared with a numerical model to prove the feasibility of the combined experimental-numerical methodology.

The numerical model for the Hopkinson tube and the SB was implemented in LS-DYNA. Prior to simulate the impact, the numerical model of the Hopkinson tube was validated by means of comparing the modal frequencies experimentally measured and numerically obtained. For this purpose, the modal package included in the implicit module of LS-DYNA was employed. Once the Hopkinson tube model was validated, the bird impact was simulated using the explicit module. The SB was defined using SPH, this meshfree approximation was selected due to the large deformation suffered by the SB during the impact. The constitutive behavior of the gelatine was modelled as a fluid like material with a polynomial equation of state, the material properties were obtained from the literature ([4][5]).

The results predicted agree with the experimental data, so the combined methodology was proved to be useful for material models calibration under impact conditions. The validated numerical simulations have been used to study the robustness of the experimental technique under impact drifts that can be occurred during the experimental tests (i.e. changes in the impact location or impact angle…).

Literature

[1] Toso N, Johnson A. LIBCOS-Load upon impact behaviour of composite Structure. Research Project EASA.2009/3. European Aviation Safety Agency (EASA). 2011

[2] J. Seidt, J.M. Pereira, J. T. Hammer and A. Gilat, C.R. Ruggeri Dynamic Load Measurement of Ballistic Gelatin. Impact Using an instrumented tube.

[3] R.Budgey; The development of a substitute artificial bird by the international birdstrike research group for use in aircraft component testing.;IBSC25/WP-IE3; Amsterdam, 17-21 April 2000.

[4] V. Nagaraj, T. Velmurugan, “Numericalbird strike impactsimulation of aircraft composite structure”, *IOSR Journal of Mechanical and Civil Engineering*, pp. 01-10,

[5] James S. Wilbeck, “Impact Behavior of low strength projectiles”, Technical report AFML-TR-77-134, 1977.

Thermal Coupling Method between SPH Particles and Solid Elements in LS-DYNA

J. Xu, J. Wang (LSTC)

Smooth particles hydrodynamics is a meshfree, Lagrangian particle method and a simple, yet flexible method for modeling fluid flows and solid bodies in a robust way. It has been applied extensively to the multiphase flows, heat conduction, high explosive problems and so on. For the thermal coupling between any two parts in LS-DYNA, the standard way is through thermal contacts which require the contact areas between those two parts.
Due to particle property, SPH particles can handle extremely large deformations, particles can be moved without limitation. In real engineering applications, SPH particles may have very complex free surface and material interface behaviors, including break-up into fragments, and new surfaces will be generated automatically every cycle when interacting with Solid elements. It is quite difficult to update new contact surfaces and calculate the true contact areas between SPH particles and Solid elements.

In this paper we introduce a new thermal coupling method between SPH particles and Solid elements through keyword *DEFINE_ADAPTIVE_SOLID_TO_SPH with icpl=3 and iopt=0 options without using thermal contacts, the ghost SPH particles inside the solid elements are used as the coupling medium between original SPH particles and the Solid elements. Some examples are presented to show the robustness of the coupling method.

Simulation of Agricultural Soil Tillage Machine using DEM

H. Mouradjalian, Z. Asaf, I. Shmulevich (Technion - Israel Institute of Technology);
B. Zion (Israeli Agricultural Research Organization)

Simulation of agricultural soil tillage machine is important for power consumption reduction during tillage processes. Potato and onion harvesting machines as well as other soil tillage machines need to penetrate into the ground and convey large amount of soil or root crops over a tilted pickup chain conveyor. These types of machines sustain extremely large loads that require heavy pulling tractors. The Israeli Agricultural Research Organization – Volcani Center developed such a machine for weed (Nutsedge) pest control. In this machine as in many others a large blade is placed in front of the conveyor which penetrates into the soil, tills the soil at a required operation depth and directs the tilled soil onto the conveyor. The mechanical design of the tillage blade and its position relative to the pickup conveyor has a great impact on the power efficiency of the machine. A small change in the operating angle of the blade or its position relative to the pickup conveyor can have a significant impact on the operation of the machine and on the acting drag forces that in turn might cause a significant waste of energy.

The aim of this work is to simulate the machine-soil interaction as a tool for better machine design and to reduce the power requirements of the machine by the developed tool. The reported work used LS-DYNA dynamic analysis for modeling the machine by Lagrangian elements and the soil by cohesive Discrete Element (DE) particles to represent clay soil. Optimization of the blade's shape, the position of the conveyor and their combination by trial and error experiments would be a long and expensive process.

The parametric examination using the developed model showed the drag force distribution over the different parts of the machine. The simulation was calibrated and validated by full scale experiments using the soil tillage machine puled by a tractor at an agricultural field. During the experiments two types of blades were compared at different tillage depths. The soil stiffness in field was measured by a dynamic cone penetrometer. Several other parameters were measured during the test: the drag forces and their direction, the tillage velocity and the depth of tillage. The experimental work validates the simulation. The results of the research show that over 30% power reduction can be obtained by an improved design of the blade's shape, angle and position relative to the pickup conveyor using LS-DYNA DE model simulation.

The simulation and the experimental validation will be presented.

Discrete Element Modelling of a Metamaterial for Launcher Tanks Dynamic Experiments

T. Legaud, E. Grippon, V. Lapoujade, P. Chiambaretto (DynaS+)

Experimental vibration certification of launcher cryogenic tanks is an important issue in the aerospace industry. Liquid hydrogen LH2 dynamic vibratory behavior in tanks experiments cause some scientific and technical problems. Indeed, the security, transport and conditioning problematics still remain critical.

Because of its demonstrative character, the space industry, particularly the launchers applications, have been chosen in order to show the effectiveness of the Discrete Element Method (DEM) recently implemented in LS-DYNA software. This project, financed by the French “Occitanie region” (FEDER), aims at understanding and reproducing the dynamic behavior of a light fluid (LH2) in a tank thanks to a granular media.

This metamaterial will be composed of hollow spheres represented by Discrete Element Sphere DES. Its aim is to represent the LH2 dynamic behavior within a tank, without presenting the same flammability, explosiveness and fugacity risks as the LH2. This new material permits to simplify and secure cryotechnical launcher tanks qualification.

In order to reach this objective, a new methodology is developed by combining a numerical, an analytical and an experimental methodology. Thus, the project is based on these three main approaches:

- Thanks to homogenized models, the analytical approach permits to effectively establish the link between the homogenized granular media mechanical properties and the dynamic response of the tank fully filled. These simplified analytical calculations are used to delimit the research in term of material type (E, ρ, v) and frequency range.

- Thanks to the DEM, the numerical approach permits to make the link between the properties of one hollow sphere, of an equivalent full sphere and of the granular media made by these spheres. The dynamic study is also performed, thanks to the entire and fully filled tank modeling.

- Thanks to the coupling between the manufacturing, the numerical tests and the experimental ones. a procedure to produce and characterize the granular media properties will be establish in order to control its dynamic (vibratory) behavior in a tank.

This article focuses on the numerical approach. The first part of the article especially deals with the numerical granular media properties establishment. Several experimental tests are reproduced in order to establish the main hypothesis that have to be taken into account to correctly reproduce the media characteristics (mechanical properties, friction, organization, dispersion…).

The second part deals with the study of the vibrating tank (empty and fully filled with spheres), and particularly focus on the methodology used to obtain the required frequencies. This methodology consists in applying a defined pressure on the granular material confined in the cylinder in order to modify its density and its stiffness. Both properties have important impacts on the tank vibratory behavior.

Compression Molding Analysis of Long Fiber Reinforced Plastics using Coupled Method of Beam and 3D Adaptive EFG in LS-DYNA

S. Hayashi (JSOL); H. Chen, W. Hu (LSTC)

Composite materials like fiber reinforced plastics (FRP) are becoming more widely used in the automotive industry and have been found very effective in reducing vehicle weight. Recently, long carbon fiber reinforced thermoplastics are increasingly being used for lightweight structural parts with high stiffness, strength and energy absorption performance. Compression molding is considered one of the most efficient manufacturing processes to mass produce FRP parts for automotive applications. Compression molding can form long fiber reinforced thermoplastics into complex shapes with relatively low manufacturing cost and short process time. However, currently there are a very limited number of high-accuracy simulation technologies available that can predict long fiber orientation, filling timing and other behavior required for compression molding.

In the first part of this paper, a new simulation technology for compression molding of long fiber reinforced plastics implemented in LS-DYNA is introduced. The main features of this new technology are fibers modelled by beam elements and matrix modelled by tetrahedron solid elements with r-adaptive remeshing function based on an Element-Free-Galerkin (EFG) formulation. In the second part of this paper, an investigation of how to define macroscopic viscosity during compression molding of long fiber reinforced thermoplastics is presented. In the final part of this paper, compression molding simulation for stampable sheet made of long fiber reinforced thermoplastics is presented using a complex shaped punch with one cross-rib geometry.

The simulation results show good agreement to experiment. In conclusion, this new simulation technology has great potential to simulate compression molding of long fiber reinforced plastics with high accuracy.

*MAT_4A_MICROMEC – Theory and Application Notes

P. Reithofer, A. Fertschej, B. Jilka (4a engineering);
A. Erhart, S. Hartmann (DYNAmore)

A huge number of short and long fiber reinforced thermoplastics play a decisive role in the automotive industry to ensure affordable lightweight design and availability in large quantities. The properties of these materials are especially highly influenced through the manufacturing process (typically injection molding for SFRT and LFRT). As seen in the last LS-Dyna Conference [1], there is a strong industry interest to consider the manufacturing process induced local anisotropy in crash and general dynamic simulations.

Looking on the material model, only constitutive composite material laws are currently available in LS-Dyna (e.g. *MAT_157). Starting with the next LS-DYNA Release R10 a new micromechanics motivated material model *MAT_215/ *MAT_4A_MICROMEC will be forthcoming, which should help to setup easier simulation process chains and to be more predictive in daily simulation work. First verification and validation examples have been shown in [2].

Current investigations on different plastic materials deal with Material Parameter Identification Procedures for *MAT_215. Besides further validation of the anisotropic material deformation behavior failure prediction of different polymer grades by using the implemented features is work in progress.

The influence of material parameters on general behavior will be shown and leads over to some real examples in following presentations [3,4].

Literature

[1] www.dynamore.de/en/downloads/papers/2016-ls-dyna-forum
[2] Reithofer, P. et. al: *MAT_4A_MICROMEC – micro mechanic based material model, German LS-DYNA Forum, Bamberg 2016
[3] Zhao, T. et. al: High-Dynamic Drop Test Simulation for Fiber Reinforced Plastics in Automotive Electronic Control Unit, 11th European LS-DYNA Conference, Salzburg 2017
[4] Steinberger, R. et. al: Considering the Local Anisotropy of Short Fiber Reinforced Plastics: Validation on Specimen and Component, 11th European LS-DYNA Conference, Salzburg 2017

High-Dynamic Drop Test Simulation for Fiber Reinforced Plastics in Automotive Electronic Control Units

T. Zhao, D. Papathanassiou (Bosch Automotive Products)

Short fiber reinforced plastics (SFRP) are widely utilized for electronic control units (ECU) in automotive fields. Failures in plastic parts often lead to re-tooling hence unpredicted increase in cost of manpower and time. Simulation indeed can offer the chance to virtual investigation without real samples therefore can deeply contribute in early development phase. As well-known anisotropic material behavior induced by non-trivial fiber distribution of SFRP strongly influences local failure prediction. Moreover in high-dynamic drop problems viscoplasticity needs to be accounted to obtain the correct failure location. An oversimplified model or alternative isotropic approach in simulation can easily raise misleading results, while by being comprehensive in material model and tool chain one can respectively describe and include dominant influencing factors.

To serve this interest, specimen made of reinforced polybutylene terephthalate with 30 wt% short glass fiber (PBT-GF30) are used for material characterization tests, mainly including 3-point bending and puncture tests by static and dynamic means respectively. By using 4a impetus an anisotropic LS-DYNA material card including the failure (DIEM) was determined reversely to describe the anisotropic viscoplasticity along with failure criteria applied in commercial program LS-DYNA. Integrative tool chain with considering fiber distribution from process simulation to explicit structural simulation is implemented for an ECU level drop simulation, with which consistent ECU level drop tests are performed. By means of comparing critical drop height acquired from test, the feasibility of this tool chain simulation is shown.

Considering the Local Anisotropy of Short Fiber Reinforced Plastics: Validation on Specimen and Component

R. Steinberger, T. Gross (Hirtenberger Automotive Group);
S. Paul (Simpatec);
P. Reithofer (4a engineering)

A Review of Structural Part Modelling for Blast Simulations

G. Balaban, I. Kurtoğlu (FNSS Savunma Sistemleri)

In this study, the effects of various element formulations and mesh sizes are investigated for buried charge simulations using the non-linear finite element code LS-DYNA. Simulations are performed according to the real test conditions and the results are compared with the plate level mine blast experiments. Tests are carried out using a test setup which is designed and manufactured by FNSS. The blast simulations are examined using ALE method.

Simulation model consists by ALE domain which includes soil, air and the explosive definitions and Lagrange domain for the bottom and side plates of the vehicle. The evaluated test plate is made of RHA steel with 20 mm thickness. Simplified Johnson Cook material model is used and the parameters are determined by Split-Hopkinson Pressure Bar tests. Plates are modelled using shell, solid and thick shell elements with different element formulations. Consequently, the elastic and plastic deformation results, effective plastic strain distributions, pressure histories and the CPU times are compared. Furthermore, the advantage and disadvantages of the considered formulations and parameters are presented.

Applying Buried Mine Blast Loads to a Structure Utilizing the User Module Capability

E. Lazerson, H. Raz, Z. Asaf (Plasan SASA)

Developing armored vehicles to withstand a buried mine blast is a challenging task. The development of solution with optimum trade-off between mobility and survivability cannot be done by trial and error alone. The development of reliable CAE model of the vehicle and threat, using a simulative tool, is essential.

In the early design phase, extensive use of simulation is done to optimize the structure. This process requires a short turnover time for the simulations. Use of ALE or Particle Blast can give good results but involves long runtimes. On the other hand, using Load Blast (ConWep) or Initial Impulse Mine is fast and simple.

Initial Impulse Mine works by applying initial velocity on selected elements. The unselected elements have initially zero velocity. This velocity discontinuity can lead to unreasonable results.

The aim of the Load Blast is to simulate an air blast but not buried mines. It is possible to calibrate Load Blast to get the correct peak pressure or the local impulse, but not both. It is impossible to change Load Blast spatial pressure distribution.

In this work a new user-defined module was implemented. This ConWep-like user loading enables the simulation of a buried mine explosion by modification of pressure distribution via a shape function. Using this technique a better pressure and momentum distribution over the target can be achieved with running times similar to *LOAD_BLAST_ENHANCED (ConWep). The new module is implemented in FORTRAN and activated using the *USER_LOAD_SEGMENT keyword. The compiled module is activated in the LS-DYNA deck by using the new *MODULE_LOAD keyword. The use of the new solvers with dynamic loading of modules enables our simulation team members to use this code easily and to issue new versions of the module as needed.

FSI with the Detailed Chemistry and their Applications in LS-DYNA CESE Compressible Solver

I. Kyoung-Su, Z. Zhang, G. Cook (LSTC)

Recently, we have developed a new module of the modeling fluid structure interaction with the finite rate chemistry in compressible CESE solver, which is based on the immersed boundary FSI method, and fully coupled with the LS-DYNA structural FEM solver.

In the CESE fluid structure interaction solver, we have two principal treatment methods, i.e., the immersed boundary method with a direct forcing strategy and the moving mesh method. Although the moving mesh method is more accurate than the immersed boundary method, the latter is most efficient and robust when the problem involves large deformation such as a structure demolition by explosion.

In the present report, we have demonstrated most practical fluid structure interaction problems by using the immersed boundary method with chemistry:

i) shock-induced combustion in front of a spherical projectile moving at supersonic speed,
ii) the blast relief wall simulation in methane and air mixture (CH4/Air), and
iii) the fracture of the shell and solid structures by high explosive spots in an H2/O2 premixed environment.

The results are validated with existing experimental data and descriptions of the keyword setup are provided in details for users.

Finite Element Modelling of a NiTi SMA Wire

W. L. H. Wan A. Hamid, L. Iannucci, P. Robinson (Imperial College London)

The development of a Finite Element Model of a NiTi shape memory alloy (SMA) wire in a commercial explicit finite element software, LS-DYNA, is presented. A user-defined material (UMAT) model has been developed by implementing one of the earliest one-dimensional SMA constitutive models, the Tanaka model [1], into LS-DYNA through a FORTRAN code. The aim is to apply this model to develop a SMA-actuated morphing wing. Morphing has attracted considerable attention among researchers for the past few decades because of the potential of providing optimum flight conditions at various flight missions. A combination of the morphing wing with smart materials such as SMAs offers further advantages such as a significant reduction in weight and system complexity, compared to an actuation achieved by mechanical motors or hydraulic systems.

The simplest example of an actuated structure is presented, that is a SMA wire connected to a linear spring in series. The SMA wire was modelled as a beam element with one integration point, which is equivalent to a truss element, whereas the linear spring was modelled as a discrete element. One complete heating-cooling cycle was applied on the SMA wire. Upon heating, a reverse transformation (martensite-to-austenite) occurred, caused the wire to shorten and consequently extended the spring. Hence the stress in the wire increased while the SMA strain decreased until the end of the transformation. Upon cooling, a forward martensitic transformation (austenite-to-martensite) took place and reduced the stiffness of the wire. As a result, the spring contracted and the wire extended, and so the stress decreased while the SMA strain increased until the end of the transformation.

This finite element prediction of the thermomechanical behavior was compared to an analytical solution for small displacement, and a close agreement was achieved. A parameter analysis was then carried out to analyse the dependence of the thermomechanical behavior on several parameters such as the length and cross-sectional area of the SMA wire, as well as the spring stiffness (stiffness of the actuated structure). As expected, the FE model showed that the recovery stress (maximum stress at the end of heating cycle) increased whereas the recovery strain (maximum strain at the end of heating cycle) decreased with increase in the SMA length and spring stiffness, and with decrease in the SMA cross-sectional area.

The user-defined SMA model was further tested as a design tool to morph a pre-curved corrugated plate. The results showed that for large diameter SMA wires, increasing the number of SMA wires in each cell resulted in a small increment in the tip deflection. A parallel configuration is preferable than a ‘V’ configuration for cells consisted of two SMA wires, because slightly higher tip deflection can be achieved. This is mainly due to the influence of the SMA length on the thermomechanical behavior. Finally, the SMA model was applied on a morphing wing consists of six corrugated plates between its leading and trailing edges. The resulted trailing edge vertical deflection is 36.6 mm, about 10% of the chord length. In conclusion, this work provides a foundation for future exploration of SMA-actuated morphing wings using LS-DYNA, such as an optimization of the internal structure using LS-OPT, and fluid-structure interaction (FSI) simulations to include the effect of incoming air flow on the movement of the wings.

Literature

Tanaka, K, Kobayashi, S, and Sato, Y, “International Journal of Plasticity”, 2, 1986, 59-72.

Process Chain Simulation for Die-Less-Hydroforming Including Welding and Forming using DynaWeld and LS-DYNA

A. Metzger, T. Ummenhofer (Karlsruhe Institute of Technology)

Within the scope of "Die-Less-Hydroforming", two or more thin metal blanks are joined at their edges by seal-welding. Thus, a two-dimensional flat "envelope" made of steel sheet results, that is formed into spatial structure through inflation of a medium (e.g. water) under continuous pressure increase. In comparison to conventional hydroforming-processes, no additional forming tool like a die or mold is used, leading to special phenomena like wrinkling or buckling during the forming process. The shape of the resulting 3D-structure is mainly controlled by the initial geometry of the blank and the forming pressure.

A FEM-simulation of this special forming process with LS-DYNA was already introduced by the authors inter alia in [1]. Furthermore, a thermo-structural mechanical welding simulation of the assembly process of the blanks was developed with the software "DynaWeld" and the related solver "LS-DYNA"; see [2]. It is now assumed, that the residual welding stresses or welding distortions resulting from seal-welding of the blanks have a strong and very sensitive influence on the "free" forming process regarding buckling and wrinkling in particular due to the small sheet thickness.

First results are presented in this contribution from a simulation of a process chain for "Die-Less-Hydroforming" with simple geometries of double-layered blanks made of austenitic stainless steel (1.4301), that are joined by an arc welding process. In the first step, the joining of the double-layered blanks is simulated with "DynaWeld" and "LS-DYNA" which means the welding process by a thermo-structural mechanical welding simulation. Subsequently, a forming simulation of the double-layered blanks is performed, whereby the essential information from the welding simulation (that means distortion, residual stresses, etc.) is transferred through a DYNAIN-file to the forming simulation. The aim is to determine the influence of the distortion respectively the residual stresses resulting from welding on the formation process (that means in particular the emergence of wrinkles and buckles).

Literature

[1] Metzger, A., Ruff, D.C. and Ummenhofer, T. (2014): FEM-Simulation of „Die-Less-Hydroforming“ using LS-DYNA, 13. LS-DYNA FORUM 2014, Bamberg

[2] Metzger, A., Ummenhofer, T. (2016): Schweißsimulation von „Die-Less-Hydroforming“-Platinen mit DynaWeld und LS-DYNA, in: Hildebrand, J., Loose, T. (Hrsg.), Tagungsband Simulationsforum 2016 Schweißen und Wärmebehandlung, Weimar, S. 148–156

Study on the Electromagnetic Flux Generation using the new 2D Axisymmetric Capability of Electromagnetism Solver in LS-DYNA

K. Takekoshi (Terrabyte)

Study on the electromagnetic flux generation using the new 2D axisymmetric capability of electromagnetism solver introduced in LS-DYNA version R9 will be presented by comparing with the result calculated using 3D electromagnetism solver previously reported.

Control System in LS-DYNA

C. Keisser (DYNAmore France);
I. Yeh (LSTC)

Control System is a very old concept aimed at improving or changing the behavior of any dynamic system. In our daily lives, there are many control applications like in a toaster or in room temperature regulators but also in our cars when using cruise control or ABS. More recently, Control System appears to be also very useful and powerful in pre-crash safety or in modeling Human Body response.

The control can be as simple as an On/Off action (open-loop control). Or, for closed-loop control, output measurements provided by sensors feed the controller which then uses actuators to modify the system behavior.

Several software like MATLAB/Simulink, Scilab, or Octave have their own control system toolbox. In LS-DYNA, several control features are currently being developed: an interface to connect with MATLAB or Scilab, an internal Control System Toolbox integrating control tools like the popular PID controller, and the piezo-electric material.

Marine Accident Integrated Analysis System using Highly Advanced M&S System of FSI Analysis Technique

S.-G. Lee, J.-S. Lee, J.-H. Park, T.-Y. Jung (Korea Maritime & Ocean University)

Investigation of marine accident causes usually depends on the judgments of maritime experts, based on the statements of the concerned persons in the case where there is no navigation equipment, such as AIS and VDR. Scientific verification also has a limitation in the case of their conflicting statements. It is necessary to develop Marine Accident Integrated Analysis System (MAIAS) using highly advanced Modeling & Simulation (M&S) system of Fluid-Structure Interaction (FSI) analysis technique for the scientific investigation of marine accident causes and for the systematic reproduction of accident damage procedure. To ensure an accurate and reasonable prediction of marine accident causes, full-scale ship collision, contact, grounding, flooding, capsize, sinking and turning simulations would be the best approach using hydro code, such as LS-DYNA, with its FSI analysis technique, propulsion force for ship velocity, and rough sea weather such as current, strong wind and wave with irregular spectrums.

The objective of this paper is to present the findings from full-scale ship collision, grounding, flooding, capsize, sinking and rapid turning simulations of marine accidents, and to demonstrate the feasibility of the scientific investigation of marine accident causes using MAIAS.

Cause Investigation of Flooding & Sinking Accident of Ro-Ro Ferry Ship using Marine Accident Integrated Analysis System

S.-G. Lee, J.-S. Lee, J.-H. Park, T.-Y. Jung (Korea Maritime & Ocean University)

Ro-Ro Ferry ship was capsized and was sunk down to the bottom in the sea water due to the rapid turning for the several reasons, such as lack of stability and poor lashing, etc. Objective of this study is to figure out the air pocket existence in the flooding & sinking accident using Marin Accident Integrated Analysis System (MAIAS; highly advanced M&S system of Fluid-Structure Interaction analysis technology). Several things were carried for this investigation of air pocket existence, such as accurate ship posture track according to accident duration using several accident photos and movies, precise ship and cargo moving track and sea water inflow amount according to accident duration using floating simulation and hydrostatic characteristics program, accurate understanding of exterior openings and interior paths of sea water inflow, simulation of sea water inflow using flooding & sinking simulation and calculation of exterior openings & interior inflow paths. There was relatively good prediction of air pocket existence.

Cause Investigation of Capsizing Accident of Ro-Ro Ferry Ship using Marine Accident Integrated Analysis System

S.-G. Lee, J.-S. Lee, J.-H. Park, T.-Y. Jung (Korea Maritime & Ocean University)

Ro-Ro Ferry ship was capsized and was sunk down to the bottom in the sea water due to the rapid turning for the several reasons, such as lack of stability and poor lashing, etc. Objective of this study is to investigate the capsize accident cause by full-scale ship rapid turning simulation through the comparison with AIS track in this capsize accident, considering several factors, such as GoM, ship velocity, rudder angle, etc., and using Marin Accident Integrated Analysis System (highly advanced M&S system of FSI analysis technology). MAIAS of full-scale ship turning simulation are verified by comparison with maneuvering performance sea trial test result of initial building ship. Several things were carried for this rapid turning simulation, such as accurate ship model modification using floating simulation according to hydrostatic characteristics of loading conditions, and investigation of cargo loading arrangement and cargo lashing states. There was relatively good agreement of full-scale ship sea trial turning simulation with sea trial test result, and good prediction of cargo loading arrangement and cargo lashing states comparing to the AIS track in this capsize accident.

Test and Simulation Approach Towards the Certification of an Aircraft Structure Subjected to a Bird Strike

H. Abdulhamid, F. Plassard (Thiot-Ingenierie)

Thiot-Ingenierie worked with a compagny specialised in Retrofit packages for the aviation industry. To improve Satcom and connectivity systems, the compagny developed a dedicated structure able to handle a radome with communication means inside.

The goal of the project is to design and certify a structure that is an interface between the radome and fuselage. This radome can be subjected to a bird strike and the study aims is to ensure that the loads transmitted by the radome and its interface structure do not cause any critical damage on aircraft fuselage.

The challenge of this project was to design the structure within a time development less than 8 months and with only one shot certification test performed at Thiot-Ingenerie laboratory.

A development strategy mixing numerical simulations and experimental tests has been performed to get material behavior of composite materials and to numerically optimize the response of the structure. This mixed strategy allowed us to perform the certification test with an improved structure. This method improves the development efficiency to save time and money.

Workshop on LS-OPT Robustness Analysis

K. Witowski (DYNAmore)

The goal of this workshop is to provide an overview of the methods for robustness analysis that are available in LS-OPT. Herein, the basic ideas of direct and metamodel-based Monte Carlo Analysis as well as RBDO/RDO will be discussed.

A life demonstration of how to set up a robustness analysis using the graphical user interface of LS-OPT and how to visualize and evaluate the results will also be given.

Thursday Morning Sessions

10:40 – 12:20

Modeling of Joints with Inserts for Sandwich Structures in Crash Simulation

P. Rochel, S. Sommer (Fraunhofer IWM)

The growing interest in saving energy and resources leads to new concepts regarding lightweight design in automotive engineering. Sandwich structures are an efficient way to reduce weight in technical constructions. Therefore it is necessary to develop tailored joining technologies to deal with the competition of joining sandwich structures. Bonded metallic inserts are one of the techniques to improve the load capacity for this kind of joints. A modelling method has to be provided for reliable predictions of the behavior and the crashworthiness of joints with inserts in FEM-simulations.

This paper introduces a technique of modelling inserts, which is based on the available simplified models for joints in LS-DYNA. It is a combination of the *CONSTRAINED_SPR3-Model (Model 2) [1], which was developed for a better modeling of the behavior of self-piercing riveted joints and a simple element-based model. The method is further validated by comparing the experimental and simulation results of tensile and lap shear specimens. The experiments were carried out as a part of the German research project: "*Entwicklung von Konzepten und Auslegungsstrategien zum punktförmigen Verbinden von innovativen, strukturell tragenden Sandwichstrukturen (PuVerSand)*" funded with budget of the Federal Ministry of Economics, Baden-Württemberg.

Literature

[1] Bier, M., Sommer, S.: "Simplified modeling of self-piercing riveted joints for crash simulation with a modified version of *CONSTRAINED_INTERPOLATION_SPOTWELD", 9th European LS-DYNA Conference, 2013

Development of Accurate Finite Element Models and Testing Procedures for Bolted Joints in Large Caliber Gun Weapon Systems

M. Koehler, G. Fish (US Navy)

Large caliber weapon systems, such as those mounted on tracked vehicles or in turrets on ships, typically develop large forces when fired, which are transmitted to the structure that houses them. In the case of modular weapon systems, often the platform for the weapon system is already developed and the weapon system is integrated at a later stage. In this scenario, it becomes imperative to fully quantify the loads experienced in the bolted connection between the weapon system and the platform in order to ensure a satisfactory fatigue life of the platform. Typically, this is accomplished through the use of finite element analysis in the early stages of development and verified through testing during later stages of development.

This work investigates the development of finite element models in LS-DYNA that accurately model bolted joints and compares the tradeoff between efficient models with a coarse mesh and more accurate models with an expensive, fine mesh. Furthermore, these models are validated against an instrumented test specimen in order to determine the accuracy of the finite element models. Throughout this comparison, various sensors are used in multiple locations in order to demonstrate that various conflicting results may be obtained from the same load applied on the joint depending on the type of sensor used and the sensor's location in the joint.

NSWCDD-PN-17-00031; Distribution Statement A: Approved for Public Release

Characterization and Modeling of Spot-Weld Joints in Press Hardening Steels Associated with Softening in Heat Affected Zone

H. Ghassemi-Armaki, Q. Khan (ArcelorMittal);
A. Gill, S. Zilincik (Chrysler)

Next generation of advanced high strength steels (AHSS) are extensively considered for usage in Body-in-White (BIW) with lower thickness to reduce the vehicle weight and increase fuel efficiency. Resistance Spot Welding (RSW) continues to be a major joining process in BIW, with an average of 4500 spot-welds per vehicle. Spot-weld strength has direct correlation with the thicknesses, and yield strength of the joining sheets in the stack up configuration.

Characterization and FEA modeling of spot-welds for accurate failure prediction is of vital importance in vehicle crash safety simulations. This will also lead to an optimized weld layout, optimum weld pitch and cycle time and this has a direct impact on assembly plant layout/investment. Press Hardening Steels “PHS” (e.g. Usibor® 1500) show high strength after hot-stamping and are one of the major category of AHSS. These steels contain a fully martensitic microstructure, which show softening in Heat Affected Zone (HAZ) after spot-welding because of martensite tempering. Accurate prediction of load-displacement and failure load of the spot-weld depends on input data for softened HAZ and connection definition of two joining sheets.

In the present study, a modeling method for the spot-weld nugget and HAZ zone was developed and compared to test results. An eight, hexahedral element assembly is used to represent a spot-weld and an element ring of width, 0.35 times of the weld diameter is used to simulate softened HAZ. Since softening in the HAZ is not uniform and shows a gradient as a function of distance from spot-weld center, the HAZ material simulated using Gleeble for different locations was used to characterize and provide material input data for the HAZ shell element.

Spot-weld failure was characterized by testing the spot-weld in different loading modes, including, tension-shear, coach-peel, cross-tension and KSII configurations (30, 60 and 90 degrees). *MAT_100_DA in LS-DYNA (version 9.1) was used along *DEFINE_CONNECTION_PROPERTIES for defining material properties for the weld. Results show that the chosen stress-strain curve representative for HAZ shell elements extracted from Gleeble samples can predict the yielding of the load-displacement curve well. Additionally, *MAT_100_DA parameters depend on sheet metal thickness, and exponents for failure equation in this model can be lower than 1 and up to 4 as reported in literature.

Investigation of Undermatched Weld Fracture for Automotive Applications

B. Hiriyur, P. Woelke (Thornton Tomasetti)

Weld fracture is an important consideration for automotive industry, especially in relation to the lightweighting efforts. Accurate quantification of the energy dissipated during the fracture process can lead to not only better predictive capabilities, but also to improvements in the material processing and joining technology. This is especially relevant for undermatched welds, where the weld metal and/or heat affected zone are significantly weaker than the parent metal (e.g. aluminum welds, martensitic steel welds). This strength reduction, caused by secondary heat treatment during welding, results in localization of plastic deformation in the weld and HAZ which negatively affects structural performance.

A comparative study of energy dissipated during ductile fracture of two aluminum plates, unwelded Al5083-H116 and friction stir welded Al6061-T6, is presented here. The results indicate that while welding in the Al6061 plate causes significant strength reduction of the heat affected zone (HAZ) material, it also leads to significant ductility increase. This results in very high relative toughness of the HAZ material in reference to the parent metal. The study also shows that, in the unwelded plate, significant energy dissipation occurs outside of the fracture process zone while plastic dissipation in the welded plate is confined to the weld and heat affected zone.

A simple modeling methodology is also presented, along with a consistent calibration procedure for mode I fracture of undermatched welds. While the validation problems focus primarily on aluminum, the proposed approach is equally applicable to any metals with undermatched welds, e.g. martensitic steels.

Enhancements to Implicit Mechanics

R. Grimes, R. Lucas, C. Weisbecker, C. Ashcraft, F. H. Rouet, J. Anton (LSTC)

Solving large sparse linear systems of equations is often the computational bottleneck for implicit calculations. The team of developers at LSTC is working on many aspects of this linear algebra.

This is the first of two talks describing this body of work. While the second talk will focus on the performance of the distributed memory linear equation solver, this talk will focus on other aspects of the overall solution process, often those directed by the user. This includes reordering to reduce the storage and operations required by the default multifrontal linear solver and our progress on a distributed memory parallel ordering package. We also discuss alternative solvers that LSTC is considering such as MUMPS and iterative solution. Finally, we will describe enhancements to the user interface making implicit mechanics easier to use.

In addition we will give an update on new features for implicit including implicit linear multiple load analysis and other user requested features.

Improving LSTC's Mulitfrontal Linear Equation Solver

R. Lucas, R. Grimes, F. Rouet, C. Weisbecker (LSTC);
N. Meng (Intel);
T. Zhu (Cray)

Solving large sparse linear systems of equations is often the computational bottleneck for implicit calculations. LSTC is continuously improving its existing solvers, as well as looking for new technology.

This talk describes improvements to the default multifrontal distributed memory linear solver used by LSTC. Changes in semiconductor technology have transformed microprocessors and their memories. In collaboration with Intel, LSTC has adapted by reducing memory movement and restructuring arithmetic kernels to increase the use of LAPACK kernels. Meanwhile, solid state persistent storage technology dramatically improves the performance of out-of-core computations. To address the ever growing size and complexity of the models being created by LS-DYNA users, LSTC is working with Cray to redesign its sparse matrix factorization kernels to solve hundreds of millions of equations using tens of thousands of cores. Finally, to address the exponential scaling of storage and operations associated with ever larger models, LSTC is once again exploring low-rank approximation technology.

An Implicit Study of High Order Elements in LS-DYNA

T. Borrvall (DYNAmore Nordic);
Prof. D. Benson, H. Teng (LSTC)

An overall trend in simulation technology is towards increasing the accuracy of model features to yield a more rapid convergence with mesh refinement. In the context of element technology, migrating from shells to solids or from low to high order elements are possible ways to accomplish this. High order elements are therefore of particular importance in implicit analysis, where spatial discretization is limited by algorithmic complexity and memory consumption. Sometimes simulation standards and company regulations even make quadratic elements mandatory, and by tradition they are an important contribution to an implicit finite element software.

LS-DYNA is a strong player on the implicit market and today offers a complete set of quadratic shell and solid elements. The intent with the present paper is to give an overview of the implicit capabilities of these elements, including reports on accuracy and convergence, in fairly standard simulation examples. Below some implicit deep drawing examples using high order shells (left) and solids (right) are shown.

A Roadmap to Linear and Nonlinear Implicit Analysis with LS-DYNA

G. Laird (Predictive Engineering)

The default LS-DYNA settings are tailored for running large explicit analyses. For new and even experienced users, it can be challenging setting up an implicit LS-DYNA analysis to match analytical solutions or other standard implicit FEA codes. For example, the default element formulations are based on single-point integration whereas implicit analyses benefits from full-integration. A series of example problems are provided that will allow the simulation engineer to exactly match industry standard implicit codes (complete keyword decks can be found at DYNAsupport.com). Along with these example decks, CPU-scaling results will be presented for each implicit analysis type from linear to nonlinear.

A Layer by Layer Approach for Simulating Residual Stresses in AM

N. Strömberg (Örebro University);
M. Schill (DYNAmore Nordic)

Distortions and residual stresses developed during additive manufacturing (AM) might be so severe that the design does not meet tolerance requirements or even cracks and fail. Predictions of such distortions and residual stresses by finite element analysis are therefore preferable performed early during the design development in order to reduce such problems. One approach for this kind of analysis is to apply techniques from welding simulations letting the Goldak heat source moving over each layer of the material.

However, this is computationally most costly and one might also argue how well this actually represent the AM process. We suggest to simplify this boundary condition to instead apply the heat layer by layer. We mean that this will represent the AM process as well as an approach of using the Goldak heat source and it is much more computational efficient than applying Goldak's heat source. The model is simply obtained by slicing the geometry in several layers using an in-house script.

Then, a volume heat source is applied layer by layer, where each layer is activated starting from the build plate until the final build layer. Sequentially thermomechanical analysis using LS-DYNA is applied, where the heat capacity, conductivity, Young's modulus, Poisson's ratio, expansion coefficient, yield stress, and the hardening modulus are given as functions of temperature. The classical heat equation and J2-plasticity with kinematic or isotropic hardening are adopted by using *MAT_THERMAL_CWM and *MAT_CWM, which makes the activation of the layers straight-forward using the ghost feature implemented in these material models. After all layers are activated and heated, a restart of cooling analysis is performed and, finally, the part is cut from the building plate using a third restart analysis.

The approach is demonstrated for a benchmark of an open cylinder in Inconel 718 showing the development of distortions and residual stresses in all stages of the AM process from heating during building until the final cut from the build plate. We suggest that this benchmark can serve as an experimental setup in order to validate the suggested approach. This is discussed in the paper and is a topic for future work.

Evaluation of Different Thermo-Viscoplastic Material Models under Simultaneous Hot/Cold Forging Conditions

M. Nahrmann, P Kühlmeyer, Prof. A. Matzenmiller (University of Kassel)

The simultaneous hot/cold forging is an innovative metal forming process, which takes advantage of low forming forces in heated parts of the workpiece respectively high geometric accuracy in cold forged areas. For the finite element analysis of such processes, material models are required that take temperature and rate dependent plasticity into account. The material models *MAT_BAMMAN and *MAT_EMMI are available in the material library of LS-DYNA and represent the thermo-viscoplastic characteristics of metals. Furthermore, a user defined material model is implemented into LS-DYNA using the keyword *MAT_USER_DEFINED_MATERIAL_MODEL. The constitutive equations of the user material model are based on an enhanced concept of rheological models, which allows a straightforward interpretation of material behavior. Each material model describes temperature dependent nonlinear isotropic and kinematic hardening, thermally activated recovery effects as well as strain rate sensitivity.

In this contribution, the aforementioned thermo-viscoplastic material models are evaluated under simultaneous hot/cold forging conditions. Therefore, the material parameters are identified with test data of the low alloy steel 51CrV4, the case-hardening steel 16MnCr5, the low carbon steel C15 as well as the aluminum alloy AlMgSi1 by using LS-OPT. The prediction accuracy of each material model is evaluated comparing the simulation and test results on the basis of the mean squared error. Due to the conditions of simultaneous hot/cold forging, the temperature ranges from room temperature up to nearly the melting point. In forging processes, the mutual interaction of the displacement and the temperature field effects a thermo-mechanical coupled problem, which is solved by the staggered solution scheme in LS-DYNA.

Orbital Forming of SKF‘s Hub Bearing Units

E. Omerspahic, J. Facht (SKF)

Orbital forming is an incremental cold forming process that can be used to shape materials such as metals with much lower forming loads compared to pressing. The orbiting tool rotates at a fixed angle to the machine axis and applies both pressure and orbital motion to progressively shaped material. Because the tool is nutating and not rotating, minimal friction is obtained between the workpiece and tool.

In SKF, orbital forming is used by SKF to close automotive hub bearing units (wheel bearings) where a nose is shaped over a ring. This process has been modelled and simulated in LS-DYNA. The model is a mock-up model where rollers and the outer ring are not included. Extensive experimental work has been done to make foundation for the modelling:

- In calibration phase, the FE model has been optimized against experiments to obtain the set of material and process parameters.
- In validation phase, the FE model has been evaluated against several experiments where geometries and process parameters have been varied to confirm the set up from the calibration phase.

The FE modelling consists of three interdependent simulations: Assembling simulation of inserting the SIR on the FIR, orbital forming simulation with the forming tool and springback simulation describing the relaxation process after forming. A good agreement between simulated and experimental results is obtained, and the FE model can be used for design of the orbital manufacturing processes.

Modelling of Hot Rotary Kiln

D. Ramanenka, G. Gustafsson, P. Jonsen (University of Lulea)

Rotary kilns are the central part in the production of many important materials, such as: cement clinkers, lime for paper production and iron-ore pellets for steel making. The main design of a rotary kiln is rather simple – consisting of a cylindrical steel casing and one or several layers of refractories in order to protect the casing from high service temperatures. The dimensions of a rotary kiln vary between some 10 to 180 m in length and 2 to 8 m in diameter. Damage to the refractory lining is common and can potentially lead to long-lasting production shut-downs and high production losses. Due to the harsh inner environment and the large dimensions of the kiln it is difficult to observe and evaluate the kiln while in service – hindering improvement of the kiln. Therefore, it is advantageous to perform computer simulations and potentially improve the design of refractories (bricks), the material choice and operation of the kiln based on the numerical results.

In this work LS-DYNA is used for simulation of a hot rotary kiln insulated with a single lining of refractory bricks.

A cross-section of a kiln of approximately 8 m in diameter is modelled. The physical time to model is up to 65 h. The model is time-dependent and includes thermal expansion and rotation of the kiln. Heat transfer coefficients are calibrated by LS-OPT.

It is found that modelling of a hot rotary kiln is rather successful but several challenges exist. Calculation time can in some cases reach 2-3 weeks (implicit, MPP, 16 cores). Instabilities due to contact associated problems is common.

The created model facilitates decision making regarding e.g. heating/cooling procedure, design changes, maintenance frequency and more.

Hybrid Laminated Glass: Material Characterization and CAE Modelling

B. Feng (Jaguar Land Rover)

Different from standard laminated glass which is made of from soda-lime glass, hybrid laminated glass comprises layers of standard soda-lime glass, PVB and chemically toughened glass. Hybrid laminated glass has advantage of weight saving, so it has drawn great attentions to the automotive industry recently. However, its application may have impacts not only to the performances of many attributes, safety, NVH, etc., but also to the CAE modelling method which we have adopted for the standard laminated glass.

This paper discusses the research activity of understanding potential impact of hybrid laminated glass application, the method of material characterizations, CAE modelling development for different safety attributes.

Validation Tests and Simulations for Laminated Safety Glass

M. Sauer, F. Kölble (Fraunhofer EMI);
K. Mattiasson (Chalmers University of Technology);
L. Schmidt (Saint-Gobain Sekurit Deutschland);
S. Wenig (Sika Automotive);
T. Carlberger (University West);
M. Buckley (Jaguar Land Rover)

We present new subsystem validation test setups for laminated windscreen glass and corresponding simulation results. The goal of the recently finished CompMethGlass project, performed in an international consortium with Swedish, German, Swiss and British participants, was the development of improved material models and simulation approaches for laminated safety glass. The load cases considered were direct impact and indirect loading by deformation of the car structure, e.g. corresponding to A-pillar impact or roof crush. Within the project, the validation tests presented here were developed. As the tests were performed both quasistatically and dynamically, they give insight into the cracking behavior of the laminated glass, the resisting forces and the influence of dynamic loading conditions.

The samples used represent a simplified windscreen system and consists of a steel frame on which a curved laminated glass was adhesively bonded. The frames were loaded in two different ways: 1) a PMMA hemisphere hits either the center of the glass or one corner from the inside in a direction normal to the frame or 2) by hitting the steel frame in a direction parallel to the plane of the frame, such that the laminated glass is loaded indirectly. The glass was observed with high-speed cameras such that the cracking sequence and patterns can be analyzed. Force-displacement curves can be used to quantitatively validate simulations. In the presentation, we will briefly discuss the advantages and limitations of the validation test.

Within the CompMethGlass project, a new user material subroutine for LS-DYNA has been developed in order to model the behavior of the individual glass sheets. The glass model uses thick shell elements and a smeared crack model based on an advanced principal tensile stress criterion to represent glass failure. For modeling PVB and adhesive, we used the LS-DYNA built-in hyper-visco-elastic material model MAT_077H. We will show both qualitative and quantitative comparisons of simulation results with experimental observations. Finally, DoE analysis results illustrate the sensitivity of the model to certain model parameters.

Acknowledgement
This work was realized within the CompMethGlass project, which was supported by VINNOVA, a Swedish government agency, Jaguar Land Rover, Volvo Car Company, Saint-Gobain Sekurit Deutschland GmbH & Co. KG, Dynamore Nordic, Sika Automotive AG, Switzerland, Volvo Group Trucks Technology, Autoliv Development AB and Sika Sverige AB, and performed in cooperation with the supporting companies by Chalmers/Sweden, Fraunhofer EMI/Germany, Glafo/Sweden, SP Technical Research Institute of Sweden/Sweden, Swerea IVF/Sweden and University West/Sweden.

A New Failure Criterion for Laminated Safety Glass

C. Alter, S. Kolling (TH Mittelhessen);
J. Schneider (TU Darmstadt)

The prediction of the head injurie criterion (HIC) for pedestrian protection in automotive applications is still a great challenge. In particular, the simulation of the head impact on windshields requires a reliable FE-modelling technique as well as predictive material laws including the constitutive behavior up to fracture.

In the present work, a new failure criterion for laminated safety glass under low velocity impact is presented. The model is implemented as a user defined material model for shell elements in LS-DYNA. In order to represent the stress intensity in the vicinity zone of a crack tip, a mixture of the element erosion technique and a decrease of strength in the direction of cracks is considered. By that there is no need to determine the current stress intensity factor. The used failure prediction is similar to a major stress criterion and the fracture strength depends on the stress rate and, in addition, on the fracture state of the neighboring elements. The basic strength is derived from the first region of the crack-velocity dependency which is approximated by an empirical power law. The reduction of element strength in crack direction depends on the element size which allows the use of a regular, coarse mesh. Experimental results of head impact tests under different configurations are used for the validation of the present model.

Laminated Amorphous Polymers Subjected to Low-Velocity Impact

A. Rühl, S. Kolling, J. Scheider (TH Mittelhessen);
B. Kiesewetter (Evonik Industries)

The substitution of conventional glass products by polymeric structures bears a huge weight reduction potential for the automotive and aviation industry. Against this background, a polymeric laminate consisting of poly(methyl methacrylate) (PMMA) and thermoplastic polyurethane (TPU) was investigated experimentally and numerically with regard to its impact behavior and applicability.

Basic experiments with PMMA and TPU were used to identify the thermomechanical characteristics of the monolithic materials. Furthermore, PMMA-TPU-PMMA laminates were subjected to impact loadings at velocities up to 5m/s using three-point bending and dart impact tests. The principle behavior, characterized by a distinct post-breakage capacity, was examined. A significant heating of the highly strained interlayer was measured in the post-breakage phase.

Based on the experimental basis, different material models for the Finite Element simulation are presented. These material models are able to capture the temperature and time dependent behavior of the laminate. Further studies regarding modeling techniques for characteristics of laminated structures were conducted.

A final validation experiment, consisting of head-dummy impacts at 10m/s on automotive side windows, was conducted for PMMA and the laminate to investigate their applicability as glass substitution products. The corresponding simulations showed very high agreement to experimental results and exhibited as reliable prediction tools for future developments.

Numerical Modelling of Symmetric and Asymmetric Punching and Post-Punching Shear Responses of RC Flat Slab

N. Ulaeto, J. Sagaseta (University of Surrey)

The design and construction of civil engineering structures take into great consideration the sensitivity of such structures in the event of local failures. Flat slab structural systems are very prone to progressive collapse after the failure of a connection or column. Hence, to improve their robustness the introduction of integrity reinforcement is recommended in Eurocode 2, ACI 318-11 and Model Code 2010.

However, very little investigation has been carried out on the asymmetric post-punching response of these connections or the actual contribution of designed integrity reinforcement to robustness at a system level. Presented in this paper, is a numeric approach developed for modelling the response of isolated RC flat slab test specimens using the finite element (FE) software LS-DYNA. This is in view of their incorporation into system models for both quasi-static and dynamic assessments of robustness in flat slab structures. Quarter FE models of four symmetric isolated RC flat slab specimens with experimental responses available in literature were developed. These quarter FE models were analyzed numerically using a quasi-static displacement controlled approach and their flexural, punching shear and post-punching shear responses observed.

A sensitivity analysis was carried out to obtain the optimum element characteristics for punching shear strength as well as other response criteria. Half asymmetric F.E models of two slab specimens were also developed and analyzed. These provided the asymmetric punching and post-punching shear response of the slab specimens, assuming the loss of an interior column. Results of quarter symmetric FE models gave accurate predictions of slab load-deformation responses, punching and post-punching shear strengths. Maximum percentage differences of 2% and 3% were obtained when comparing test and FE results of symmetric slab specimens for peak punching and post-punching shear strengths respectively. Asymmetric FE models gave post-punching shear strengths lower than values obtained from tests on symmetric specimens. Robustness of flat slab structures after the loss of an internal column could be significantly overestimated where models adopted do not take into consideration such reductions in post punching shear strength.

The results presented validate the use of this FE approach on LS-DYNA to predict the response of concrete flat slab connections.

Evaluation of Advanced Element Formulations for Failure Prediction of Highly Complex 3D-Printed Parts

S. Mohapatra (Sabic Research & Technology Center)

Additive manufacturing has moved beyond rapid prototyping and is now starting to be used for manufacturing of functional parts such as in aircraft engines and medical transplants. This has resulted in a paradigm shift in design requirements for 3D printed parts from simply aiding in rapid physical visualization of designs to sustaining operating loads. Furthermore, along with the functional load bearing requirements, 3D printed designs should demonstrate benefits of lower weight and cost which is possible through innovative shapes often arrived at through detailed topology optimization. However, these innovative shapes often offer significant challenges in predictive performance evaluation due to their geometric complexity. This is amplified for structures where failure prediction under operating environment is a part of the design process such as in automotive applications.

Typically, explicit solvers such as LS-DYNA are used to predict highly non-linear events such as component failures. However, the highly complex shapes arrived through topology optimization and manufactured through 3D-printing process are extremely cumbersome to model with traditional elements used in explicit solvers which are hexahedral in shape. On the other hand, tetrahedron elements can model such shapes with considerable ease, but are avoided in classical explicit analysis methodology due to loss of simulation accuracy. LS-DYNA has recently introduced higher-order tetrahedron elements which has the potential to reduce the loss of accuracy while holding onto the ease of modeling. But this benefit comes with a significant increase in simulation time due to the higher order integration approach.

This paper deals with evaluation of the suitability of higher order tetrahedron elements to model highly complex shapes typical of additive manufacturing process for failure prediction of a highly complex 3D-printed specimen part representing a commonly used structural element. The predictions are compared with actual physical test results which point to satisfactory performance of the element formulation for such use. Additionally, recommendations are made to improve the material property specifications from 3D-printing material suppliers to further improve the simulation fidelity.

Reduced Ductility due to Local Variation in Material Properties for 3D-printed Components

T. Tryland (Sintef Raufoss Manufacturing)

It is often useful to have a physical model to display geometry as an alternative to the 3D-model on a computer screen. In addition, 3D-printed components may work well to evaluate the assembly process. The question here is whether parts that are manufactured in this way have representative ductility to give valuable results in structural tests where the material is loaded outside the elastic regime. There is a wide range of processes and printers available, and is it possible to get more realistic material properties with an expensive machine compared with a simpler one? It is likely that 3D-printed components may have local variation in the material properties, and a study on an aluminum alloy for crashboxes shows that this phenomenon may reduce the ductility [1]. Uniaxial tensile tests may not detect the effect due to sufficient strain hardening for small strains. It may be better to use a shear test that evaluates the material into a higher degree of deformation [2]. However, axial compression with a sensitive geometry may clearly demonstrate the effect when the lowest of several deformation modes at the same force level suddenly wins and this result in a brittle behavior.

Note that axial compression of a circular tube defined by length, diameter and thickness L = 2D = 60T was chosen with purpose to get a sensitive folding pattern and thereby challenge the simulation tool to capture this. This requires a combination of element formulation and element mesh that represent correct local and global stiffness even at severe deformation. It also means that the whole component stores elastic energy that is suddenly released when the capacity is met. The result may be either a brittle behavior in case the ductility is too low or more or less nice folding in case the ductility is sufficient. However, it is important to remember how geometry and material interact. Aluminum has elastic modulus about 70 GPa, and the result is four concertina rings when this tube is compressed to half of its initial length. Polymer has significantly lower stiffness, and 4-lobe buckling is more likely.

Numerical simulations and test results with axial compression of a sensitive tube geometry defined by length, diameter and thickness L = 2D = 30T shows that 3D-printed components may behave brittle due to local variation in material properties that is above a certain limit.

Literature

[1] Tryland, T., Berstad, T.: "Keep the Material Model simple with input from Elements that predict the Correct Deformation Mode", 10th European LS-DYNA Conference, Würzburg, Germany, 2015.

[2] Tryland, T., Berstad, T.: "A Simple Shear Test to Evaluate Material Ductility based on Specimens cut from Thin-Walled sections", 11th LS-DYNA Forum, Ulm, Germany, 2012.

A 3D Discontinuous Galerkin Finite Element Method with the Bond-Based Peridynamics Model for Dynamic Brittle Failure Analysis

W. Hu, B. Ren, C.T. Wu, Y. Guo, J. Wu (LSTC)

Peridynamics is a new nonlocal theory that provides the ability to represent displacement discontinuities in a continuum body without explicitly modelling the crack surface. In this paper, an explicit dynamics implementation of the bond-based peridynamics formulation is presented to simulate the dynamic fracture process in 3D elastic solid. Based on the variational theory, the Discontinuous Galerkin (DG) approach is utilized to formulate the classic peridynamics governing equation.

As a result, the spatial integration can be carried out through finite element approach to enforce the boundary conditions, constraints, contacts as well as to handle the non-uniform mesh in the engineering practices. The classic material parameters, such as the elastic modulus and fracture energy release rate are employed for the determination of material response and failure in brittle material. Several numerical benchmarks are conducted to invest the convergence and mesh sensitivity of simulations of dynamic crack propagation process with different refinements.

The results demonstrate that the proposed peridynamics formulation can capture the 3D dynamic crack process in brittle material effectively and accurately including multi-crack nucleation, propagation and branching.

Characterization and Modeling of Engineering Friction and Wear with LS-DYNA

S. Dong (Ohio State University);
A. Sheldon (Honda R&D Americas)

In mechanical systems, friction and wear usually take place concurrently between two surfaces when they slide or roll relative to each other. Friction exhibits resistance to the relative motion which may vary with different normal loads and relative velocities, while wear is the removal and displacement of the materials from one or both surfaces.

The initiation of significant surface wear, in turn, changes the friction, and therefore the overall contact properties. This occurs more often when there are differences in hardness and/or surface roughness between the two surfaces. This paper presents experimental characterization of friction and surface wear between different material combinations. Materials tested, from hard to soft, include coated steel, plastics, ATD rubber, and roof linear fabric. Static and dynamic friction coefficients are measured using a pin-on-disc tribometer under various normal loads and linear velocities. Surface wear, if significant, is characterized using an optical profilometer or a microscope.

It was found that the friction coefficient decreases when surface wear first takes place. Part of the displaced material gets stuck in between the two surfaces and acts as a third-body solid lubricant. This type of lubrication mechanism plays more significant role when the material opposite to the worn material is harder. The modeling of friction and wear is then implemented in LS-DYNA. The relationship between friction coefficients, normal stress, and linear velocity is defined in *CONTACT and the wear properties are defined in *CONTACT_ADD_WEAR. Parametric studies are then performed to study the influence of wear and friction properties in crash models.

Numerical Model to Predict Kickback for Angle Grinders

G. Fleury (INRS)

Angle grinders are powertools used widely by companies and craftsmen for grinding and cutting applications. The operator must comply with the correct operating procedures and conditions, including the choice of an appropriate tool (cutting disc, grinding wheel, backing pad or brush), to ensure his safety. Even so, some working conditions can cause the rotating tool to be either pinched or snagged. As the rotational kinetic energy of the wheel decreases, a fraction of the energy is lost due to friction between the wheel and the workpiece, but if sliding occurs, some kinetic energy may be transmitted to the grinder leading to it being ejected from the point at which it binds with the workpiece. This sudden reaction of the powertool, known as “kickback”, can, in rare cases, cause serious or even fatal injuries to the worker. Through a working group, INRS, the French Research and Safety Institute for the Prevention of Occupational Accidents and Diseases, is investigating kickback events to identify the working conditions that are most likely to cause it.

Preliminary research led to the development of a prototype test bench dedicated to reproducing kickback events in the laboratory. The test consists in pinching the rotating wheel of a grinder and observing the resulting ejection of the powertool. In order to improve our test bench and to increase our understanding of the mechanical phenomena leading to grinder ejection, a numerical model using the functionalities of LS-DYNA was developed. The model input parameters describe the features of the rotating wheel (mass, diameter, moment of inertia and initial angular velocity) and the features of the angle grinder (mass, moment of inertia, and position of the centre of gravity). The magnitude of the pinching force applied to the rotating wheel, the initial position of its application point and the coefficient of friction between the wheel and the pinching device are also taken into account by the numerical simulation. The kinetic energy of the grinder at the time of ejection and the direction of ejection are calculated in order to assess a kickback event. The task of this paper is to present the model and the results of the parametric study performed to assess the effect of each parameter on the magnitude and the direction of kickback.

Biotex BigBag Simulation – LS-DYNA Airbag Tool Unusual Application

C. Weinberger, B. Hirschmann (4a engineering);
J. Eichelter (Franz S. Huemer)

Verification and Validation of LS-DYNA for the Transport and Storage of Radioactive Materials

G. Marchaud, V. Saint-Jean (Areva)

For 50 years, AREVA TN has been supplying customer-focused, innovative transportation and storage solutions for radioactive material with the highest levels of safety and security.

Transportation and storage casks are designed to comply with stringent regulations. For instance, a cask designed to transport radioactive material may be required to withstand a 9m drop onto a flat unyielding target.

AREVA TN performs LS-DYNA analyses to evaluate the crashworthiness of casks and to reduce the number of costly real tests. Such a methodology relies on the capability of the computer code to model the main physical phenomena that occur in a cask and its content when they are subject to a transient mechanical event.

The validity of LS-DYNA is confirmed by comparing its results with reference results, for a variety of test cases covering these phenomena. The reference results are obtained either analytically or from real tests. AREVA TN defined specific test cases to validate:

- The main constitutive laws suitable for shock-absorbing materials (e.g. woods) and containment vessel materials (namely metals),
- The time integration scheme, with a view to the conservation of total energy,
- The implementation of geometric nonlinearities (large displacements and rotations),
- The correct representation of mass/stiffness distribution (vibrations, rigid-body displacements),
- Shockwave propagation,
- Energy dissipation in a complex structure composed with components in contact with each other.

Such a qualification is performed every time AREVA TN chooses to use a more recent version of LS-DYNA.

The present paper will focus on a selection of these test cases and present their features as well as their results.

Workshop on
Blast Analysis with LS-DYNA

D. Hilding (DYNAmore Nordic)

An overview is given of the available methods in LS-DYNA to calculate explosive blast loads from conventional explosives on structures for structural engineering purposes. Advantages and disadvantages with the different methods will be pointed out. The focus is on air-blast.

The intended audience is LS-DYNA users interested in methods for blast & explosion simulation with the goal to be able to do strength/performance analysis of blast loaded structures. Time permitting, a short demonstration will be given.

DYNAmore Gesellschaft für FEM Ingenieurdienstleistungen mbH

Industriestr. 2
70565 Stuttgart, Germany
Tel.: +49 (0)711 - 45 96 00 - 0
Fax: +49 (0)711 - 45 96 00 - 29
E-Mail: info@dynamore.de
www.dynamore.de

Subsidiaries in Sweden, France, Italy and Switzerland.

Livermore Software Technology Corp. (LSTC)

7374 Las Positas Road
Livermore, CA, 94551 USA
Tel.: +1-925-449-2500
Fax: +1-925-449-2507
www.lstc.com