Book of Abstracts

13th LS-DYNA EUROPEAN CONFERENCE & USERS MEETING – ONLINE AND ON-SITE

5 - 7 October 2021 ▪ Ulm, Germany

Diamond Sponsor

ARUP

Platinum Sponsor

intel

FUJITSU

BOOK OF ABSTRACTS

13th LS-DYNA EUROPEAN CONFERENCE & USERS MEETING – ONLINE AND ON-SITE

5 - 7 October 2021 – Ulm, Germany

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 978-3-9816215-7-0

SPONSORS

Diamond

Platinum

Gold

Silver

MATERIALS: TEST / CALIB.

SIMULATION METHODS I

FORMING I

OPTIMIZATION I

Room 6 "Fulda", Tuesday 5 October, 10:00 – 12:25 Page

SIMULATION MODELS I

Room 7 "Gartensaal", Tuesday 5 October, 10:00 – 12:25 Page

COMPOSITES I

Room 9 "Virtual Room", Tuesday 5 October, 10:00 – 12:25 Page

CRASH II

Room 1 “Keplersaal“, Tuesday 5 October, 13:45 – 15:25 Page

CCUPANT SAFETY I

Room 2 “Bonn“, Tuesday 5 October, 13:45 – 15:25 Page

MATERIALS: MODELLING I

Room 3 "Würzburg", Tuesday 5 October, 13:45 – 15:25 Page

SIMULATION METHODS II

Room 4 "Nürnberg", Tuesday 5 October, 13:45 – 15:25 Page

FORMING II

Room 5 "Hannover", Tuesday 5 October, 13:45 – 15:25 Page

IMPACT / BLAST I

Room 6 "Fulda", Tuesday 5 October, 13:45 – 15:25 Page

SIMULATION MODELS II

Room 7 "Gartensaal", Tuesday 5 October, 13:45 – 15:25 Page

OPTIMIZATION II

Room 9 "Virtual Room", Tuesday 5 October, 13:45 – 15:25 Page

CRASH III

Room 1 "Keplersaal", Tuesday 5 October, 16:00 – 18:05 Page

OCCUPANT SAFETY II

Room 2 "Bonn", Tuesday 5 October, 16:00 – 18:05 Page

MATERIALS: MODELLING II

Room 3 "Würzburg", Tuesday 5 October, 16:00 – 18:05 Page

IGA

FORMING III

IMPACT / BLAST II

Room 6 "Fulda", Tuesday 5 October, 16:00 – 18:05 Page

MULTIPHYSICS I

Room 7 "Gartensaal", Tuesday 5 October, 16:00 – 18:05 Page

GEOMECHANICS

CRASH IV

COMPOSITES II

MATERIALS: MODELLING III

Room 3 "Würzburg", Wednesday 6 October, 08:30 – 10:00 Page

SIMULATION METHODS III

Room 4 "Nürnberg", Wednesday 6 October, 08:30 – 10:00 Page

ELASTOMERS / POLYMERS

Room 5 "Hannover", Wednesday 6 October, 08:30 – 10:00 Page

IMPACT / BLAST III

BATTERY I

COMPOSITES III

CLOUD COMPUTING

Room 3 "Würzburg", Wednesday 6 October, 10:45 – 12:25 Page

SIMULATION METHODS IV

Room 4 "Nürnberg", Wednesday 6 October, 10:45 – 12:25 Page

MULTIPHYSICS II

Room 5 "Hannover", Wednesday 6 October, 10:45 – 12:25 Page

CRASH V

Room 7 "Gartensaal", Wednesday 6 October, 10:45 – 12:25 Page

CFD I

Room 8 "Travemünde", Wednesday 6 October, 10:45 – 12:25 Page

BATTERY II

COMPOSITES IV

CFD II

CRASH VI

MULTIPHYSICS III

SPONSOR INFORMATION

ARUP

Oasys Software – PRIMER, D3PLOT, T/HIS, REPORTER & SHELL

Oasys Suite Workshop – Overview of Latest Features

Developed specifically for LS-DYNA, the Oasys Suite is at the leading edge of pre and post-processing software and is used worldwide by many of the largest LS-DYNA customers. The latest version of the software, Oasys Suite 18.0, has exciting new features and updates to help you power through your workflow and achieve high-quality results.

Key highlights include:

- D3PLOT Viewer – a brand new 3D web viewer to transform the way you review, communicate, and deliver engineering analysis
- New seatbelt creation tools for fully fabric seatbelts
- Further improvements and additions to the pedestrian protection tools
- Creation and management of SPR (Self Piercing Rivet) type connections
- New and improved automatic REPORTER templates, including MPDB
- Easier access to the powerful LS-DYNA implicit analysis solver through a new tool for quick and easy set-up of implicit analyses
- Improved support for human body models

Join us for a session explaining more about some of these new features. Note this session is a recorded presentation however, Oasys team members will be on hand to answer any questions you have about the Oasys Suite.

Oasys

LS-DYNA ENVIRONMENT

PRIMER | LS-DYNA® | D3PLOT | T/HIS | REPORTER | SHELL

Prepare ● Analyse ● Visualise ● Process ● Report ● Workflow

www.arup.com/dyna

SPONSOR INFORMATION

Shaping Tomorrow with You

The Fujitsu Group's purpose is "to make the world more sustainable by building trust in society through innovation". Since being founded in Japan in 1935, we have continued to demonstrate our technological prowess and persistent pursuit of innovation. As a world-leading digital transformation partner, our business structure is aligned with the modern digital world.

Using a wide portfolio of trusted technology services, solutions and products, we work with our customers to co-create solutions that help them on their journey to enterprise-wide digitalization. At the same time, we are using technology and working with our customers and wider ecosystem partners to help solve social issues. A recent example is the development of the world's fastest supercomputer Fugaku that has been designed in co-creation with RIKEN.

By making a contribution to the United Nations Sustainable Development Goals we are helping to transform our world and build an inclusive, sustainable and trusted society. Approximately 126,000 Fujitsu people support customers in more than 100 countries. Fujitsu Limited (TSE: 6702) reported consolidated revenues of 3.6 trillion yen (US $33 billion) for the fiscal year ended March 31, 2021. For more information, please see http://www.fujitsu.com.

www.fujitsu.com

Power New Discoveries with Intel® HPC Technologies

From autonomous driving to disease research to clean energy, some of the world's most important discoveries depend on high performance computing (HPC). Intel offers a unique, comprehensive HPC portfolio to help customers reach new insights faster, powering discovery and innovation. Intel-based systems deliver outstanding performance for applications of all kinds. With deep learning acceleration built directly into the chip, Intel® CPU hardware supports both HPC and AI workloads. And Intel's widely used HPC toolkit and multi-architecture programming model, plus a broad global ecosystem, help HPC users get more value from their hardware and application investments.

3rd Generation Intel® Xeon® Scalable Processors deliver outstanding Ansys LS-DYNA performance compared to the previous generation. LS-DYNA users have seen up to 48 percent Gen-to-Gen performance improvement this includes the popular Intel Math Kernel Library (Intel MKL), which ensures new instructions like AVX-512 work seamlessly for developers. Users have seen Intel® Optane™ Persistent Memory yield double-digit performance improvements as well. Ansys collaborates closely with Intel to optimize LS-DYNA at scale on Intel architecture. Intel Xeon processors also offer built-in configuration flexibility with Intel® Speed Select Technology. Discover new ways to extract better performance from LS-DYNA with Intel hardware and software.

www.intel.com

SPONSOR INFORMATION

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. Through our strategy of Pervasive Engineering Simulation, we help the world's most innovative companies deliver radically better products to their customers.

By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and create products limited only by imagination. Founded in 1970, Ansys is headquartered south of Pittsburgh, Pennsylvania, U.S.A. Visit www.ansys.com for more information.

Ansys and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

www.ansys.com

BETA CAE Systems transformed CAE by introducing revolutionary automation software tools and practices into Simulation and Analysis processes almost 30 years ago.

Committed to our mission to enable engineers to deliver results of high value, we continue to offer state-of-the-art, high-performance software and best-in-class services. Our simulation solutions liberate low risk and high Return-On-Investment innovation.

The ground-breaking technology, the excellent services and our high standards of business ethics are the three pillars on which BETA was founded and grows since then.

Our passion for engineering, our drive for excellence, and our loyalty to customers and partners, are the key ingredients of our success. We first established our reputation in the Automotive sector and now we are proud of the deployment of our software also in the Aerospace, Defence, Biomechanics, Electronics, Energy, and other industries. Our solutions exceed their requirements in all the simulation disciplines, and allow for the development of the right product, for the right market, at the right time.

www.beta-cae.com

SPONSOR INFORMATION

4a engineering GmbH is a technically oriented Research and Development company with a focus on plastics engineering and materials science. The core competence of 4a engineering GmbH resides in concept finding and optimization of product ideas based on the profound understanding and interpreting of physical and mechanical processes. 4a engineering GmbH works with a wide range of partially specially developed simulation software and analyzing methods. We are offering engineering and R&D services as well as our products VALIMATTM, FIBERMAPTM, MICROMECTM and IMPETUSTM.

IMPETUSTM sets new standards in the area of dynamic material characterization. The 4a-inhouse-developed testing system for the characterization of materials, especially plastics, is based on a pendulum working without additional actuation power. The testing system can be operated on a common office desk located in the Research & Development department.

VALIMATTM is the software solution for managing material tests and for the generation of validated material cards for plastics, composites, metals and foams. The software automatically generates FE-Models for the relevant tests. The parameters for the material card are optimized by an automatic reverse engineering process. VALIMATTM supports the most common FE-solvers and material cards. The functionality was proven by hundreds of projects.

www.4a-engineering.at / http://impetus.4a.at

DatapointLabs was founded in 1995 with a mission to provide scientifically accurate material data for use in engineering design and simulation. With our testing services and software for materials, we help companies build enduring data collections that accurately represent the materials used in their products. To date, over 30,000 materials have flowed through the laboratory, providing data for over 1,200 companies globally.

DatapointLabs offers TestPaks® to provide CAE-ready material properties data for most major simulation software packages. In 2018, DatapointLabs joined the Applus+ Group, a worldwide leader the testing, inspection and certification sector. In 2020, DatapointLabs celebrated 25 years of putting materials into product design.

www.datapointlabs.com

SPONSOR INFORMATION

CASCATE GmbH was founded in June 2017 as a wholly-owned subsidiary of DYNAmore GmbH, based in Stuttgart. Motivated by the possibility of entering into a sales partnership with Siemens PLM Software, CASCATE GmbH offers software solutions in the area of simulation/CAE around the software packages:

- STAR-CCM+®
- SimcenterTM
- FemapTM

With a team of experienced engineers, we are happy to provide our customers with the appropriate support and services.

www.cascate.de

GNS has been active in the field of computer-aided simulation since 1994. The main business areas of the company include calculation services and the related software development.

The calculation services focus on crash simulation, numerical calculations for statics, strength and vibration behavior of structures as well as numerical simulation of forming and manufacturing processes.

GNS developed the software products Animator4, Generator4, OFSolv and OpenForm. With thousands of licenses sold worldwide, Animator4 is one of the most successful post-processors in the field of finite element structural simulation.

www.gns-mbh.com

SPONSOR INFORMATION

GNS Systems - IT Services for Engineering

GNS Systems offers innovative IT services for virtual product development.

Our portfolio includes customised services in the areas of High Performance Computing, Technical Data Management, Software Engineering and Systems & Application Management. We use state-of-the-art methods and technologies - from Cloud Computing to Data Management & Analytics, Deep Learning & Artificial Intelligence and Virtual Reality. In these areas we plan, implement and operate complex systems and application infrastructures - either on-premise or in the cloud.

Our agile, cross-functional teams work both nationally and internationally for well-known companies in the automotive, aerospace and mechanical and plant engineering industries. Thanks to our many years of experience, we offer our customers comprehensive, detailed consulting and proven implementation.

www.gns-systems.com

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

Recent Developments in LS-DYNA

J. Wang (Ansys/LST)

Complexity of a Solver Change and the Transfer to Project Business

F. Bauer (BMW Group)

Digital Tradition at IWC

P. Steinhäuser (IWC)

Water Shock Analysis

E. de Hoff (Honda)

New Features for Crash in LS-DYNA R13.0

T. Erhart (DYNAmore)

Using history variables in materials to reduce modelling effort and increase model accuracy

Tobias Erhart[1], Michal Styrnik[2]

[1]Dynamore GmbH
[2]BMW AG

In crashworthiness simulation the definition of material properties is one of the key aspects to obtain reasonable results. However, a lot of materials come with properties that either change locally or are generally of stochastic nature. Additionally, production processes (e.g., welding) might change the behavior of certain materials. To overcome the necessity of defining an individual part for each region where material properties differ a new approach was developed. With the new keyword **`*DEFINE_TABLE_COMPACT`** it is now possible to define material properties by means of a multi-dimensional table with arbitrary variables controlling for example the plastic flow curve or the damage behavior. Secondly, the keyword **`*INITIAL_HISTORY_NODE`** enables the user to set these variables individually on each node in the model. This presentation shows possible applications of this approach and the benefits on parametric modelling and simulation.

Modeling of Adhesives in Crash Simulation

M. Helbig (DYNAmore)

Modeling of component failure due to notch effects in press-hardened steel caused by mechanical and thermo-mechanical joints under crash load

Philipp Bähr[1], Silke Sommer[1], Eduard Unruh[2], Gerson Meschut[2]

[1]Fraunhofer - Institute for Mechanics of Materials IWM
Wöhlerstraße 11, 79108 Freiburg, Germany

[2]Laboratory for Materials and Joining Technology (LWF), University of Paderborn
Pohlweg 47-49, 33098 Paderborn, Germany

Abstract

The increasing application of press-hardened strength steel in combination with aluminum sheets in the construction of car bodies results in the use of mechanical joining techniques such as self-piercing riveting and thermo-mechanical joining techniques such as friction and resistance element welding. These joints generally represent a notch within the component. The cause of the notch effect is different for the investigated joining techniques. Riveted joints result in a pierced hole with high plastic strains at the edge. Thermo-mechanical joints in press-hardened steel result in a softening zone around the weld due to the applied heat during the joining process. All of these notch effects can be critical for the structural integrity of a component under crash load. Due to limited computational resources, a detailed modeling of joints within a component simulation is not possible. Thus, simplified modelling approaches such as the *CONSTRAINED_INTERPOLATION_SPOTWELD (model 2) are applied in order to represent the load bearing capacity and failure behavior of the joints. These simplified models are usually used with shell elements for the sheet metals, whereby no notch effects such as pre-damage or differences in the material behavior are taken into consideration. Therefore, crack initiation in plane of the sheet metal due to these notch effects cannot be described in the simulation. Subject of this research project is the development of a modeling approach, which takes material failure due to notch effects at mechanical and thermo-mechanical joints into consideration. Two distinct approaches are developed for the two investigated notch effects. Both of the approaches are based on an explicitly modeled notch zone at the joint. For the mechanical joining techniques, a hole in the press-hardened steel is modeled, with an additionally defined pre-damage for the shell elements surrounding the hole. For the thermo-mechanical joints, a softening zone with modified flow and failure behavior is modeled. The modelling approaches are calibrated based on experimental results of tensile and punch tests, which have been conducted with notched specimens. Additionally *CONSTRAINED_INTER-POLATION_SPOTWELD (model 2) models are calibrated based on the results of LWF-KS-2-specimens in order to include the load bearing capacity and failure behavior of the joint itself. The generated models are validated based on experimental results of component tests, which have failed because of notches due to the investigated joining techniques. It can be shown, that the suggested modeling approach is able to reproduce the load bearing capacity of the joint as well as material failure due to the imposed notch effect under complex loading conditions. Thus, the prediction quality of the simulation can be significantly improved, when the observed notch effects are considered in the setup of the simulation model.

SMILE – Alternative Input Language for LS-DYNA (and Other Solvers)

Bastian Näser[1], Darius Friedemann[2], Jörg Rademann [2]

[1]BMW Group
[2]HTW Berlin

1 Introduction

To ensure today's development cycles for products and components, the main part of the development process has to be supported by numerical simulations. For complex parts, many simulation disciplines have to be considered to meet all the requirements. This results in a usage of different simulation tools with different input file languages. Moreover, a typical simulation engineer has to be an expert for numerical simulation (in most cases for a specific solver), and also an experienced engineer (for a specific development discipline).

2 Unified Simulation Modelling Language

With SMILE (Unified **S**imulation **M**odellI**ng** **L**anguag**E)** at hand, we developed an alternative process to create specific input files out of a universal input file language. The solver specific input files will be composed by a SMILE translator and the three main components of a SMILE model:

1. The model file.
   It contains all the physical properties such as geometry, materials, connection and many more. Numerical settings, which depend on the simulation discipline and tool, are not part of the model file.
2. The configuration file.
   This file describes the load case. It contains the information: what is investigated with the model.
3. Modeling guidelines.
   The modeling guidelines contain all the numerical settings which are necessary for an accurate, predictive and performant simulation. Of course, these modelling guidelines are simulation discipline and tool dependent. Moreover they can be different from department to department inside a company.

The split into these three components enables a template based modeling approach. The user is able to replace the component to be investigated by changing the model file, without changing the configuration file and the modeling guidelines. This approach ensures a reliable process in the simulation workflow. Moreover, the split into model file and modeling guideline is a step to more democratization of simulation. The engineer carrying out the simulation does not have to be a simulation expert. The focus can solely lay on engineering topics. At the end, simulation experts control the numerical settings of the simulation by providing suitable modeling guidelines for this specific task.

3 Features

With the concept of a model file, which can be used for different simulation disciplines and tools, we follow the single source of truth approach for simulation models. This is an enabler for a 100+% model containing the physical properties for all simulation disciplines.
Beside this and the object-oriented approach (split into model file and configuration file) the following features have to be mentioned:

- Level of detail and geometry support.
  SMILE supports the possibility to provide different discretizations for the geometry. Moreover, SMILE can use links to CAD geometry for automatic meshing, isogeometric analysis, surface smoothing or similar features.
- Hierarchical structure.
  SMILE supports a hierarchical structure instead of a flat input file structure. Therefore the structure of the corresponding CAD model can be used.
- Cascading levels of modeling guidelines.
  SMILE supports cascading levels of modeling guidelines.

4 Workflow

The SMILE Workflow can be understood as an additional translating step to create the native solver input file.
This translator takes the three main components of a SMILE model. The model files, which are independent of solver and simulation discipline. The configuration file, determining the load case, the simulation discipline and finally the solver as well as the files containing the modeling guidelines.
The result of this workflow will be demonstrated with two.

5 Examples

The first example simulates a vase. The first simulation extracts the natural frequencies of this component. In the second step, an impact simulation on a table is executed. Both simulations use the same input file, but different configuration files. Solver dependent modeling guidelines ensure the usage of the correct element types.
The second example investigates the deployment process of a simple airbag. To demonstrate the power of the modeling guidelines, the airbag will be inflated with different numerical methods (e.g. uniform pressure, particle) by using the identical model file.
For both examples, native input files for LS-DYNA and Abaqus are created by the SMILE translator.

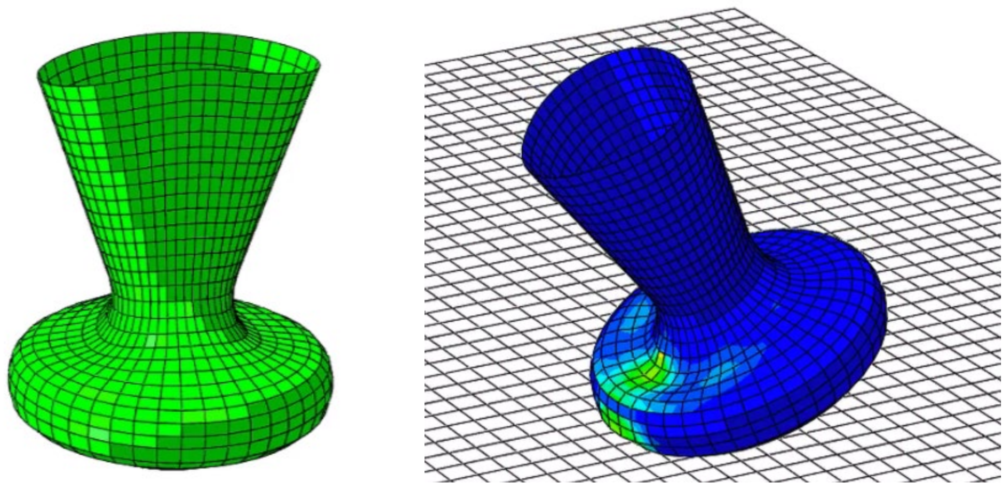


Fig.1: Eigenmode and impact of a vase.

6 Summary

With SMILE as a unified simulation modelling language, we introduced an alternative input language for numerical simulation tools. SMILE improves the simulation workflow with respect to reliability, usability and generalized applicability. This is demonstrated with a POC-implementation for LS-DYNA.

Data Representation of Crash Scenarios by Graph Structures

Anahita Pakiman[1,2], Jochen Garche[1,3], Axel Schumacher[2]

[1]Fraunhofer SCAI
[2] University of Wuppertal
[3] University of Bonn

1 Abstract

In 30 years of crash-worthiness history, car crash simulation has focused on the reliability of the finite element (FE) method in crash behavior prediction. This caused into more detailed simulations, which results in more complexity in the data. Consequently, perception and summary of cause and effect in simulations are time-consuming. Moreover, the growth of computing power has increased the number of simulations. The growth in number and complexity of simulations causes the data to be somewhat unexplored, due to the limited available engineering time.

To find undiscovered trends in simulation results, a data representation approach that is called a vehicle knowledge graph (car-graph) is the basis of this research. The visionary goal of the car-graph ansatz is to assist the engineer during the evaluation of data from previous simulation studies. This assistance envisions searchability, capturing trends, predicting, and recommending solutions. To achieve this, we transfer crash simulation data to a graph database representation to empower machine learning (ML) algorithms. Due to the complexity of the simulation data, storing the input and output of each simulation as raw meta-data is not feasible. The complexity and size of the data prevent ML algorithms from capturing and detecting the trends in crash simulations. Here we present how to simplify and characterize this data to be able to store it as a graph database.

The database generates a network of main input features and output performance characteristics. This simplification and characterization will introduce a new ML-oriented data representation for crash simulation. We investigated classical crash simulation techniques to sub-sample the current FE models. This sub-sampling reduces the data to the key features of the crash simulations from the problem physics. The investigation focuses on coarsening the FE details and solver outputs for each crash simulation. The coarsening introduces features to summarize a simulation to a graph in a way that the graph shape identifies the crash behavior.

The graph empowers trend investigation in the development history chain in two aspects. Firstly, a metric for ML algorithms to quantify the similarity between simulations. The similarity enriches the graph database by link prediction between disconnected simulations. Additionally, we introduce a data representation named platform/vehicle fingerprint. The fingerprint assists in summarizing design space exploration to find new solutions and facilitating cross-platform and/or various load case comparisons.

The distinction in this research is its industrial orientation to shorten the gap between ML methods and the current CAE workflow. The methods were developed on several OEMs development data to evaluate the feasibility and practicality. To illustrate the idea, we studied a sub-model to ease the communication of the idea. Here we will introduce the crash graph by the sub-model. Additionally, we will present results from data exploration of development stages in a China Euro Vehicle Technology AB (CEVT) project.

***KEYWORDS:** crash-worthiness, CAE data management, CAE knowledge, Car knowledge graph, data representation

Analysis of LS-DYNA MOR Approaches for Application in Crash Analysis and Integration in SDM Workflows

Zeidoun El Khatib[1], Uwe Reuter[1], Marko Thiele[2]

[1]TU Dresden
[2]SCALE GmbH

Abstract

The automotive industry is highly dependent on numerical modeling. Companies perform numerical simulations for crash analysis in the preliminary phases of design, because it eliminates the need to perform expensive physical crash tests of prototypes. Optimization, another time-consuming process, is also often performed during the different stages of a project and is only possible using virtual testing. However, optimization adds to the scale of computational time needed in the automotive industry for virtual product development. The longer it takes to perform such numerical processes, the greater the time-to-market of a certain vehicle model and eventually, the higher the cost.

In order to speed up the calculation and assessment process, Model Order Reduction (MOR) techniques are utilized. MOR reduces the complexity of a system while maintaining its input-output behavior to a certain accuracy. These approaches aim to find a reduced order model (ROM) in a reduced subspace $\mathbb{R}^k$ from the full order model (FOM) by the aid of a transformation $\mathbf{T}$. In structural dynamics, the reduced equation of motion becomes $\mathbf{M}_k\underline{\ddot{x}}_k(t) + \mathbf{C}_k\underline{\dot{x}}_k(t) + \mathbf{K}_k\underline{x}_k(t) = \underline{f}_k(t)$ where the system matrices $\square_k$ are obtained using $\square_k = \mathbf{T}^T\square\mathbf{T}$ and the reduced vector of forces $\underline{f}_k(t)$ as $\underline{f}_k(t) = \mathbf{T}^T\underline{f}(t)$. By solving this system of equations, the reduced vector of displacements $\underline{x}_k(t)$ is obtained, from which the full vector of displacements $\underline{x}(t)$ can be approximated and reconstructed using $\underline{x}(t) = \mathbf{T}\underline{x}_k(t)$.

There exist different MOR methods which handle linear and non-linear (geometry, material, and boundary) systems. Methods handling linearity are much more mature and have been around for a long time. Although, a crash exhibits several non-linearities, there exist large portions of the vehicle that behave linearly, thus are candidates for linear MOR.

LS-DYNA incorporates two linear reduction approaches, Superelement (SE) and Linearized Flexible Body (LFB). SE can be based on static condensation or Craig-Bampton reductions while LFB can be based on modal truncation or Craig-Bampton, empowered by rigid-body-motion displacement[1]. In this work, SE based on Craig-Bampton and LFB based on Modal Truncation were first evaluated on a structural frame (see Fig. 1).

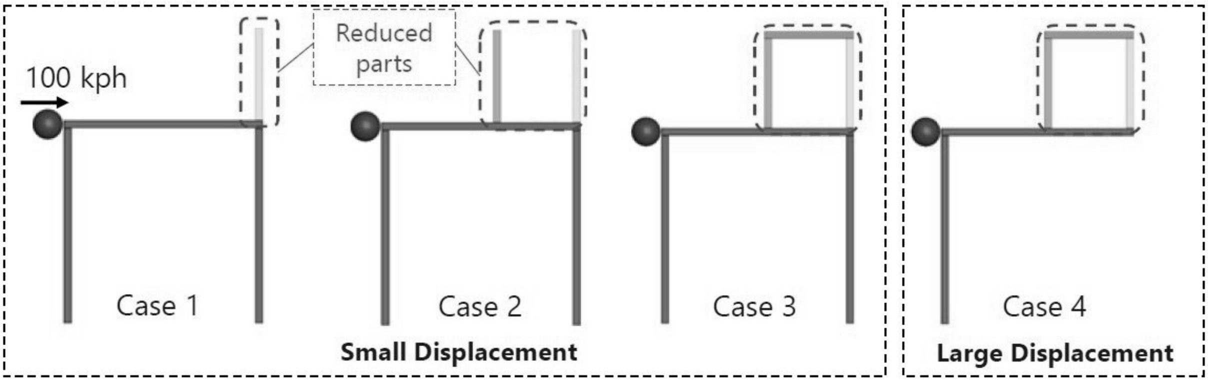


Fig. 1: Structural frame cases used for testing the MOR approaches in LS-DYNA

[1] A detailed explanation of these methods can be found in Z. Qu. *Model Order Reduction Techniques: with Applications in Finite Element Analysis*. Springer-Verlag, London, 2004.

Several observations were made regarding the performance of these methods. In general, SE outperforms LFB in small displacement problems but fails to provide a good approximation in large displacement problems. In some cases, reductions exceeding 60% were achieved with high accuracy. In addition, SE handles rigid bodies and contact algorithm much better than LFB. Nevertheless, LFB proved useful and provided reductions exceeding 40% with high accuracy.

The LFB approach was then tested on a Toyota Yaris model in a side-crash-scenario (see Fig. 2). It was noticed that several factors influence its performance such as: contact between the reduced parts, choice of eigenmodes, scalability of LS-DYNA, number of processors, model decomposition logic (see Fig. 3), and computing environment.

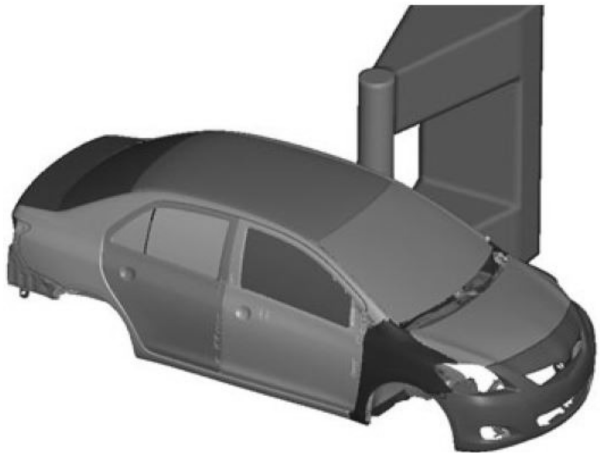


Fig. 2: Yaris Model side crash scenario

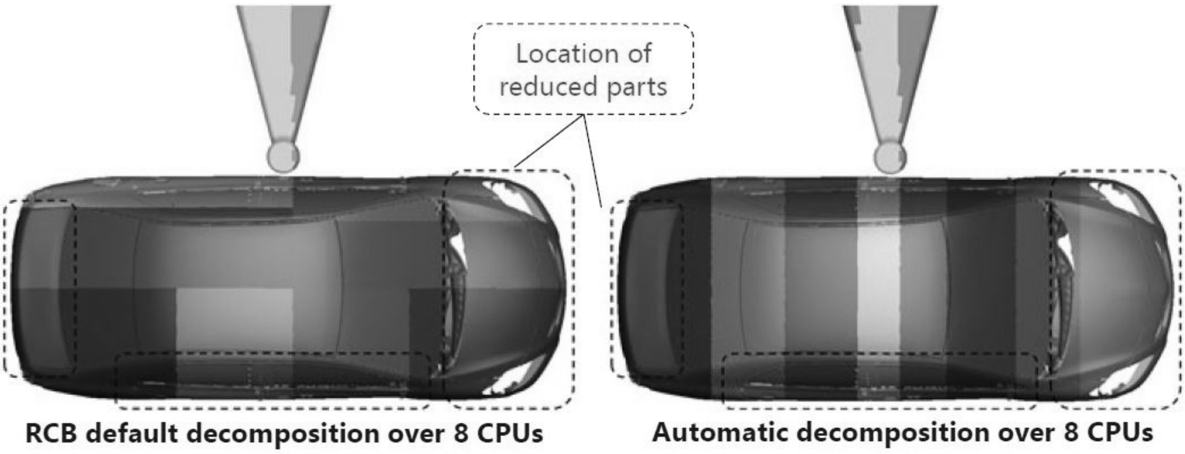


Fig. 3: Two variants of model decomposition logic

Another challenge of using MOR is its integration into simulation. Depending on the method (SE, LFB, etc.) used, a model must be modified in a particular manner in order to properly incorporate reduced model parts. It is essential that no redundancies or contradictions exist in the model, such as having a part in both, reduced and full order representation. These could cause a series of errors that would end up increasing the working time rather than reducing it, and complicating an engineer's life rather than easing it. To overcome such a challenge, a simulation data management (SDM) system can be used to automate the integration process. Using an SDM, the engineer can easily manipulate a model, deciding which parts (sub-models) are to be represented using an MOR-variant and which parts are to be retained in full order representation. Furthermore, an automated post-processing can be used to identify areas (sub-models) exhibiting linear behavior. These sub-models would then be candidates for linear MOR. In this work, the SDM system "SCALE.sdm (LoCo)" was used to set up the whole process, submit runs to the HPC, and perform all the required automated post-processing.

Automatic Outlier Detection for Crash Simulation Results

D. Borsotto, L. Jansen, V. Krishnappa, S. Mertler, C. A Thole (SIDACT)

To cope up with the ever growing amount of simulation runs being performed, tools and techniques are needed to make use of the huge amount of simulation data being stored. While current Simulation Data Management systems and the IT infrastructure already allows to store and access huge datasets and would facilitate putting this into action for analysis, the user usually only has tools and the time to make rather straight forward model to model comparisons between current model versions and their immediate predecessors. To take analysis capabilities and model development a leap forward it is necessary to also make use of whole model development branches to learn from the gathered simulation information. Having developed a database which allows querying simulation information of thousands of simulations, we will on the one hand give some insights about challenges like the management of changing designs during the development phase and how to still being able to compare geometrically altered parts. On the other hand we will point out how to make use of the simulations of the entire model development branch to speed up and improve the engineers daily work. The approach currently being developed illustrates how it is possible to automatically detect anomalies/outliers within the crash deformation behaviour pinpointing exactly to the location in space and time where the model is showing unknown or unwanted deformation patterns. While in daily work the engineer often only has time to compare single simulations with each other this approach shows how to compare the current simulation with hundreds of predecessors at a time.

Yield Locus Exponent Modelling of Packaging Steel for an Optimized Simulation of Limited Dome Height Experiments

Fabian Knieps[1], Ioana Moldovan[1], Benjamin Liebscher[1], Manuel Köhl[1], Marion Merklein[2]

[1]thyssenkrupp Rasselstein GmbH, Koblenzer Straße 141, Andernach, Germany
[2]Institute of Manufacturing Technology, Egerlandstr. 13, Erlangen, Germany

In packaging steel forming processes, conditions in-between plane strain and biaxial tension are mostly relevant as they lead to failure in deep drawing applications and characterize e.g. the process of the rivet forming in easy-open end applications. To receive precise simulation results in finite element analysis, it is important to consider an accurate modelling of the yield locus in this area. Complex anisotropic yield functions like e.g. Yld2000-2d which was proposed by Barlat and is implemented in LS-DYNA using keyword ***MAT_133** do not consider the characterization of this area and maintain an uncertain variable by the yield locus exponent. Therefore, Lenzen and Merklein [1] developed an approach to model the plane strain behaviour by means of the yield locus exponent using elliptic bulge tests. In the present work, this approach was applied to determine the yield locus exponent of the packaging steel TH415 precisely and to simulate limited dome height experiments, which represent the forming state in packaging steel applications very well. For a further improvement, anisotropic behaviour in biaxial tension was considered as well. The yield exponent showed a strong influence on the simulation results of the friction induced forming ring in limited dome height experiments and the characterization of the yield exponent via elliptic bulge tests revealed great improvement to determine this parameter precisely. To implement the optimized characterization via the keyword ***MAT_133** in LS-DYNA simulations, the yield locus model was directly parametrized by the underlying α-values and the yield exponent which were determined throughout an optimization process minimizing the residuum between experimental data and the model prediction.

1 Introduction

Research activities in the packaging food market are mainly driven by down gauging interests in order to reduce costs via weight reduction and an increasing importance of a minimized carbon footprint. To fulfill stability requirements as for example in aerosol applications, the development of higher-strength packaging steels has become an important market driver. Accelerating the process of tolling adaption on to the requirements of high-strength packaging steels and to minimize the trial-and-error effort, finite element simulation have become an important tool in this field. The distinctive description of the plastic material behavior is mandatory to receive precise simulation results in sheet metal forming. Beside the implementation of a flow curve to describe the strain hardening in uniaxial stress state, the usage of an appropriate yield locus model is necessary to describe even the plastic anisotropic material behavior. Knieps et al. already showed the specific requirements in the characterization of packaging steel and even the necessarily of an appropriate flow curve description and the selection of a complex anisotropic yield locus model became obvious [2]. In commercial LS-DYNA code, several material models are implemented which fulfill these requirements. Table 1 gives an overview about two common material models in LS-DYNA code and the required material parameters. One of the more simple models was proposed by Hill in 1948 [3] and considers already plastic anisotropy via Lankford coefficients and is implemented by the keyword ***MAT_122** in LS-DYNA commercial code. However, there is a lack in the description of material behaviour in the area between biaxial tension and plane strain especially for material with Lankford coefficients smaller than one and a biaxial strength greater than the uniaxial strength [4]. This lack is overcome by the even more complex anisotropic yield locus model Yld2000-2d proposed by Barlat in 2000 [5], which considers even different strength behaviour in different orientations and experimental biaxial data. This model can be applied in LS-DYNA code using the keyword ***MAT_133.** It is based on two linear transformations of the deviatoric stress tensor with eight free parameters $\alpha_1,\dots,\alpha_8$ (e.q. 1-5). Two options are available in the LS-DYNA keyword to parametrize the yield function Yld2000-2d. On the one hand, there is the possibility to calibrate the material card by the experimental data σ_0, σ_{45}, σ_{90}, σ_{xx}, σ_{yy}, σ_{xy}, r_0, r_{45}, r_{90} and the responding α-values are calculated by LS-DYNA. On the other hand, LS-DYNA offers the possibility to parametrize the model directly by the underlying values of α_1,.., α_8.

$$\phi = \varphi' + \varphi'' = |X_1' - X_2'|^a + |2X''_1 + X_2''|^a + |X_1'' + 2X''_2|^a \tag{1}$$

$$\begin{pmatrix} X_{11}' \\ X_{22}' \\ X_{12}' \end{pmatrix} = \begin{pmatrix} C_{11}' & C_{12}' & 0 \\ C_{21}' & C_{22}' & 0 \\ 0 & 0 & C_{66}' \end{pmatrix} \begin{pmatrix} s_{xx} \\ s_{yy} \\ s_{xy} \end{pmatrix} \quad \begin{pmatrix} X_{11}'' \\ X_{22}'' \\ X_{12}'' \end{pmatrix} = \begin{pmatrix} C_{11}'' & C_{12}'' & 0 \\ C_{21}'' & C_{22}'' & 0 \\ 0 & 0 & C_{66}'' \end{pmatrix} \begin{pmatrix} s_{xx} \\ s_{yy} \\ s_{xy} \end{pmatrix} \tag{2}$$

$$\boldsymbol{X}' = \boldsymbol{C}'.\boldsymbol{s} = C'.\boldsymbol{T}.\boldsymbol{\sigma} = \boldsymbol{L}'.\boldsymbol{\sigma} \qquad \boldsymbol{X}'' = \boldsymbol{C}''.\boldsymbol{s} = C''.\boldsymbol{T}.\boldsymbol{\sigma} = \boldsymbol{L}''.\boldsymbol{\sigma} \tag{3}$$

$$\boldsymbol{T} = \begin{pmatrix} 2/3 & -1/3 & 0 \\ -1/3 & 2/3 & 0 \\ 0 & 0 & 1 \end{pmatrix} \tag{4}$$

$$\begin{pmatrix} L_{11}' \\ L_{12}' \\ L_{21}' \\ L_{22}' \\ L_{66}' \end{pmatrix} = \frac{1}{3} \begin{pmatrix} 2 & 0 & 0 \\ -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 2 & 0 \\ 0 & 0 & 3 \end{pmatrix} \begin{pmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_7 \end{pmatrix} \quad \begin{pmatrix} L_{11}'' \\ L_{12}'' \\ L_{21}'' \\ L_{22}'' \\ L_{66}'' \end{pmatrix} = \frac{1}{9} \begin{pmatrix} -2 & 2 & 8 & -2 & 0 \\ 1 & -4 & -4 & 4 & 0 \\ 4 & -4 & -4 & 1 & 0 \\ -2 & 8 & 2 & -2 & 0 \\ 0 & 0 & 0 & 0 & 9 \end{pmatrix} \begin{pmatrix} \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \\ \alpha_8 \end{pmatrix} \tag{5}$$

However, in the standard procedure to calibrate this model, it is not possible to model the area of plain strain precisely and there is still one free parameter remaining by the yield exponent, which is standardized set to six for ferritic steel [4]. Lenzen and Merklein [1] used this fact to optimize the Yld2002-2d yield exponent with elliptic bulge tests receiving a much more precise modeling of the plane strain forming behaviour demonstrated in cruciform cup drawing experiments for materials in a thickness of one millimeter.
However, even for thin packaging steel sheets the demand of a precise plane-strain modelling is relevant and not investigated yet. In this context, Moldovan [6] demonstrated already the need to determine the yield locus exponent of the yield locus model Yld2000-2d precisely in order to obtain precise results by simulating limited dome height experiments. These experiments are well suited to represent the forming conditions in applications like the rivet forming in easy-open end forming processes. In addition, Knieps et al. demonstrated the suitability of this experiment for the requirements of packaging steel [2].
Thus, aim of the present study was to apply the approach of Lenzen et al. on the requirements of packaging steel to simulate limited dome height experiments more precisely. To extend this method, biaxial anisotropy coefficients were considered, as well.

Model	**Material no.**	σ_0	σ_{45}	σ_{90}	r_0	r_{45}	r_{90}	σ_b	r_b
Hill’48	**`*MAT_122`**	x	-	-	x	x	x	-	-
Yld2000-2d	**`*MAT_133`**	x	x	x	x	x	x	x	x

Table 1: Commonly used material models for forming simulation available in LS-DYNA

2 References

[1] Lenzen, M., Merklein, M., 2018. Improvement of Numerical Modelling Considering Plane Strain Material Characterization with an Elliptic Hydraulic Bulge Test. Journal of Manufacturing and Materials Processing 2 (1), 6.

[2] Knieps, F., Liebscher, B., Moldovan, I., Köhl, M., Lohmar, J., 2020. Characterization of High-Strength Packaging Steels: Obtaining Material Data for Precise Finite Element Process Modelling. Metals 10 (12), 1683.

[3] Hill, R., 1948. A theory of the yielding and plastic flow of anisotropic metals. Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences 193 (1033), 281–297.

[4] Banabic, D., 2010. Sheet Metal Forming Processes. Springer Berlin Heidelberg, 27-140.

[5] Barlat, F., Brem, J.C., Yoon, J.W., Chung, K., Dick, R.E., Lege, D.J., Pourboghrat, F., Choi, S.-H., Chu, E., 2003. Plane stress yield function for aluminum alloy sheets—part 1: theory. International Journal of Plasticity 19 (9), 1297–1319.

[6] Moldovan, I., Linnepe, M., Keßler, L., Köhl, M., 2019. Virtual modelling of forming processes in metal packaging industry. 12th European LS-DYNA Conference 2019, Koblenz, Germany.

The New Full Field Calibration Approach for Solid Elements to Model Constitutive Behaviour in Crashworthiness

C. Ilg (DYNAmore)

An Experimental and Numerical Investigation on Vulcanized Fiber

Kardelen Bayram, Marcus Pfeiffer, Christian Alter, Stefan Kolling

Institute of Mechanics and Materials
Technische Hochschule Mittelhessen, Giessen, Germany

1 Abstract

As the automotive industry and its companies start to look for sustainability, materials made of natural fibers receive a growing interest. In the present work, the mechanical properties of a vulcanized fiber material were investigated to understand the orthotropic and rate-dependent material behavior and make it more predictable and thus practicable. ***MAT_PAPER** was used to represent the anisotropic elastic-plastic behavior of the material for crash and impact simulations.
To confirm numerical results from explicit FEM calculations, a user material subroutine comparable to ***MAT_PAPER** was utilized. Therefore, it was possible to perform implicit calculations of the conducted validation experiments which were performed at low strain rates.

Neural network representation of mechanical fasteners in large-scale analyses

Victor André[1], David Morin[1], Miguel Costas[1], Magnus Langseth[1]

[1] Structural Impact Laboratory (SIMLab) and Centre for Advanced Structural Analysis (CASA), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), NO-7491 Trondheim, Norway

1 Abstract

This paper presents an artificial neural network (NN) modeling approach for representing mechanical fasteners in large-scale finite element crash simulations for explicit analysis using LS-DYNA version R9.3.1. The NN-model is established to describe the local force-deformation response of point-connectors in automotive applications like self-piercing-rivets and flow-drill-screws. The behaviour from initial loading until failure or unloading is covered. Various architectures and complexities of feedforward NNs were evaluated and trained based on synthetic experiments generated from the constraint model proposed by Hanssen et al. [1]. The constraint model is available as ****CONSTRAINED_SPR2*** *but was used in form of a cohesive element (8-noded, 4-point cohesive element with offsets for use with shells).*

The NN consists of three hidden layers each having 100 nodes. Weights and bias where trained using the Python/Keras package on the basis of synthetic stretch paths calculated by the constraint model. The NN is represented by a sequence of matrix multiplication inside a ****MAT_USER_DEFINED_MATERIAL_MODELS*** *where the joints force-components and damage are computed each time step.*

This forms a proof of concept for implementing such a machine learning modeling technique not based on physics-motivated equations. Both the impact of network complexity and training data diversity was investigated. The numerical results are compared to physical tests and the ****CONSTRAINED_SPR2****-model for five different joint configurations including self-piercing rivets and flow-drill screws. Experimental data was obtained by Sønstabø et. al [2, 3, 4, 5]. All joint configurations show variation in their loading and failure behaviour which gives a wide selection for validating a NN-design that fits all applications. It can be shown that a rather basic machine learning technique like a feedforward NN is able to reproduce path-dependent force-deformation relation for the application in the explicit* LS-DYNA *solver.*

2 Literature

[1] A. G. Hanssen, L. Olovsson, R. Porcaro, and M. Langseth. A large-scale finite element pointconnector model for self-piercing rivet connections. European Journal of Mechanics, A/Solids, 29(4):484–495, 2010. ISSN 09977538. doi: 10.1016/j.euromechsol.2010.02.010. URL http://dx.doi.org/10.1016/j.euromechsol.2010.02.010.

[2] Johan Kolsto Sonstabo, Petter Henrik Holmstrom, David Morin, and Magnus Langseth. Macroscopic strength and failure properties of flow-drill screw connections. Journal of Materials Processing Technology, 222:1–12, 2015. ISSN 09240136. doi: 10.1016/j.jmatprotec.2015.02.031. URL http://dx.doi.org/10.1016/j.jmatprotec.2015.02.031.

[3] Johan Kolstø Sønstabø, David Morin, and Magnus Langseth. Macroscopic modelling of flow-drill screw connections in thin-walled aluminium structures. Thin Walled Structures, 105:185–206, 2016. ISSN 0263-8231. doi: 10.1016/j.tws.2016.04.013. URL http://dx.doi.org/10.1016/j.tws.2016.04.013.

[4] Johan Kolstø Sønstabø, David Morin, and Magnus Langseth. Testing and modelling of flowdrill screw connections under quasi-static loadings. Journal of Materials Processing Technology, 255:724–738, 5 2018. ISSN 09240136. doi: 10.1016/j.jmatprotec.2018.01.007. URL https://linkinghub.elsevier.com/retrieve/pii/S0924013618300086.

[5] Johan Kolstø Sønstabø, David Morin, and Magnus Langseth. Static and dynamic testing and modelling of aluminium joints with flow-drill screw connections. International Journal of Impact Engineering, 115(January):58–75, 2018. ISSN 0734743X. doi: 10.1016/j.ijimpeng.2018.01.008. URL https://doi.org/10.1016/j.ijimpeng.2018.01.008.

DM.inspect: customizable quality control of LS-DYNA input files

Steffen Mattern[1], Robert D. Bitsche[2], Marcel Koch[3]

[1]DYNAmore GmbH
[2]Scale GmbH
[3]Dr. Ing. h. c. F. Porsche AG

Abstract

LS-DYNA models for industrial applications are often composed from several smaller sub-models. For example, in the automotive industry the different components of a car are usually modelled separately. These “include files” may be built-up in different departments of the same company or even developed by external suppliers. Due to the huge variety of features and functionalities in LS-DYNA, it is a good idea to set up general rules for the sub-models to achieve robust and efficient simulation models for production. Also, the successful assembly of models can be assured by defining generalized modeling guidelines, especially for the interaction of the different include files.

In order to ensure compliance with their modeling guidelines already during the development process of the sub-models, Porsche tasked DYNAmore with the development of a batch program that performs quality checks on LS-DYNA input files. This program is now an inherent part of Porsche’s CAE process chain and is also used by some of their external suppliers. Based on the experiences from this collaboration, DYNAmore has now developed a new software called DM.inspect, a batch program allowing customized quality control for LS-DYNA input files. The software requires the individual definition of quality criteria by the user rather than containing any pre-defined checks. This modularized approach allows the use of DM.inspect for various simulation disciplines with different requirements.

The check procedure performed by DM.inspect can be divided into three subsequent phases. An optional pre-phase where checks from a pre-processor can be applied, a simulation-phase performing an LS-DYNA initialization run and a post-phase with various checks based on the LS-DYNA input and output files. In the end, DM.inspect writes a report containing the results of the performed quality checks. This paper introduces DM.inspect and describes installation, configuration, and application of the software. Based on a small example, different checks are presented and the report generated by DM.inspect is discussed. Finally, the integration of DM.inspect into the simulation data management system at Porsche is briefly described.

Forming Simulation of Tailored Press Hardened Parts

Marcel Triebus[1], Alexander Reitz[2], Olexandr Grydin[2], Julian Grenz[3], Andreas Schneidt[3], Rüdiger Erhardt[3], Thomas Tröster[1], Mirko Schaper[2]

[1] Paderborn University, Automotive Lightweight Design, Warburger Str. 100, 33098 Paderborn, Germany
[2] Paderborn University, Material Science, Warburger Str. 100, 33098 Paderborn, Germany
[3] BENTELER Automobiltechnik GmbH, An der Talle 27-31, 33102 Paderborn, Germany

1 Abstract

Hot forming of metal parts is characterized by forming over recrystallisation temperature [1]. For steel, press hardening is a popular production technology for creating hardened parts under hot forming conditions. In the conventional press hardening process, the blank is heated above austenitizing temperature and then transferred to the forming tool. The tools are water cooled and therefore ensure a martensitic transformation of the steel material. The most popular alloy is the boron steel 22MnB5, where a tensile strength of around 1500 MPa is reached through press hardening processes. The latest body-in-white concepts show a broad range of press hardened parts. The underlying forming methods are aiming to create purpose build components through variations of the press hardening process like tailored property processes, the use of tailor-welded or tailor-rolled blanks [2]. In the tailored property process, tailoring of the material properties is realized through the decrease of the cooling rate in a designated area of the part e.g., with a heated tool region. Due to the lower cooling rate, a softer and more ductile state is created in this area with microstructures of ferrite, perlite and bainite. As a result, from the multiphase microstructure of tailored property parts, shape distortion is more pronounced then in conventional press hardening parts with a fully martensitic microstructure. Increased shape distortion can lead to additional rework cycles in the tool manufacturing.
This research proposes a method for simulation-based prediction of shape distortion to reduce rework. For the thermo-mechanical coupled simulation of the tailored property process the phase transformations are considered with the material model **`*MAT_248`** [3]. The parameter identification for **`*MAT_248`** is done with 1-element simulations, time-temperature experiments, and metallurgical evaluation of the resulting microstructure. The simulations show a good agreement with the experimental results for tailored property parts regarding the microstructure. Moreover, the predicted geometry of the parts after the forming process is validated with optical measurements.

Literature

[1] Neugebauer R, Altan T, Geiger M, Kleiner M and Sterzing A: “Sheet metal forming at elevated temperatures” CIRP Annals 55, 2006, 793–816
[2] Mori K, Bariani P F, Behrens B-A, Brosius A, Bruschi S, Maeno T, Merklein M and Yanagimoto J: “Hot stamping of ultra-high strength steel parts“, CIRP Annals 66, 2017, 755–77
[3] Hippchen P: “Simulative Prognose der Geometrie indirekt pressgehärteter Karosseriebauteile für die industrielle Anwendung”, Erlangen-Nürnberg, Univ., Diss., (Bamberg: Meisenbach), 2014

Synthetic data and Artificial Intelligence: forming process control using real-time data acquisition and AI trained with simulation datasets

Michael Schmiedt[1], Jürgen Lenz[2], Wolfgang Rimkus[1], Sebastian Feldmann[1]

[1]Aalen University of Applied Science
[2]INNEO Solutions GmbH

So far, training an Artificial Intelligence (AI) required a high testing effort or access to large datasets, making it unsuitable for small and medium-sized companies. The innovation of the research project "SimKI" is based on training an AI by means of several datasets: a) simulation data from a parameter study, b) live data from forming experiments, c) a combination of both. Therefore, process parameters and the respective quality information of deep drawn components are recorded both inline as well as in real time and transferred to the cloud-based IoT platform "ThingWorx". The quality information is determined by an optical measurement system (GOM Aramis) and laser triangulation (Gocator line scanner). In order to support small and medium-sized companies with a toolchain applicable to their existing machinery, an older forming press is retrofitted with digital interfaces. The simulation model created contains a thermal material model of the formed blank (LS-DYNA MAT36) and the parameters of the real forming process such as the press speed.

The collected data is correlated and evaluated using an Artificial Intelligence algorithm. In contrast to conventional training with scanned components and the corresponding genuine datasets (82.96 % accuracy), the testing results show a comparable validation accuracy of the DNN-based quality assessment by using simulation data alone (83.84 %) or a hybrid dataset (84.38 %). This allows to significantly minimize both the implementation effort as well as the testing effort in production and thus to increase the industrial applicability.

A gray-scale mapping method to consider locally varying properties for wood forming simulations

Christian Liebold[1], David Zerbst[2], Thomas Gereke[3], Sebastian Clauß[4]

[1]DYNAmore GmbH
[2]German Aerospace Center
[3]ITM, TU Dresden
[4]Mercedes Benz AG

Keywords: *finite element mapping, veneer laminates, interior design*

1 Abstract

Automotive interior components in upper-class vehicles are often made of wood veneer sheets that are subject to a forming process [1]. Due to the anisotropy and inhomogeneity of the material caused by the development of annual rings during the growth of the tree, establishing a stable production process based on trial-and-error forming tests is time-consuming and costly [2]. Hence, numerical methods for simulating the forming process are in high demand to support the development of feasible trim part geometries. The key for reliable process simulations of wood-based materials is the consideration of the variability of material properties.

In this paper, the authors present a method to account for the locally varying properties of early and late wood in finite element simulations using a gray-scale mapping procedure. The method was introduced in [3] and was implemented in the software tool Envyo®[5]. The developed approach, which consists of a gravity loading step and a forming simulation with **`*MAT_LAMINATED_COMPOSITE_FABRIC`** (**`*MAT_58`**) in LS-DYNA [4], is validated using the example of a palm rest of the Mercedes-Benz X167 series. Furthermore, the necessary steps and assumptions for material data calibration will be demonstrated.

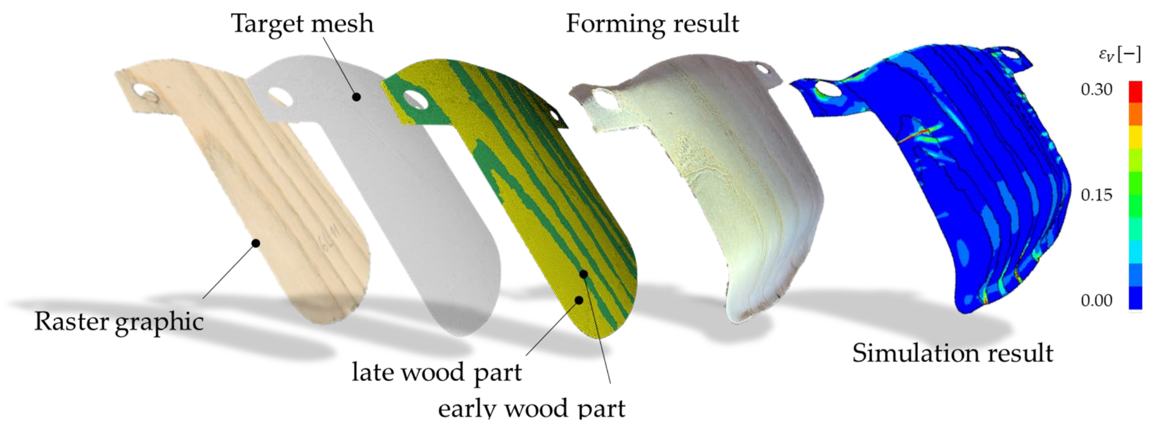


Fig.1: Illustration of the established process chain.

2 Literature

[1] Zerbst, D., Liebold, C., Gereke, T., Clauß, S., Cherif, C.: “Modelling inhomogeneities of veneers with a grayscale mapping approach”, ECCOMAS CompWood, 2019.
[2] Zerbst, D., Liebold, C., Gereke, T., Clauß, S., Cherif, C.: "Numerical Simulation of the Forming Process of Veneer Laminates", J. Compos. Sci., 2021, 5, 150. https://doi.org/10.3390/jcs5060150
[3] Zerbst, D., Liebold, C., Gereke, T., Haufe, A., Clauß, S., Cherif, C.: “Modelling Inhomogeneity of Veneer Laminates with a Finite Element Mapping Method Based on Arbitrary Grayscale Images”, Materials, 2020, 13, 2993. https://doi.org/10.3390/ma13132993
[4] Livermore Software Technology (LST), an ANSYS Company: “LS-DYNA User’s Manual, Vol. II – Material Models”, 2020.
[5] DYNAmore GmbH, “Envyo® User’s Manual”, GER, 2021.

A three-dimensional finite element model for the roll bending of heavy plates using a 4-roll plate bending machine

Lukas V. Kappis[1], Emanuel von Cramon-Taubadel[1], Pascal Froitzheim[1], Wilko Flügge

[1]Fraunhofer Institute for Large Structures in Production Engineering IGP

Roll bending is a manufacturing process for the continuous round bending of sheet metal. It is used in particular for the production of thick-walled pipes and shells with a high volume, which are used in the maritime sector, in pipeline construction and in the field of renewable energies. In its current application in the thick plate sector, the process is controlled purely manually, which is why there is considerable potential for optimization through automation. A detailed understanding of the process is required to develop automation approaches, and further research needs to be done to achieve this. Since the experimental investigation of the process is associated with a high material and cost effort, in this work a modular three-dimensional finite element model was developed in LS DYNA, which can be used for a computationally efficient numerical process analysis.

As the forming tool, the geometry of a real round bending machine with four rolls was optically measured and remodeled. To achieve a high level of computational efficiency, the forming process was divided into several substeps. A separate model was developed for each substep, whereby for example the time integration method used and the modeling strategy of the rolls were specifically adapted to the process steps. Between the process steps, the stress and deformation state of the sheet metal is transferred using a Dynain-file. A Python script was developed for the transfer of consistent boundary conditions, which can be used, for example, to position the rolls on the basis of a predefined test plan. The model can be used to simulate flexible forming strategies in which the substeps can be arranged in an arbitrary sequence.

In this study, the developed model was used for a sensitivity analysis to investigate the influence of various parameters on the forming behavior of a sheet during a single rolling pass. On the workpiece side, the sheet dimensions, the material strength and the extent of residual stresses were investigated. On the machine side, the infeed of the forming tool and the stiffness of the mounting of the bottom roll were investigated. As a result of the work, significant and negligible influencing variables on the forming behavior of heavy plates during roll bending were identified.

PCA-based sensitivity analysis of response fields using LS-OPT®

Charlotte Keisser[1], Max Hübner[2], Tobias Graf[2], Anirban Basudhar[3], Nielen Stander[3]

[1]DYNAmore FRANCE SAS, Versailles, France
[2]DYNAmore GmbH, Stuttgart, Germany
[3]LST LLC, Ansys, Livermore, California, USA

ABSTRACT

When performing an optimization, it is important to avoid introducing unnecessary variables that do not impact the design objectives and constraints. Such variables increase the design space size and lead to unnecessary sample evaluations, which can significantly increase the overall computation time or cost. A sensitivity analysis can be performed to quantify the significance of the variables; only the important variables are then used in the sampling and optimization, thus reducing the computational cost.
Linear ANOVA (Analysis of Variance) and Global Sensitivity Analysis (GSA/Sobol) sensitivity measures, which have been available in LS-OPT® for several years, allow the users to analyze the sensitivity of scalar responses to parameters. However, sensitivity analysis of vector entities such as time histories was not available earlier.
Since LS-OPT 7.0, a new sensitivity measure based on PCA (Principal Component Analysis) is available to analyze the importance of parameters for non-scalar entities. This sensitivity measure is available for both time histories and for multi-point responses (spatial data), which were also added in LS-OPT 7.0. This measure will be extended to multi-histories (time and spatial data) in the future.
PCA is a method similar to Proper Orthogonal Decomposition (POD), which is commonly used in exploratory data analysis or for model prediction. It performs a dimensionality reduction of possibly correlated data sets by projecting each data point onto only a few principal components (PCs) forming a linearly uncorrelated orthonormal basis while preserving the data's variation. The PCs are sorted in decreasing order according to their variance, so that the first PC accounts for the largest variability in the data set. The new sensitivity measure provides the contribution of the parameters to each PC as well as the contribution of each PC to the results. A cumulative history/multi-point responses sensitivity measure is also available, allowing the analysis of the overall model.
A component test of a B-pillar will be used to highlight this new feature. In crash scenarios the nature of impact as well as the critical spatial points to consider on the structure may not be known a priori. Therefore, it may be useful to consider multi-point responses/histories, as will be demonstrated through this work.

Multi-material Design via Discrete Material Sampling and Topology Optimization

Mariusz Bujny[1], Satchit Ramnath[2], Nathan Zurbrugg[3], Stefan Menzel[1], Duane Detwiler[3]

[1]Honda Research Institute Europe GmbH, Offenbach/Main, Germany
[2]SIMCenter, The Ohio State University, Columbus, Ohio, USA
[3]Honda Development and Manufacturing of America, LLC, Raymond, Ohio, USA

Abstract

In order to meet stringent requirements concerning vehicle safety, costs, and weight, vehicle structures composed of multiple material types have become a standard in the automotive industry. At the same time, the complexity of the design process has grown to a stage where relying solely on intuition is very difficult and more sophisticated methods are needed to develop conceptual designs at early development stages. Topology Optimization (TO) [1] is one such tool that allows for a fully automated derivation of design concepts for a variety of loading conditions.

In this work, a novel method for multi-material TO based on LS-OPT® and LS-TaSCTM is proposed. The optimization is carried out in two nested loops. In the outer loop, LS-OPT® assigns materials from a given set to predefined parts dividing the design space into multiple sub-domains. The partitioning of the model is defined based on the preferences of the designer, reflecting potentially the manufacturing limitations. In the inner loop, LS-TaSCTM is used to perform TO based on the multi-material input model. The mass fractions for each of the parts are determined based on guidelines assuming maximization of structural performance under a specific cost constraint. As a result, a wide spectrum of multi-material topological concepts, meeting mass and cost constraints, can be explored and optimized for arbitrary criteria by treating material types as design variables for a non-gradient discrete optimization problem [2] in the outer loop.

Unlike the state-of-the-art methods for multi-material TO [3, 4], the proposed approach is very general and allows for the incorporation of considerably different types of materials. This is possible because the material properties are not just adjusted according to a specific parametrization, but sampled in a discrete fashion from a fixed set of material models. This, however, comes at the price of higher computational costs and requires the definition of a limited number of parts for assigning the materials.

To evaluate the method, a multi-disciplinary TO of a solid beam subject to static and crash load cases is considered. The design space is split into multiple parts, which are assigned material properties corresponding either to aluminum or steel. The proposed method produces structures superior to the single material designs. Finally, the generality of the proposed approach offers the potential for an easy integration into the development process in industry.

Literature

[1] M.P. Bendsøe and O. Sigmund, Topology Optimization, Springer, Berlin Heidelberg, 2004.
[2] W. Hare, J. Nutini, and S. Tesfamariam, A survey of non-gradient optimization methods in structural engineering, Adv. Eng. Softw. 59 (2013), pp. 19-28.
[3] W. Zuo, and K. Saitou, Multi-material topology optimization using ordered SIMP interpolation, Struct. Multidisc. Optim. 55 (2017), pp. 477-491.
[4] R. Tavakoli, S. Mohseni, Alternating active-phase algorithm for multimaterial topology optimization problems: a 115-line MATLAB implementation, Struct. Multidisc. Optim. 49 (2013), pp. 621 – 642.

Creation of 3D geometry from topology optimization results, for thin-walled and casted parts

Alexandros Kaloudis[1], Apostolos Poulias[2]

[1]BETA CAE International AG
[2]BETA CAE Systems SA

Abstract

The results of a topology optimization cannot be directly used as CAD geometry for further analysis, verification or optimization, or even be manufactured. In order to address this problem, the "organic mesh" model, resulting from topology optimization, is processed with two different techniques, depending on the manufacturing process that will be used for the part/s it will represent, resulting in a usable 3D geometry. These two techniques are : i) “the organic mesh” model will be used as a basis for producing the 3D geometry of new thin-walled parts utilizing cross sections. Cross sections which can be derived from already existing structures or generated from scratch or exist in a library, and which can be parameterized. ii) The "organic mesh" model, at the beginning can be smoothed, and then, using a sampling technique, new geometrical faces will be generated upon this later smoothed mesh, creating a solid description (casted like) of a component.

This presentation describes the way these two approaches are integrated by means of tools and methodologies into ACP OpDesign and ANSA.

Optimization of Test Specimen Geometries to Construct a GISSMO Failure Curve for a High Strength Steel

B. Gürsoy, E. Tamer (BIAS Engineering

The demand for reducing CO2 emission is a trendy topic in automotive industry as well as all other industries. The use of high strength steels instead of conventional lightweight steels has a major role with predictive modelling methods in finite element analysis. A full-scale crash test is both time consuming and expensive. On the other hand, a high-fidelity crashworthiness analysis in finite element environment requires accurate damage and material model parameters description. In this study, the damage model GISSMO-Generalized Incremental Stress State dependent damage model has been studied. At a certain region of the material failure curve is modeled using different geometric specimens. The proposed simulations were executed in LS-DYNA and LS-OPT environments. A full material failure curve calibration with different specimen sets were used and compared with experimental data. Furthermore, the process is optimized in terms of used specimen sets and runtime. Thus, the results of the numerical simulations are verified through the comparison of the results with the measurements of experimental results of those optimized test specimens.

New developments of SPG method for large deformation and material failure analysis of multistagemanufacturing processes

Youcai Wu, Xiaofei Pan, Wei Hu, C.T. Wu

Livermore Software Technology, an ANSYS company
7374 Las Positas Road, Livermore, CA 94551, U.S.A.

Abstract

The SPG (Smoothed Particle Galerkin) method [1, 2], targeted for modeling ductile material failure, has been applied in various industrial problems such as aerospace, automobile, civil, defense, and manufacturing processes that are very challenge to traditional finite element method (FEM) due to large deformation and material failure, which typically occurs in metal cutting, friction drilling, and grinding processes.

In the recent years, the SPG method was enhanced with new algorithms that not only improve the efficiency and stability of the method but also expand its application area, which includes analysis of multistage manufacturing processes (MMPs). First, a momentum consistent SPG (MCSPG) formulation was developed [3, 4] for coupled thermo-mechanical analyses of manufacturing problems featured with large deformation and material failure. Second, a two-stage SPG/MCSPG algorithm was developed for mechanical joining process and joint performance analysis (typically in automobile industry) where significant boundary condition change (e.g., coach peel, lap shear) could occur between different stages in the MMPs analysis. On the other hand, a small restart can be employed to perform the 2nd stage of the MMPs analysis if boundary condition change between the two stages is not significant, e.g. screw pullout after screwing.

The new features and the corresponding LS-DYNA keywords will be introduced in this paper. Afterwards, a coupled thermal mechanical analysis of orthogonal cutting process of titanic alloy Ti6Al4V will be presented to demonstrate the effectiveness of the MCSPG method. Followed by that, MCSPG two- stage analyses of self-piercing riveting (SPR) joining process and joint performance will be presented to show the capability of the two-stage algorithm. In this analysis, the joining process is analyzed only once in the 1st stage. Virtual mechanical tests such as lap shear, coach peel and cross tension of the SPR joint are analyzed in the 2nd stage performance analysis, which utilized the exact same histories (e.g., effective plastic strain, stress, deformed shape) from the 1st stage as the initial conditions. As such, consistent strengths (i.e., coach peel, cross tension and lap shear) of a single joint can be obtained through the two- stage analysis algorithm.

Reference:

1 Wu CT, Koishi M, Hu W. A displacement smoothing induced strain gradient stabilization for the meshfree Galerkin nodal integration method. Comput. Mech. 2015: **56**, 19-37.

2 Wu CT, Chi SW, Koishi M, Wu Y. Strain gradient stabilization with dual stress points for the meshfree nodal integration method in inelastic analysis. Int. J. Numer. Methods Engrg. 2016:**107**, 3-30.

3 Pan X, Wu CT, Hu W, Wu Y. A momentum-consistent stabilization algorithm for Lagrangian particle methods in the thermo-mechanical friction drilling analysis. Comput. Mech. 2019:**64**, 625-644.

4 Wu CT, Wu Y, Lyu D, Pan X, Hu W. The momentum-consistent smoothed particle Galerkin (MC-SPG) method for simulating the extreme thread forming in the flow drill screw driving process. Comp. Part. Mech. 2020:**7**,177–191. https://doi.org/10.1007/s40571-019-00235-2

Spectral Element Methods for Transient Acoustics in ANSYS LS-DYNA®

Tom Littlewood[1], Yun Huang[1], Zhe Cui[1]

[1]Livermore Software Technology, an ANSYS Company

Abstract

Always considered a leading software package for nonlinear transient analysis in the automotive industry, LS-DYNA has in recent years increasingly added capabilities for acoustic analysis. This paper introduces recent work on the implementation of spectral element methods for transient acoustics. It identifies why spectral elements are compelling for transient acoustics when contrasted with iso-parametric finite elements, especially for ultrasonic acoustics, and then reviews the features currently implemented. Finally, the paper presents a few illustrative examples using the acoustic spectral elements in LS-DYNA on multicore platforms.

Recent developments in NVH and fatigue solvers in Ansys LS-DYNA®

Yun Huang[1], Tom Littlewood[1], Zhe Cui[1], Ushnish Basu[1], Stefan Hartmann[2], David Benson[1]

[1]Ansys, Inc., US
[2]DYNAmore GmbH, Germany

Abstract

As one of the mainstream software packages widely used in automotive industry, LS-DYNA provides not only strong nonlinear capabilities for crashworthiness simulation, but also a suite of solvers for NVH and fatigue (durability) analysis. For NVH analysis, a series of vibration and acoustic solvers have been implemented to meet the need from CAE analysis of automotive of different levels and phases. They include FRF (Frequency Response Function), SSD (Steady State Dynamics), random vibration, response spectrum analysis, and acoustic analysis based on BEM (Boundary Element Method), FEM (Finite Element Method) and SEM (Spectral Element Method). The fatigue analysis features include fatigue damage solvers in both time domain and frequency domain (based on random vibration and steady state vibration).

Since the last European Users' Conference in Koblenz, Germany, 2019, many new solvers / options were implemented to the NVH and fatigue (durability) analysis modules, including

- Acoustic spectral element method
- Mode-based FEM acoustics
- New boundary / load conditions for transient acoustic analysis
- SSD analysis with multiple load cases
- SSD with coupled fluid-structure system
- Fatigue analysis based on modal dynamics
- Random vibration fatigue analysis based on IGA models
- D3MAX for stress and strain envelope

This presentation gives a brief review of these new options for NVH and fatigue analysis with LS-DYNA, and discusses the area of applications of them. Some examples are included for illustration purpose. The plan for the future development in this area is also discussed.

Latest in AI/ML application to modeling complex geometry

Prakash “Krish” Krishnaswamy[1], Umesh Mallikarjunaiah[2], Yoshikazu Nakagawa[3]

[1, 2] Xitadel CAE Technologies Pvt. Ltd., India

[3]Honda Motor Co., Ltd, Japan

Abstract

There is rapid convergence of multiple technologies that are creating unprecedented capabilities in every field of technology. The incorporation of new technologies like Artificial Intelligence/Machine Learning (AI/ML) in the CAE process has been quite gradual. Xitadel’s XIPA technology is a pioneering effort to leverage the power of ML to transform the CAE model build process and bring this to production level. CAE modeling is a critical path in the overall CAE process. CAE modeling however is very time consuming, particularly because plastic subsystems typically contain multiple complex features and variable thicknesses.

XIPA uses Deep Learning multi-layered Neural Network that automates the recognition of geometry features in the CAD data and automatically applies the appropriate mesh. XIPA can be customized (“ML trained”) to expand its recognition capabilities.

The application of XIPA technology to the modeling and assembly of complex plastics components and subsystems like door inner panel, instrument panel, bumper/fascia, etc. achieves modeling productivity gains of 80% or more. Moreover, specifications and organizational best practices for modeling are also captured, facilitating the institutionalization of process and practice in the organization.

Keywords: Machine Learning, Deep Neural Networks, Convolutional Neural Networks, Faster R-CNN, VGG16 Neural Network, XIPA, Machine Learning for Plastics

Modelling Laminated Glass in LS-Dyna under extreme loading conditions

Daniel Aggromito[1], Jon Farley[1], Jessica Klimenko[1], Marc Tartasky[1], Will Wholey[1], Luke Pascoe[1],

[1]Arup

Abstract

In the event of an explosion in a populated urban area, fragmentation from glass is a significant contributor to human injury. The mitigation of glass fragmentation hazards is well-established through the use of laminated glass featuring a polymer interlayer, such as DuPont Sentry Glass Plus (SGP) or polyvinyl butyral (PVB). These interlayers work by exploiting the inherent viscoelastic and adhesive properties of the polymer, providing a mechanism to dissipate the energy of the blast through work done in deformation of the interlayer while retaining fragments of broken glass. This behavior is fundamental to limit the projection of fragments of otherwise brittle glass, thereby reducing or eliminating highly hazardous secondary fragmentation associated with glazing. As a result, laminated glass has been designed to withstand substantial blast loadings, absorbing and dissipating the energy from the blast wave through cracking and deformation. This pane deflection can result in substantial load transfer to the window framing system, resulting in a dynamic in-plane and out-of-plane loading scenario that requires adequate restraint in order to avoid total pane failure. Whilst the benefits of laminated glass have been utilized for decades in blast hazard mitigation, relatively few analytical models exist that are capable of capturing the complex mechanics of the pane and frame loading under the high strain-rate conditions observed in blast events. This research paper presents recent activities by Arup to develop a methodology that has been demonstrated to be effective in replicating both lab (specimen) and full-scale arena test results of single pane laminated glass response to blast scenarios in LS-DYNA. This paper outlines a number of development activities undertaken in the course of development examining dependencies on erosion criteria, mesh sensitivity, adhesion variance, and strain-rate effects. Through the investigation of sealed double-glazed units subjected to blast loads, the authors propose a methodology to model PVB laminated glazing including crack growth representation and material characterization studies of the silicone edge restraints. Combined together, this methodology is shown to generate accurate edge reactions, thereby enabling designers to efficiently design façade framing elements and glazing restraint.

J-Composites/Compression Molding Version 2.0: New Simulation Tool for CFRP Composites

Shinya Hayashi[1], Shaun Dougherty[1], Shinya Hiroi[1], Sean Wang[1], Atsushi Yoshida[1]

[1]JSOL Corporation, Japan

1 Abstract

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, discontinuous long carbon fiber reinforced plastics are increasingly 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 discontinuous long fiber reinforced plastics into complex shapes with relatively low manufacturing cost and short process time. LST and JSOL developed new compression molding simulation techniques for discontinuous long fiber reinforced plastics using a beam-in-adaptive EFG coupling function in LS-DYNA®. Then JSOL developed a modelling tool called J-Composites®/Compression Molding to generate an input deck for this new compression molding simulation. In this paper, new functions of J-Composites/Compression Molding Version 2.0 are introduced and two compression molding simulations using hybrid lay-up composites are presented.

In March 2021 JSOL released the Ver.2.0 of J-Composites/Compression Molding. This new version has the following new functions:

1. Generates new fiber orientation types
   - Unidirectional Fiber type (UD)
   - Chopped Unidirectional Fiber type (CUD)
   - Hybrid lay-up composites consisting of ROS, ROF, UD and CUD types
2. Supports more complex matrix resin geometries of arbitrary shape
   - Arbitrary shape with contact thickness
   - Convex shape
3. Adds new commercial CFRP material data into Material Database
   - Flexcarbon (ROS type) from Suncorona Oda Co., Ltd.
4. Upgrades contact definitions for more stable calculation
5. Enhances set-up system for analysis and forming conditions
   - Set time when the deformation mode switches from the press forming phase to the compression molding phase
   - User-defined curves for time step and adaptive remeshing interval
   - Adjustable function for deletion of uncoupled beam elements
   - Optional function to apply mass scaling to all elements for faster calculation time (sometimes for more stable simulation)
   - Preview tool movement animation (available in Forming Conditions panel)

Multiscale Simulation of Short-Fiber-Reinforced Composites: From Computational Homogenization to Mechanistic Machine Learning in LS-DYNA

Haoyan Wei[1], C. T. Wu[1], Dandan Lyu[1], Wei Hu[1], Francois-Henry Rouet[1], Kevin Zhang[1], Philip Ho[1], Hitoshi Oura[2], Masato Nishi[2], Tadashi Naito[3], Leo Shen[4]

[1]Livermore Software Technology LLC, ANSYS, USA
[2]JSOL Corporation, Japan
[3]Honda Motor Co., Ltd., Japan
[4]CoreTech System Co., Ltd., Taiwan

Abstract

Injection-molded short-fiber-reinforced composites (SFRC) have been widely used for structural applications in automotive and electronics industries. Due to the heterogeneous microstructures across different length scales, the nonlinear anisotropic behaviors of SFRC are very challenging to model. Therefore, an effective multiscale approach that links the local microscopic properties (e.g., fiber orientation, fiber volume fraction) to the global behaviors is required. To this end, multiscale analysis functions are recently developed in the engineering simulation software LS-DYNA to enable high-fidelity micromechanical finite element analysis, mechanistic machine learning-based reduced-order modeling, and accelerated concurrent multiscale simulation of SFRC composite structures.

In this paper, we will firstly introduce the LS-DYNA RVE (Representative Volume Element) analysis module [1], which employs the FEM-based computational homogenization technique to predict high-fidelity microscopic stress/strain fields within the composite microstructures and averaged macroscopic constitutive responses. The RVE technique is available in LS-DYNA R13 for virtual testing of numerically re-constructed material samples, which is of great importance to design and analysis of advanced materials.

In addition, we will introduce a mechanistic machine learning technique referred to as Deep Material Network (DMN) [2,3,4]. Based on DMN, a highly accurate and efficient concurrent multiscale simulation framework is being developed in LS-DYNA to simulate injection-molded SFRC structures, where microstructural distributions predicted by Moldex3D are imported into LS-DYNA through the LS-PrePost mapping function.

Literature

[1] Wei, H., Lyu, D., Hu, W., Wu, C. T. RVE analysis in LS-DYNA for high-fidelity multiscale material modeling. FEA Information Engineering Solutions. 2021, 7, 37-47. http://www.feapublications.com/2021/07_FEAIES_July_2021.pdf

[2] Liu, Z., Wu, C. T. Exploring the 3D architectures of deep material network in data-driven multiscale mechanics. Journal of the Mechanics and Physics of Solids, 2019, 127, 20-46.

[3] Liu, Z., Wu, C. T., Koishi, M. Transfer learning of deep material network for seamless structure–property predictions. Computational Mechanics, 2019, 64(2), 451-465.

[4] Liu, Z., Wei, H., Huang, T., Wu, C. T. Intelligent multiscale simulation based on process-guided composite database. 16th International LS-DYNA Users Conference, 2020. https://arxiv.org/abs/2003.09491

Creep study of expanded polystyrene used in Refrigerator packaging

Dayabhushan Thorat[1], Shrikant V Jagtap[1], Doray Chhetri[1], Sushilkumar Vishwakarma[1], Michael Fiori[2]

[1]Global Technology and Engineering Center, Whirlpool of India Ltd., Weikfield IT CITI Info ParkPune, India
[2]Whirlpool Benton Harbor Technical Center, Benton Harbor, MI, USA

1 Use of Templates

The efficient staging and storing of home appliances in warehouses is critical to avoid human injuries and huge loss to the company. After manufacturing the packaged product needs to be stored for a particular duration in the warehouse before it’s shipped out to suppliers. For optimum space management those products are stacked one over the other. So it is important to have robust packaging design to ensure even distribution of stacked product loads maintain their stability.
Considering the constant loading over a longer period there is a high possibility of creep effects in packaging material, and it becomes even more crucial when packaging material is Expanded Polystyrene (EPS). So it's important to study material creep behavior in designing product packaging.
In this study creep behaviour of EPS material is evaluated with customised indigenious test setup, where precise measurement is done using laser based deflection gauges. The results obtained for long term constant compressive loading at different stress levels for multiple material densities, at ambient temperature 23°C.
FEA methodology has been developed using LS-DYNA implicit code, *MAT_UNIFIED_CREEP material card is used to map creep behavior of EPS. This approach has further been used to evaluate product stack lean behavior. A good agreement was observed between the experimental and numerical investigations.
This simulation capability will help to reduce the significant amount of time and resources used for physical tests. Also, This early prediction of stacked product behavior helps to reduce safety hazards and occurrences in the warehouse.

Keywords: Refrigerator, Expanded Polystyrene, Appliance, Creep, Stack, LS-DYNA Implicit

Gradient enhanced damage: modelling, implementation and applications

Anton Matzenmiller[1], Henning Schmidt[2], Marvin Nahrmann[3]

[1]Professor of Applied Mechanics, Institute of Mechanics, Department of Mechanical Engineering, University of Kassel
[2]Institute of Mechanics, Department of Mechanical Engineering, University of Kassel and Bertrandt AG
[3]Institute of Mechanics, Department of Mechanical Engineering, University of Kassel

Abstract

FE simulations with constitutive models for softening material, such as in the case of standard continuum damage mechanics based approaches, suffer from pathological mesh sensitivity as a result of strain localisation into a single element row. To overcome this major drawback, the local damage has to be enhanced towards nonlocal damage evolution. A suitable method for this purpose is the integral nonlocal formulation, available in LS-DYNA® by the keyword *MAT_NONLOCAL. However, its costly underlying search algorithm can result in a strong increase of the simulation time, leading to an impractical application for engineering problems. This paper presents the modelling and implementation of a gradient enhanced nonlocal damage model, based on a Taylor series expansion of the integral nonlocal formulation. The resulting Helmholtz type differential equation represents an analogous field problem to the heat equation, allowing a very efficient implementation into LS-DYNA® by exploiting its thermo-mechanical coupled solver. Thereby, the nodal temperature is replaced by the nonlocal damage variable. The implementation is discussed in detail and verified by means of a simple tensile bar simulation. The gradient enhanced damage model is applied to two existing continuum damage models, demonstrating the versatile range of application and the simplicity of integrating the gradient enhancement for user defined material models. Therefore, a stress state dependent damage approach for ductile steels is considered as well as a failure model for hyperelastic PUR adhesives. The simulation results are compared to the according test data for various specimens, showing a good agreement within the region of softening material behavior. Moreover, mesh insensitivity is demonstrated and a critical comparison with the *MAT_NONLOCAL model is carried out, revealing a strong reduction of the simulation time in favor of the gradient enhanced damage method instead of the *MAT_NONLOCAL option.

Parameter Identification of Coating Parameters to Improve Webbing Bending Response in Passive Safety Crash Simulations

Anurag Soni, Stefan Schilling, Martin Grikschat
Autoliv, North Germany

Srivathsa Jagalur, Naveen Chandra, Sagar Venkatesh, Naganna Puttegowda, Abhiroop Vishwanatha
Autoliv, India

Mikael Dahlgren
Autoliv, Sweden

Background

Inadequate modeling of bending response often results in rope-like effect in lap belt during passive safety crash simulations, causing loss of contact area and eventually incorrect pelvis coupling. Including appropriate bending stiffness could improve it. Several options are available but the use of 2D slipring in system models poses limitations. Recently added coating feature in *MAT_SEATBELT_2D in LS-DYNA seems promising. The coating can be visualized as an added layer of elasto-plastic material on the surface of membrane element which enables transfer of bending load. The coating is defined by Young's modulus (Ecoat), yield stress (Scoat) and thickness (Tcoat).

Objective

This paper aims to identify optimized coating parameters based on static bending of webbing in physical tests and simulations. Sensitivities were analyzed. Finally, the performance of the belt model with the optimized coating values was demonstrated in four different load cases: belt-to-foam compression (single layer and double layers of foam), webbing-only pull, and Belt-to-THOR pelvis compression in sub-system test.

Method

Results of webbing static bending tests were replicated in simulations. In the test, one side of a webbing sample was clamped into a fixture and movement on the other side was restricted by a fixed probe. The webbing was bent when the fixture was rotated. Fixture rotation and force at probe were measured in the tests. Coating parameters in matched simulations were varied. Force to fixture rotation from simulations was compared with test data.

Results and Discussion

For the simulated variations, bending in belt increased up to a certain rotation angle and further rotation did not add more bending. Consequently, in force-rotation curves force linearly rose to peak value but yielded to further rotation. While variations in Ecoat and Tcoat affected the slope of force-rotation curves, variations in Scoat affected yield force. Higher Ecoat reduced rotation to produce similar peak force whereas, increase in Tcoat increased peak force for the same rotation angle. Optimized values of these parameters were thereby obtained from simulations. These optimum values improved belt performance in simulations for four load cases in terms of folding behavior and force level.

Conclusion

Based on the belt response in simulations for the four application load cases, the improved webbing model is expected to improve belt-to-dummy interaction in crash simulations.

Creating a complete crash model with GNS. High accurate Barriers, handy preprocessing with dummy positioning and fast tailored result analysis

Leyre Benito Cia[1], Christoph Kaulich[1]

[1] GNS mbH

1 Abstract

The current transformations in the automotive industry are forcing all car manufacturers to check their processes constantly and repeatedly for economic efficiency. Regarding the areas of body development, vehicle safety and occupant as well as pedestrian protection, economic efficiency can only be achieved through a high degree of automation in virtual product development.
Such a degree of automation is only feasible, if the software used and the integrated models are well coordinated.
In this presentation it will be shown how a high precision barrier model and several dummy models can be combined and positioned for any vehicle model within the preprocessor Generator4 with ease. Both direct positioning tools and simulation driven ones can be fully automated in a simple way. Information on positioning and kinematic relationships can be taken directly from the input files. This simplifies the following processes and reduces the error rate due to less manual data input. Generator4 is then able to directly start the simulation with LS-Dyna or other solvers.
After the simulation, the generated result files can be evaluated with the postprocessor Animator4. Predefined session files are available for obtaining all the result values defined by the commonly used regulations. Curves, images, videos and slides in standard format can be generated directly in Animator4 for the management-suitable presentation.
This short-coordinated sequence can be completely encapsulated and integrated into larger data management systems or HPC environments. In this way it is possible that the interactive work of the simulation engineer and the automated generation of variants is carried out in the same software with identical work steps.

Damage modeling of aluminum casting components considering defect distribution for crashworthiness prediction

Florence Andrieux[1], Christian Frie[1], Dong-Zhi Sun[1]

[1] Fraunhofer IWM

Abstract

Aluminum die casting components are widely used in vehicle constructions because of their good compromise between weight reduction and improvement of mechanical properties. The complex geometries of these components with inhomogeneous defect distribution are a relevant issue, as material with higher defect content shows lower fracture strain. It makes the analysis of the damage behavior for crash simulation more challenging. An extensive experimental investigation is required to quantify the scatter as well as the development of a suitable material model to describe it. The casting alloy AlSi9Mn is investigated. First, a screening investigation based on tensile tests from specimens cut from different positions of a component together with metallographic and computer tomographie (CT) analyses is performed. CT scans are realized by project partners. It appears that the scatter of material properties is not only deterministic (property variation from position to position) but also stochastic (property variation from component to component for the same position) and these variations can be correlated with the inhomogeneous distribution of microdefects observed on fracture surfaces. A defect dependent material model is derived based on the relationship between defect fraction and failure strains and assuming the failure strain of the defect free material. To determine the stress state dependent matrix failure strain, specimens with different geometries extracted at the same position in a component are tested and the model is calibrated considering the upper bound of the experimental results. Digital image correlation (DIC) analyses are performed to determine local strains. The model requires a defect distribution as initial condition. After segmentation of the fracture surfaces and CT scans the defect distributions are mapped into FE mesh. Each element gets a local porosity. Moreover, global defect features are extracted from the CT scans. Considering defect size, the porosity can be split into three populations. The global porosity, the relative fraction and the defect size of each population are chosen as defect features. A stochastic model is developed. For a given global feature, synthetic defect distributions are realized and mapped into FE mesh. Simulation of tensile and bending tests are performed using defect distribution from the fracture surface, the CT scans and the stochastic model. It is shown that the developed method well captures the scatter of local material properties, however its accuracy remains subjected to the accuracy of the microdefect distribution prediction.

The new rear airbags in the Mercedes-Benz S-Class - a challenge for hard- and software

Lutz Quarg[1], Martin Reinitz[2]

[1]Mercedes Benz AG
[2]iSi Automotive Berlin GmbH

A highlight of the new Mercedes-Benz S-Class is the airbag for rear outer occupants for the case of a severe frontal impact. The rear airbag, internally named Rear-Structure-Airbag (RSB), deploys out of the backrest of the front seat, and may help to reduce the loads on head and neck of the rear occupant in addition to the belt.
This airbag is based on a new concept that was developed exclusively for the use in the back seat and its special needs. It consists of a tubular structure which can be inflated by compressed air. The space in-between, the low-pressure chamber, will be filled by the surrounding air. A patented vent foil is keeping the sucked in air inside the bag to create the restraint effect.

For the central new approach of filling the low-pressure chamber with ambient air, the functionality of the solver had to be extended. With the direct help of DYNAmore and LSTC, LS-DYNA Keywords could be modified so that a vacuum in a uniform-pressure airbag causes surrounding air to be sucked in. However, filling by suction is a self-limiting process due to the associated pressure equalization. To achieve the necessary higher degrees of filling, a gas mass flow must additionally pass from the outer shape giving tubular structure into the low-pressure chamber. For a correct representation of the deployment, the tubular structure was modelled with the corpus scalar method. The requirement was now to be able to model mass flow between the CPM and the UPM chamber. The final and functional model of a complex multi-chamber airbag in LS-DYNA subsequently offered the possibility to analyze, understand and coordinate the overlaying, influencing and hardly quantifiable processes. This was one of the keys to the digital development of the Rear-Structure-Airbag.

Latest Developments towards a sophisticated THOR-5F Dummy Model

Ismail Maatouki[1], Christian Kleessen[1], Daniel Riemensperger[1], Chirag Shah[2], Jerry Wang[2]

[1] Humanetics Europe GmbH, Heidelberg, Germany
[2] Humanetics Innovative Solutions, Farmington Hills, MI, USA

1 Abstract

According to risk analysis studies females are at higher risk for whiplash, lower extremity, chest, and spine AIS 2+ injuries compared with males in comparable crashes. This is due, in large part, to the relative fragility of females' necks particularly within younger populations. (Ref.: Am J Public Health. 2011 Dec; 101(12): 2368–2373. doi: 10.2105/AJPH.2011.300275)

In September 2015, NHTSA awarded a contract to Humanetics to design and build three THOR-5F ATDs incorporating improvements similar to those adopted in the THOR-50M. The THOR-5F design, while based on many of the THOR-50M design concepts, also includes incremental changes to the head, neck, shoulder, thorax, abdomen, pelvis and extremities. Humanetics incorporated these changes to improve the manufacturability and usability of the dummy and to comply with the latest anthropometry and biofidelity requirements. In Addition to NHTSA, several other agencies are also evaluating THOR-5F for potential use in their assessments.

The THOR-5F is the potential successor to the HIII 5th percentile small female dummy (HIII-5F) for frontal crash safety applications.

In parallel to the hardware developments, Humanetics has developed a detailed FE model for the THOR-5F dummy to represent the latest hardware. Through significant biofidelic validations, certification test setups and a broad matrix of sled tests, the latest model version 1.0 obtained significant improvements to its accuracy and robustness. These improvements will help users to get the right analysis results in their early numerical production phase.

In this paper the authors will present and explain the ongoing development work towards highly detailed and validated THOR-5F FE model.

Latest Development of the advanced Pedestrian Legform Impactor CAE Model

Christian Kleessen[1], Chirag Shah[2]

[1] Humanetics Europe GmbH, Heidelberg, Germany
[2] Humanetics Innovative Solutions, Farmington Hills, MI, USA

1 Abstract

Different consumer ratings are planning to introduce a new legform impactor for pedestrian safety. CNCAP will introduce the legform in 2022 and EuroNCAP in 2023. This new legform impactor, the advanced Pedestrian Legform Impactor (aPLI), will replace the current Flexible Pedestrian Legform Impactor (FlexPLI).

The aPLI compared to the FlexPLI has an additional upper body mass with a rotational joint representing the pelvis and hip joint. Mass distribution of the aPLI is more human like. The latest hardware status of the aPLI is build level B (SBL-B).

In this paper the authors will present and explain the current and ongoing development work towards a highly detailed and validated aPLI SBL-B CAE model. Different vehicle like loading conditions based on different car structures have been tested with the latest aPLI hardware and used to validate the model.

Overview of pedestrian analysis setup and post-processing using the Oasys LS-DYNA Environment with a focus on new features

Gavin Newlands, Ben Crone

Arup

1 Introduction

The Oasys LS-DYNA Environment provides a comprehensive solution across the whole LS-DYNA workflow.

One aspect of that workflow in automotive engineering is pedestrian protection. Pedestrian protection is important to the design and development of the front end of vehicles. The various protocols, impactors and methods relating to pedestrian protection mean that the CAE process can be complex and time consuming. The Oasys tools aid in this process. These tools are available for both head and leg impact analyses and have been used successfully on past and current vehicle design projects within Arup to accelerate the workflow. This paper gives an overview of the tools, with a focus on the latest features introduced to the recent releases of the Oasys LS-DYNA Environment – The **HIC Area Calculator** and the **Pedestrian Run Builder**.

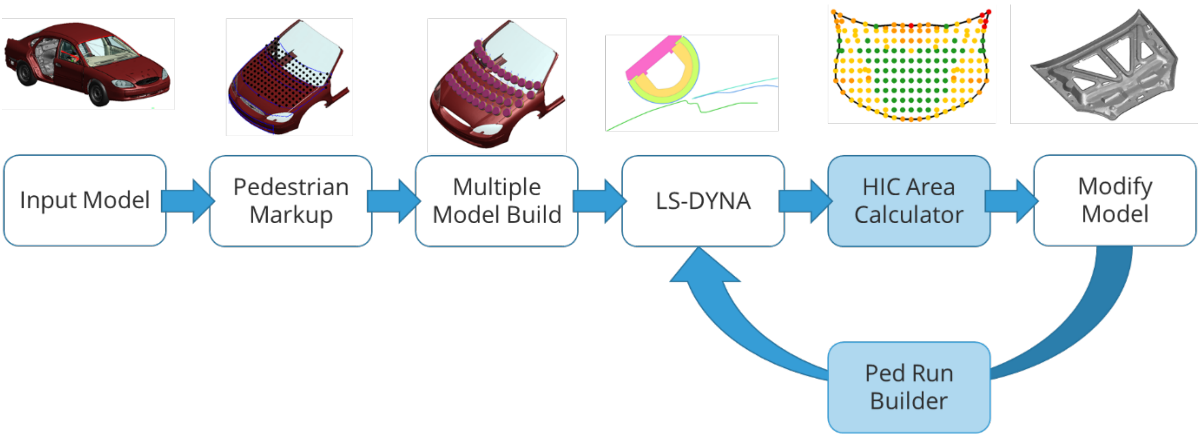


Fig.1: Pedestrian protection workflow in the Oasys LS-DYNA Environment.

2 Overview

The tools within the Oasys LS-DYNA Environment can broadly be grouped into **Mark-Up, Model Build** and **Post-Processing.**

2.1 Mark-up

The mark-up tool within Oasys PRIMER can be used to identify impact zones relating to various protocols. The tool will work out the various reference lines that form the bounds of the impact zones. Once the impact zones are determined, the tool can be used to create target points for head/leg impacts in a number of ways:

- Automatically creating a grid of target points over the zones based on specified spacing
- Manually create target points at specified locations
- Automatically determine "hard points" below the bonnet based on packaging space and create an appropriate target point on the outer surface of the bonnet based on impactor line of flight
- Create "robustness points" surrounding specified target points to investigate the sensitivity of impacting at particular locations

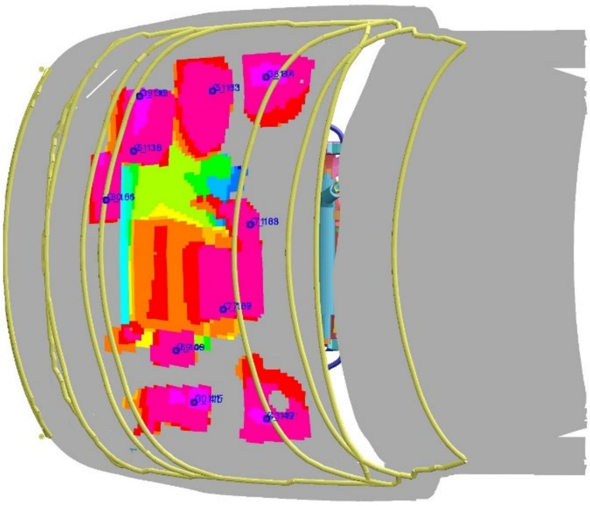


Fig.2: Zone boundary calculation and “hard point” calculation.

2.2 Model Build

Once target points have been specified, the next stage is to build multiple models automatically based on those target points. This is done by:

- Moving the impactor to each target point in turn
- Depenetrating the impactor according to set rules
- Outputting models to run in LS-DYNA with *INCLUDE_TRANSFORM definitions to move the impactor model to the calculated starting positions

Fig.3: Multiple build of many LS-DYNA models based on target points.

2.3 Post-Processing

The final stage is to post-process the many individual LS-DYNA analyses that are created. Reports can be created automatically using built in templates within Oasys REPORTER. This reduces the time required to manually post-process many runs. Various features are available:

- Visualise and analyse results using the Oasys PRIMER HIC tool
- Produce individual reports for each impact

- Produce a summary report of all individual impacts
- Plot results and calculate points scored
- Create contour plots and “areas of HIC” from multiple analyses

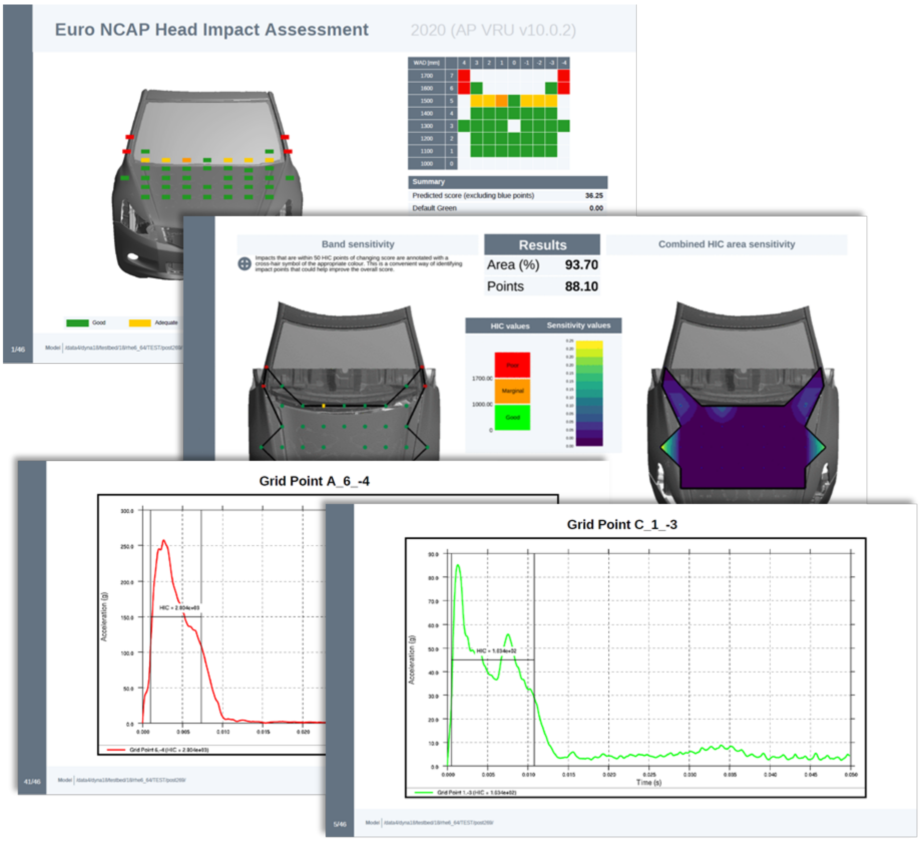


Fig.4: Oasys REPORTER pedestrian protection output

3 Latest functionality

The following new functionality has been added to recent versions, including the recently released Oasys 18.0.

3.1 Enhancements to the HIC Area Calculator tool

The **HIC Area Calculator** is a tool for analysing the many results that are produced during pedestrian head impact. It supports the latest GTR, Euro NCAP and China NCAP protocols, and as part of Oasys 18.0 has several new enhancements. This tool allows you to:

- Import HIC results and plot/contour results within your model
- Calculate “areas of HIC”, or NCAP scores
- Various tools to allow investigation of results, and identify points/areas of interest
- Comparison of results across multiple “sweeps” of analyses

The **HIC Area Calculator** is designed to give you the tools you need to make design decisions quickly within the pedestrian protection workflow.

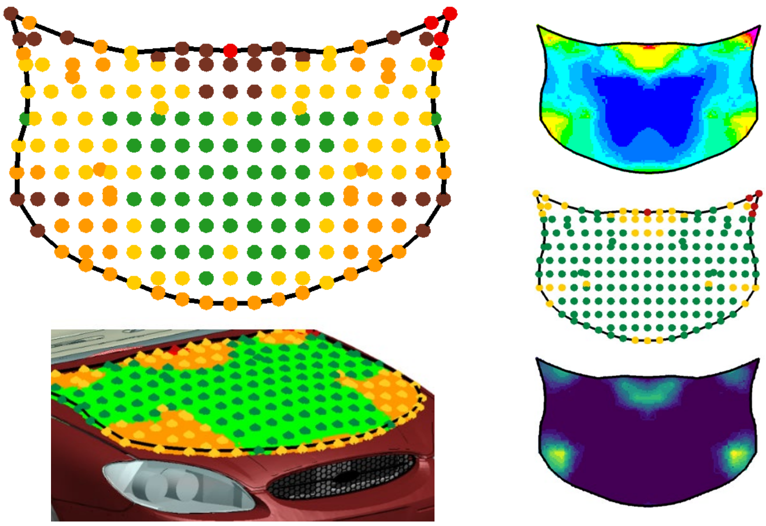


Fig.5: HIC Area Calculator results visualisations

3.2 Introduction of the Pedestrian Run Builder tool

The **Pedestrian Run Builder** tool allows you to easily create a new sweep of analyses, based on an existing sweep, after a design iteration. This is an interactive way of duplicating the input models from an existing set of baseline runs, and making all the necessary changes to update those models to the design iteration you want to analyse. Features include:

- Interactive panel for visualising and selecting impact points for analysis
- Tools for creating "corridors" or radii around points of interest to select regions to re-analyse
- New studies, perhaps including hundreds of input decks, created quickly in a few clicks

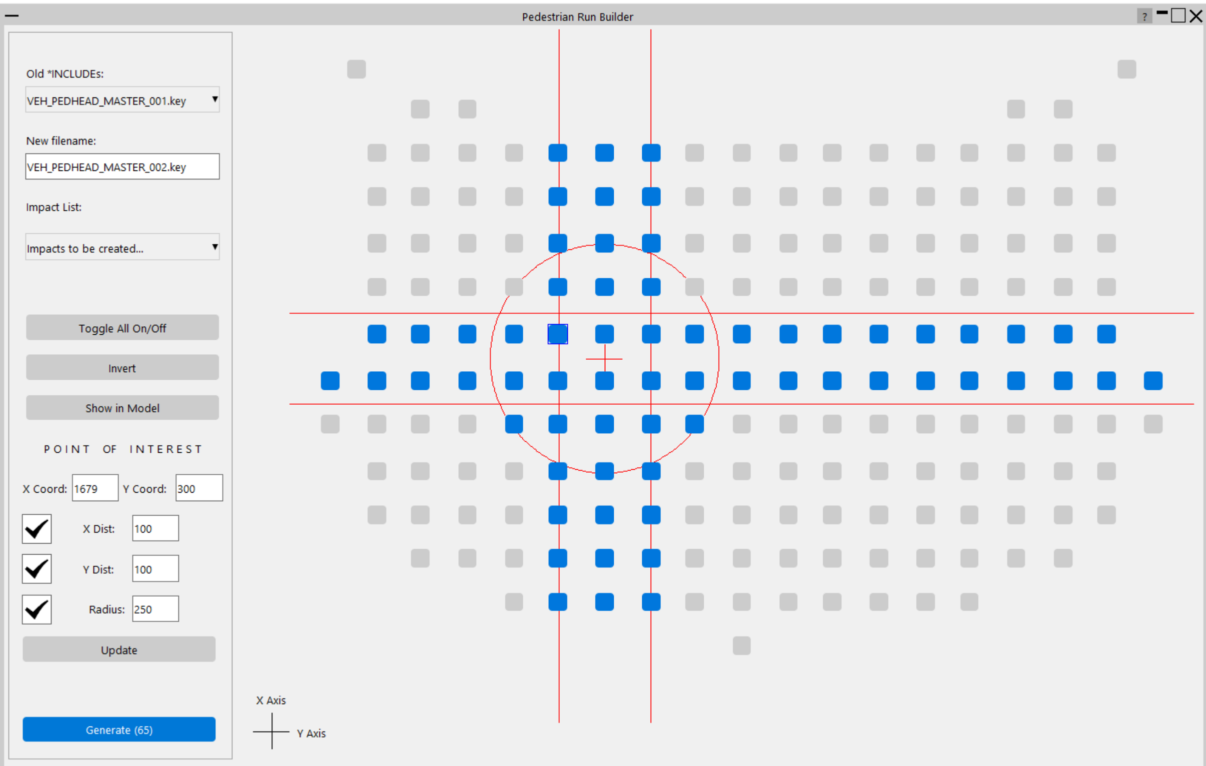


Fig.6: Pedestrian Run Builder

4 Summary

The Oasys pedestrian tools help you to produce models quickly, and spend less time post-processing results. This means input to the design process can be quicker, and you have more time to investigate issues and solutions. The tools are also useful in the initial stages of development for investigating concepts relating to styling, materials, design and packaging. The recent additions of the **HIC Area Calulator** and the **Pedestrian Run Builder** give you further tools to accelerate the pedestrian design-analyse-decision process.

Prediction of temperature induced defects in concrete with LS-DYNA: cement hydration implementation and applications

Mattia Bernardi[1], Fragkoulis Kanavaris[1] and Richard Sturt[1]

[1] Advanced Digital Engineering, ARUP

Abstract

The cement hydration reaction has long been recognized as an important contributor to defects throughout the service life of concrete structures.
As the hydration reaction is highly exothermic, and the thermal conductivity of concrete is relatively low, high temperatures and temperature gradients have special relevance in massive concrete structures. Massive concrete structures can endure significant cracking when temperature induced deformations are restrained. Uncontrolled cracking may compromise the structure durability and reliability, e.g. in massive concrete slabs for rail infrastructures or marine structures or the structure functionality, e.g. watertightness in liquid retaining structures or may even represent an aesthetically unacceptable defect for a concrete structure with demanding architectural finishing requirements.
The heat generation and the consequent temperature rise in concrete structures is also a problem for the damaging effects on the concrete mechanical properties following deleterious chemical reactions such as Delayed Ettringite Formation (DEF). This chemical reaction is known to be associated with thermal fields in early-age concrete usually of the order 65°C to 75°C.

Modelling such temperature effects may result in mitigation of thermal cracking and DEF in concrete. However, to include these effects in a numerical analysis of concrete structures, a special form of heat source dependent on temperature and time is required.
A novel thermal material is introduced to model the cement hydration within LS-DYNA. Three different relatively well-established hydration models were implemented and were calibrated and validated based on experimental results. The selected models were also chosen to accurately capture the thermal activation characteristics of different cement types, such as CEM I or those containing fly ash, ground granulated blast-furnace slag or limestone powder. The calibration of the models was based on isothermal and/or adiabatic calorimeter results whilst the validation was based on adiabatic and semi-adiabatic tests, as well as on comprehensive tests found in literature for given cement types. Herein two case studies where the cement hydration modelling capability in LS-DYNA was used to predict temperatures in concrete elements of considerable thickness are presented. First, the study of delayed ettringite formation and thermal cracking risk of a massive wind turbine base, then the estimation of maximum temperatures and temperature differentials during the early construction stages of a massive foundation raft on piles are presented. Insights on ongoing work and developments are given, particularly towards fully coupling thermal and mechanical analyses accounting also for concrete viscoelastic behavior, drying shrinkage, and cracking characteristics.

Survey of four material models for ballistic simulations of high-strength concrete

Andria Antoniou[1,2], Martin Kristoffersen[1,2], Tore Børvik[1,2]

[1]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, NTNU – Norwegian University of Science and Technology, NO-7491 Trondheim, Norway
[2]Centre for Advanced Structural Analysis (CASA), NTNU, NO-7491 Trondheim, Norway

1 Summary

The protection of critical infrastructure against ballistic impact scenarios is crucial for the safety of the public. Shielding barriers are commonly made of concrete and their design exposes significant challenges. Accurate numerical models of concrete are in great demand especially for conditions involving high stain rates, high triaxial pressures and complicated fracture modes. Today, finite element (FE) constitutive models are extensively used to address the problem.

Three of the most conventional concrete models, i.e., 1)*MAT_072R3 or the Karagozian and Case (K&C) model – Release III [3], 2) *MAT_159 or the continuous surface cap model (CSCM) [2] and 3) *MAT_272 or the Riedel-Hiermaier-Thoma (RHT) model [1], are standard models available in LS-DYNA with the optional setting for automated generation of material parameters. A thorough calibration of these models is notably challenging because of their numerous parameters. Further, a modified version of the Holmquist-Johnson-Cook (MHJC) model [4] has been implemented in LS-DYNA as a user subroutine due to its convenient parameter calibration.

In the current study, we consider high strength concrete (C75) slabs of 50 mm thickness. We calibrated MHJC using laboratory material experiments (quasi-static compression and tension, and triaxial hydrostatic compression) and we verify the accuracy of the model on the ballistic perforation resistance of concrete slabs impacted by ogival projectiles. Then, we evaluate the automatic generation option for the parameter determination of the other three models (*MAT_072R3, *MAT_159 and *MAT_272) by comparing their ballistic response with MHJC in numerical simulations. We adopt the automatic generated parameters and investigate their performance on the laboratory material test. Finally, we readjusted the parameters and re-validated their perforation resistance with impact simulations. The numerical results are compared with experimental data obtained during ballistic impacts tests at SIMLab.

2 Literature

[1] Grunwald, C., Schaufelberger, B., Stolz, A., Riedel, W., & Borrvall, T. (2017). A general concrete model in hydrocodes: Verification and validation of the Riedel–Hiermaier–Thoma model in LS-DYNA. *International Journal of Protective Structures, 8*, pp. 58-85.

[2] Murray, Y. (2007). *Users Manual for LS-DYNA Concrete Material Model 159.* Federal Highway Administration, publication no. FHWA-HRT-05-062.

[3] Magellanes, J., Wu, Y., Malvar, L., & Crawford, J. (2010). Recent Improvements to Release III of the K&C Concrete Model. *11th Inernational LS-DYNA Users Conference, Dearborn, MI, A.US*.

[4] Polanco-Loria, M., Hopperstad, O., Børvik, T., & Berstad, T. (2008). Numerical predictions of ballistic limits for concrete slabs using a modfified version of the HJC concrete model. *International Journal of Impact Engineering, 35*, pp. 290-303.

Non-Isochoric Plasticity Assessment for Accurate Crashworthiness CAE Analysis. Application to SAMP-1 and SAMP-Light.

Eduardo Martin-Santos[1], Lluis Martorell[1], Pablo Cruz[1], Megan Lobdell[2], Hubert Lobo[2]

[1]Applus IDIADA
[2]Applus Datapointlabs

1 Abstract

A deep understanding of advanced material plasticity and fracture is one of the cornerstones of mechanical engineering to overcome present and future challenges in the automotive industry with respect to lightweight multi-material body solutions.
The correct material law selection may imply a design lightweight efficiency improvement of between 10% and 20% depending on the material, component geometry, manufacturing technology and performance requirements. The accurate implementation of the plastic behaviour becomes mandatory when material fracture is a central design parameter.
In this paper, the authors propose a clear process to experimentally measure and assess how far uniaxially tested materials are from pure isochoric plastic behaviour. This process will be named Non-isochoric Plasticity Assessment (NPA). In order to illustrate the process, NPA will be applied to actual experimental results of representative automotive metals and thermoplastics.
Material plastic dilation behaviour is studied. A general description is provided regarding plasticity theory concepts required for the usage of non-isochoric plasticity material laws. An approach for the validation of the experimental input data consistency for both SAMP-1 and SAMP-Light material laws is also proposed.
The overall approach is finally applied and validated on an extruded aluminium and a thermoplastic showing a proper level of correlation between CAE and experimental results for shell-based FE-models.

Keywords: Crashworthiness, material plasticity, material fracture, non-isochoric plasticity, SAMP-1, SAMP-Light, von Mises, eGissmo, MAT_ADD_GENERALIZED_DAMAGE, extruded aluminium, steel, thermoplastics.

Modelling liquefaction of soils with LS-DYNA using a SANISAND-based material model.

Richard Sturt[1], Cihan Cengiz[1], Yuli Huang[2], James Go[1]

[1]Arup, UK
[2]Formerly Arup, USA

1 Abstract

Saturated sandy soils can be prone to liquefaction in earthquakes: the soil loses strength and stiffness due to cyclic shear loading, becoming more like a liquid or quicksand. When liquefaction occurs, structures founded on such soils may experience severe damage, large settlement or may even overturn. Designers of structures in seismically-active regions where potentially-liquefiable soils are present need to assess the likelihood of liquefaction occurring under design-level earthquakes and, if required, provide mitigating measures to ensure the survival of the structure. Three-dimensional nonlinear finite element analysis can be used to understand the effects of liquefaction on a structure and, if sufficient validation of the soil properties has been carried out under a range of stress conditions, can potentially predict the extent of liquefaction that will occur as a result of a given earthquake time-history. However, this requires a soil material model capable of reproducing liquefaction phenomena.

This paper describes the liquefaction phenomenon and shows how LS-DYNA can be used to model it. A material model has been developed based on SANISAND, which is a family of soil models in the framework of critical state soil mechanics and bounding surface plasticity that has become well-established in the academic community over the last two decades for purposes of modelling liquefaction. Numerous researchers have published adaptations of the SANISAND theory that enable key experimental findings to be reproduced, meaning that the choice of algorithms to be adopted for LS-DYNA is not straightforward. The paper illustrates comparisons with experimental results that can be achieved using the new material model, which will be included in a future release of LS-DYNA.

Enabling Interoperability for LS-DYNA Users with Envyo using the VMAP Standard

C. Liebold, T. Usta (DYNAmore)

VMAP enabling interoperability in integrated CAE simulation workflows

Klaus Wolf, Priyanka Gulati
Fraunhofer Institute for Algorithms and Scientific Computing – SCAI, Sankt Augustin GERMANY
Klaus.Wolf@scai.fraunhofer.de Priyanka.Gulati@scai.fraunhofer.de

Gino Duffett
NAFEMS GmbH, Munich, GERMANY
Gino.Duffett@nafems.org

Keywords: interoperability, standards, manufacturing simulation, software interfaces, data mapping, material models, ontologies, simulation workflows

Introduction

With the progress in CAE simulation leading to more complicated and integrated workflows, data control and transfer becomes essential. This is extremely important in the manufacturing industry where complicated simulation workflows are necessary in tracking material changes throughout the manufacturing process. Traditionally this has led to many one-off solutions created by engineers in their desire to solve their design-process problems within the defined timescales. Naturally, the result was that many methodologies for transferring information became available but were stand-alone and/or unavailable to different departments or entities.

The VMAP data standard is the first to address this issue and solve the problem by standardising the data being transferred to enable complete software interoperability. Additionally, VMAP provides a library of IO routines to help engineers speed-up the creation of their workflows thereby removing the emphasis for considering data formats. This enables easier and more flexible data transfer, use of different software for different simulations and the creation of re-useable processes that can be easily adapted to include more or different data. Moreover, this enables software interoperability for post-processing and data manipulation and processing.

VMAP is cost-free and is supported by the international VMAP Standards Community comprising independent software vendors (ISVs), developers, academia and other entities and provides the CAE industry with a focus group to provide guidance, collaborate, evolve and maintain VMAP.

This paper describes the VMAP standard and IO libraries, example of its successful implementation via various use cases. The important roles of the VMAP Standards Community are also described.

The VMAP Standard

VMAP is defined as a vendor neutral standard for CAE data-storage to enhance interoperability in virtual engineering workflows.

The data standard is based on HDF5, a widely accepted implementation platform for many IO related applications. The data formats currently included in the release version 0.5 relate to geometry and discretization, coordinate and unit systems, result and state variables, parameters for (material) models as well as meta- and user-defined data. The list is under continual enhancement.

Many ISVs, both large and small players, will implement the VMAP Standard directly within their software to extract the maximum speed and efficiency, shown schematically in Figure 1.

However, all data defined within the standard can be written or read using the IO software library provided as part of the VMAP Standard, also shown schematically in Figure 1. This enables engineers

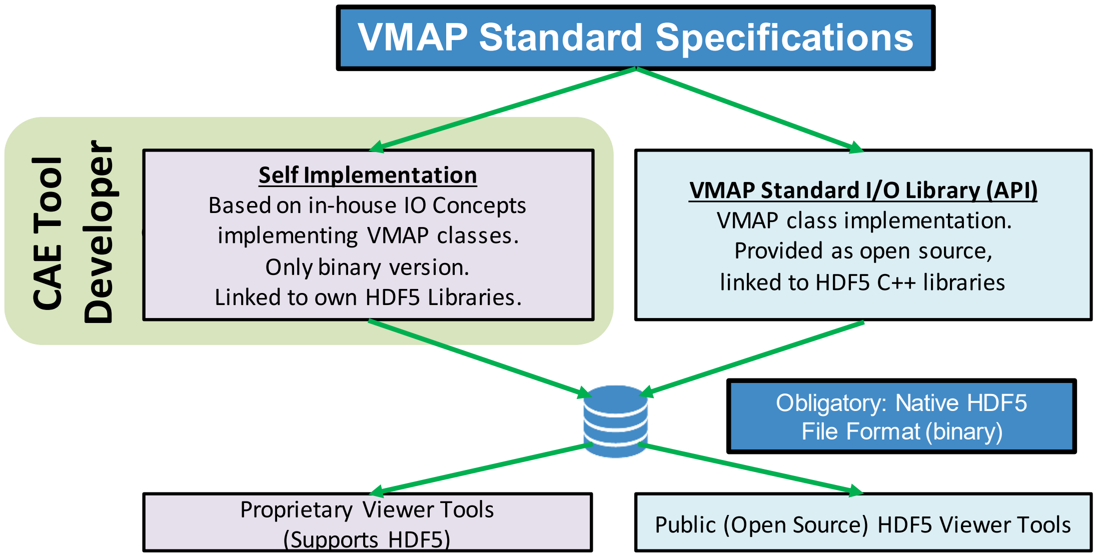


Figure 1: VMAP Standard implementation either within ISV software or utilising the supplied IO software library.

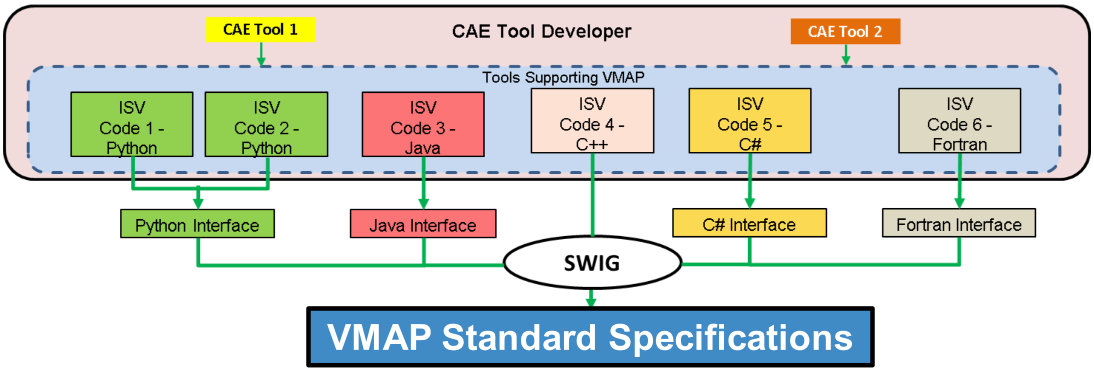


Figure 2: VMAP Standard implementation with other programming/scripting languages via the SWIG wrapper

to utilise VMAP without having any knowledge of the standardised data formats. The freely available SWIG wrapper tool can be utilised to bind the VMAP IO software library into software written in any other programming or script language, see Figure 2. As such the VMAP IO library is universally available.

To aid users and implementers a series of simple test cases are supplied within the release pack so that implementations can be quickly verified.

In summary, the complete VMAP Standard release includes:

- defined VMAP Standard (document),
- use case descriptions and background information (document),
- IO software library (software),
- a set of test cases to verify any implementation (files),
- contact information for the VMAP Standards Community (document).

Figure 3 shows a few ontology levels in VMAP. The green text represents a group, blue text represents datasets and red text represents attributes, all containing further details about a model. The HDF5 format stores the data in a similar format with groups, datasets and attributes.

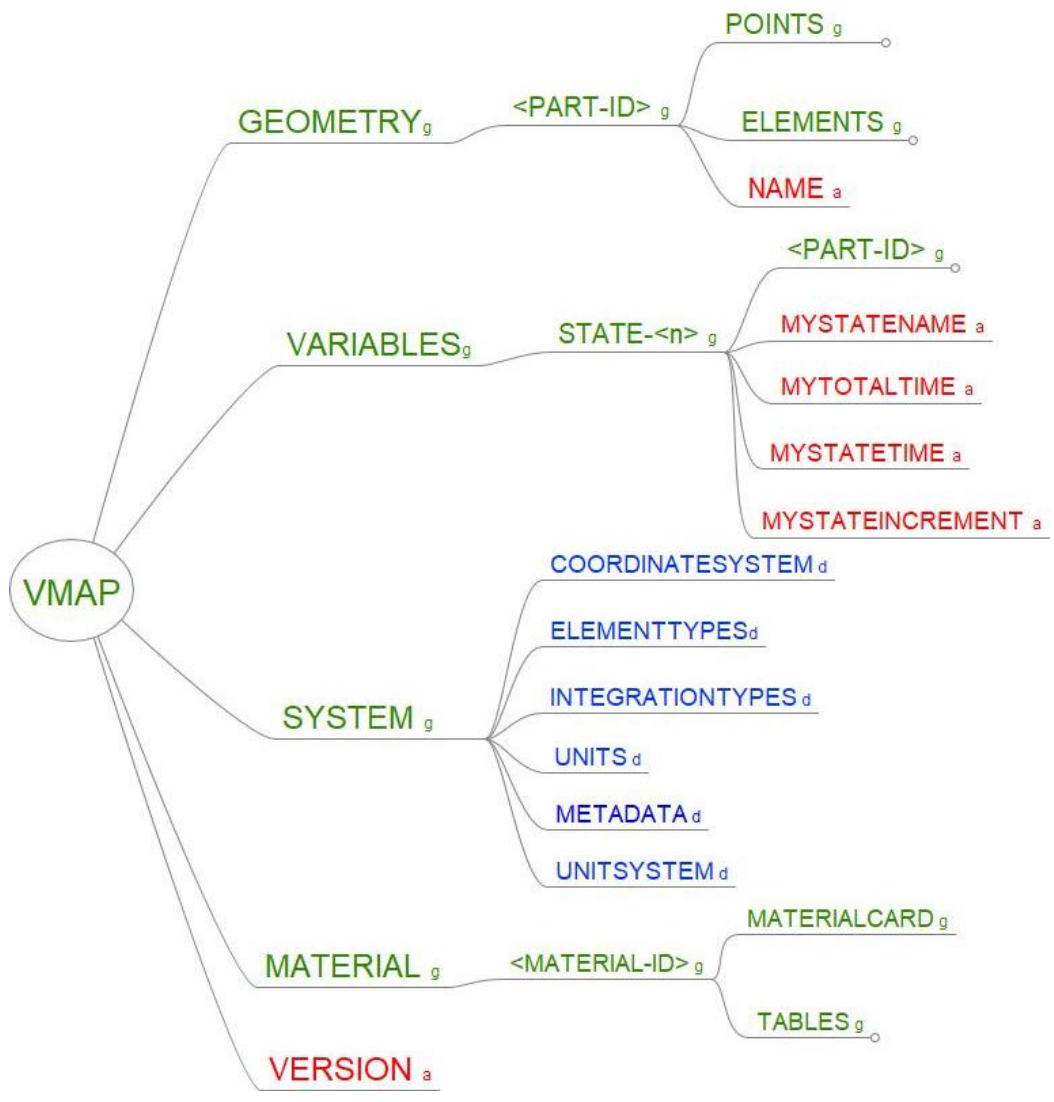


Figure 3: The VMAP Group

As can be seen in the ontology, VMAP is capable of storing dynamic time based analyses. The VMAP library can also offer the possibility to store multiple process simulation steps in the same VMAP file, useful for SMEs who do not wish to invest further in DBMS systems.

COTS Software Implementations

To date the VMAP Standard has been implemented in many Commercial-Off-The-Shelf (COTS) software. This list, shown in Figure 4, is growing with discussions continuing with other groups regarding the VMAP implementation and creation of use cases defined by themselves (not listed).

Use Cases

The VMAP Standard has been verified by many industrial use cases; some examples are shown in Figure 5. All these use cases include complex simulation processes requiring the tracking of material information throughout the process to ensure the correct component design with its desirable characteristics and to accurately design the manufacturing process details. Use cases that are being created in collaboration with entities external to the project are not defined here.

- **Blow Forming**: The blow moulding use case (4 simulation stages, 4 softwares) shows the detailed prediction of shrinkage and warpage of plastic bottles and containers. The product range of extrusion blow-moulded plastic parts ranges from thin-walled packaging products like bottles or cans, to highly stressed technical parts like fuel tanks or intermediate bulk containers (IBC).
  The process simulations provide information, for example, about the wall thickness distribution and the shrinkage and warpage, which significantly influences the product properties of the final part. Therefore, all the information regarding the process history (e.g. temperatures, residual stresses, or wall thickness) needs to be stored and transferred between the different simulation steps. In combination with advanced material models, this integrated simulation approach makes it possible to predict the product properties of blow moulded parts with a very high accuracy.
  More accurate simulation methods enable higher product performance of blow moulded plastic parts with less material consumption and shorter

Software	Platform(s)	Integration Type	VMAP import	VMAP export
4a FIBREMAP		Integrated	v 0.5.2	v 0.5.2
4a MICROMEC		Integrated	(under development)	(under development)
4a VALIMAT		Integrated	(under development)	(under development)
ANSYS Mechanical		External Wrapper	v 0.5.0	v 0.5.0
Autodesk FUSION 360 Platform		Integrated	(technical evaluation)	(technical evaluation)
Autodesk Moldflow		External Wrapper	(under development)	v 0.5.0
Beta CAE Ansa		Integrated	v 0.4.0	v 0.4.0
Beta CAE Epilysis		Integrated	(under development)	(under development)
Beta CAE Metapost		Integrated	(under development)	(under development)
Convergent Compro		External Wrapper	v 0.5.2	v 0.5.2
Convergent Raven		Integrated	v 0.5.2	v 0.5.2
DYNAmore Envyo		Integrated	v 0.5.2	v 0.5.2
ESI Visual Environment		External Wrapper	v 0.4.0	v 0.4.0
e-Xstream Digimat		Integrated	v 0.4.0	v 0.4.0
Forge (Transvalor)		Integrated	v 0.4.0	v 0.4.0
inuTech Diffpack		External Wrapper	v 0.5.0	v 0.5.0
inuTech OOT		External Wrapper	(under development)	(under development)
Magmasoft		Integrated	v 0.4.0	v 0.4.0
MpCCI Mapper (SCAI)		Integrated	v 0.5.2	v 0.5.2
MSC Marc		External Wrapper	v 0.5.1	v 0.5.1
OOFEM		Integrated	v 0.5.2	v 0.5.2
OpenFOAM		External Wrapper	v 0.5.2	v 0.5.2
Simcon CADMould		Integrated	v 0.5.0	v 0.5.0
Simulia Abaqus		External Wrapper	v 0.5.2	v 0.5.2

Legend: Existing tool available
Tool development in progress
Tool in initial development or technical evaluation stage

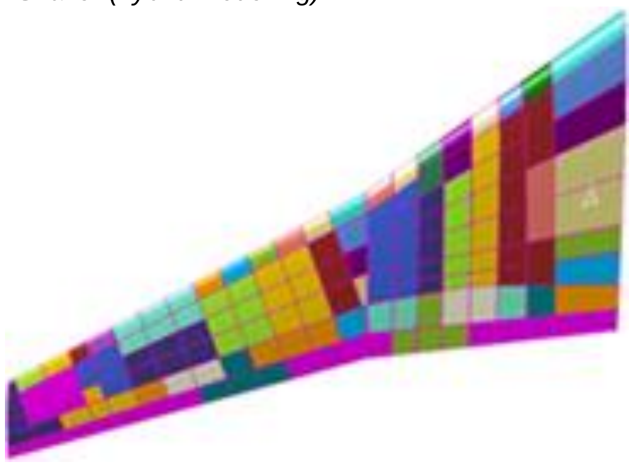
Figure 4: VMAP Standard ISV implementations. The latest release is version 0.5.2 (15 October 2020). All release versions prior to release 0.4.0 are compatible.

design/manufacture cycle times. Due to standardization and automation of the CAE workflow, time consuming data transfer between the different simulation stages can be avoided.
This case is representative for many blow moulded components as the material interfaces can easily be expanded.

- **Composites for Lightweight Vehicles**: The lightweight vehicle composites use case (4 simulation stages, 5 softwares) shows an application of the continuous virtual process chain to a practical-oriented automotive structural part. An automated and accelerated workflow for product verification including both manufacture process and product design is available.
  Mapping of results from solid to shell mesh is the most crucial point here and this has been achieved with a VMAP standardized data storage format. Most common variables include fibre orientation, volume & density.
  Standardization of the data format will facilitate a complete CAE workflow for high-performance composites in relevant structural automotive applications.
- **Injection-Moulding**: This includes four sub cases: Impact, Foaming, Fatigue and Creep. These 4 plastic moulding use cases (2 to 4 simulation stages, 3 to 6 softwares) show the engineering potential of low-cost materials available for lightweight and safety applications.
  Short- & Long-Fibre Reinforced Thermoplastics are used and fibre orientation including weld lines are transferred from an injection moulding simulation to a structural simulation, the main issue being the mapping of fibre orientation and other parameters.
  With a completely integrated simulation process higher product performance with reduced product development times can be achieved.

Detergent bottle (blow forming)

Shaver (hybrid modelling)



Composite aircraft wing

Figure 5: Some components simulated in the defined use cases.

- **Additive Manufacturing (AM)**: The additive manufacturing use for plastic components and parts case (4 simulation stages, 4 softwares) demonstrates the needs and capabilities for AM in plastics. The product performance of additively manufactured parts is highly influenced by the process conditions. Therefore, the whole process history needs to be transferred between several simulation stages also involving different solvers and meshes. The main challenge is the transfer of time dependent boundary conditions from printer to simulation. This can be carried out effectively with the standardized VMAP data storage format.
  A standardized format will help reduce efforts & cost during product development times.

- **Hybrid Modelling of Consumer Products**: The consumer lifestyle use case explores the complex improvement of the production processes and the performance of shaver products. The goal is to have a complete virtual process chain which, prior to VMAP, not been possible. Now, with using VMAP the teams are working on developing a virtual process chain which will enable seamless virtual product development.
- **Composites in Aerospace**: The aerospace composites use case (5 simulation stages, 6 softwares) shows how standardized information can be passed between all the relevant simulation modules for the different phases of commercial aeronautics product development program utilizing the autoclave manufacturing process. The computational fluid dynamics simulation requires a different mesh and boundary conditions than for the thermo-chemical simulation, saturated flow, and stress/deformation simulations. So far, VMAP has been developed with a focus on structural simulation, however, it is possible to store and transfer data from and to other analyses types. Developing the standardized material models to enable process simulation will significantly accelerate and optimize the many simulation steps of an aerospace composite component development program.

VMAP Standards Community

The VMAP Standard Community (VMAP SC) supports and promotes the complete VMAP Standard. Current members comprise ISVs, developers, academia and other entities committed to evolve VMAP into a truly international standard used by the CAE sector. At the date of writing the VMAP SC is being legalized as a not-for-profit association with a founder group of at least 15 entities (the legal statutes are currently being revised by the tax office) and membership will be open to any entity using, involved and working with VMAP.

During 2020, the last year of the project, the VMAP International Conference on CAE Interoperability was held with 25 presentations and about 120 attendees (see vmap.eu.com/vmap-conference-2020/).

All VMAP Standards Community information can be found on the website vmap.eu.com/community

Acknowledgements

ITEA is the EUREKA Cluster programme supporting innovative, industry-driven, pre-competitive R&D projects in the area of Software-intensive Systems & Services (SiSS). ITEA stimulates projects in an open community of large industry, SMEs, universities, research institutes and user organisations.

The Austrian part of the joint project is funded by the Austrian Research Promotion Agency (FFG). (number: Projekt 864080 – EUREKA ITEA 3 2017 VMAP Moulding).

The Belgian part of the joint project is funded by the companies partaking.

The Canadian part of the joint project is funded by the Scientific Research and Development Tax Credit Program (SR&ED)

The German part of the joint project is funded by the German Federal Ministry of Education and Research (BMBF) with 3.5 million euros via the ITEA 3 cluster of the European research initiative EUREKA. (number: DLR-Projektträger, Softwaresysteme und Wissenstechnologien – Funding Sign 01|S17025 A – K).

The Netherlands part of the joint project is funded by the Netherlands Enterprise Agency

The Swiss part of the joint project is funded by the companies partaking.

Virtual Tool Commissioning using LS-DYNA Functional Mock-up Interface

Stefan Heiland[1], Lars Klingel[2], Lars Penter[1], Florian Jaensch[2], Christer Schenke[3], Steffen Ihlenfeldt[1], Alexander Verl[2]

[1]Technische Universität Dresden, Institute of Mechatronic Engineering
[2]University of Stuttgart, Institute for Control Engineering of Machine Tools and Manufacturing Units
[3]Department of Cyber-Physical Production Systems at the Fraunhofer IWU

Abstract

The commissioning of forming tools includes the mechanical spotting of the active surfaces and the identification of suitable actuator set values. It represents a time-consuming and expensive step in the tool development process. A major cause for the necessity of this manual die spotting is the elastic compliance of press and die that results in geometric deviations between predicted and produced part geometry as well as the control characteristics and the resulting accuracy of the drives. The interactions between the machine and the forming process can be computed numerically. An integration of simulation models that includes the machine behavior into the tool development process offers potential for time and cost savings for die manufacturers and machine operators.
In this article, a concept for Virtual Commissioning of forming tools is presented. The aim of this concept is to simulate the basic methodology of the die spotting process and the identification of set values on a virtual counterpart of the press. This so-called "Digital Twin" bases on an extended process model that represents the interaction between process, machine and control.
As a suitable approach, the method of cross-platform Co-Simulation by means of a Functional Mock-up Unit (FMU) is used. LS-DYNA is able to generate FMUs and thereby exchange data with other software via Functional Mock-up Interface (FMI). The LS-DYNA plug-in “FMU Manager” supports FMI 2.0 standard.
The coupled simulation model extends the established approaches for the simulation of forming processes (FE-method) with elastic-static machine influences as well as dynamic influences of the drive and die cushion control. Updating the simulation model with real time data allows to predict the optimum tool topology and process parameters for a specific production press.

Simulation of Hot Plate Rolling using LS-DYNA©

Mikael Schill[1], Jesper Karlsson[2], Hans Magnusson [3], Fei Huyan[3],
Nima Safara Nosar[4], Jonas Lagergren[4], Torbjörn Narström[4], Fredrik Johansson[4]

[1]Dynamore Nordic, Linköping, Sweden
[2]Dynamore Nordic, Gothenburg, Sweden
[3]Swerim, Kista, Sweden
[4]SSAB, Oxelösund, Sweden

Creating a virtual model of a hot plate rolling process involves many challenges. In an attempt to address these, a research project called FINBEAM ("Full Scale Integrated Workability Modelling") was initiated by Jernkontoret and the Swedish steel industry, financed by the Swedish innovation agency, VINNOVA. The purpose was to bring research institutes, industry and software developers together to reach a common modeling ground for simulation based design of hot working processes for the steel industry in Sweden. A proper description of the material behavior is of outmost importance and a material modelling approach was initiated where the material behavior at high temperatures, strains and strain rates is described. At these temperatures, the material will recover between the rolling passes which demands a stress relaxation model to be added as well. The material test basis for calibrating the data for the material model is the Gleeble test and a test scheme was carried out with various deformation rates and temperatures. During hot plate rolling, the material experiences very high deformations which puts demands on the Finite Element model. The way to remedy this was to use a result mapping approach between each roll pass. To accommodate this, LS-OPT© was used as a simulation driver for transferring the result between passes, starting simulations and extracting results. This paper will present the theory behind the material model and the simulation modeling approach together with numerical results compared with physical hot plate rolling passes.

Cross-Sectional Warping in Sheet Metal Forming Simulations

Tobias Willmann[1], Alexander Wessel[2], Thorsten Beier[3], Alexander Butz[2], Manfred Bischoff[1]

[1]Institute for Structural Mechanics, University of Stuttgart
Pfaffenwaldring 7, 70550 Stuttgart, Germany
{willmann;bischoff}@ibb.uni-stuttgart.de

[2]Fraunhofer Institute for Mechanics of Materials IWM
Woehlerstrasse 11, 79108 Freiburg, Germany
{alexander.wessel;alexander.butz}@iwm.fraunhofer.de

[3]thyssenkrupp Steel Europe AG
Eberhardstrasse 12, 44145 Dortmund, Germany
thorsten.beier@steeleurope.com

1 Abstract

For most sheet metal forming simulations, shell elements that consider a reduced stress state, in particular assuming zero transverse normal stress and neglecting the shear stress components σ_{13} and σ_{23} in the yield function, are used. Moreover, certain kinematic assumptions, like cross-sectional material fibers being assumed to remain straight during deformation, are typically applied. However, for some applications, like bending with small radii and thick sheets, this approach is not a workable solution to obtain accurate and reliable results, since the prerequisites that justify the aforementioned kinematic assumptions are not met.

In this contribution, a 3D-shell element is presented that allows for cross-sectional warping. For the evaluation, numerical results of a metal stripe drawn through a draw bead are compared against experimental data. The results demonstrate that the 3D-shell element is able to represent warping of cross-sectional material fibers during deformation. In addition, further numerical tests conducted with this element are shown.

Roll Forming Simulation using Higher Order NURBS-based Finite Elements

Pierre Glay[1], Stefan Hartmann[2]

[1]DYNAmore France SAS, Paris, France
[2]DYNAmore GmbH, Stuttgart, Germany

1 Abstract

Roll forming is a continuous bending operation of a long strip of metal sheet. The sheet is gradually formed through pairs of rotating rolls (called stands) until the desired cross-sectional configuration is obtained. Although roll forming is a classical method to produce constant cross-sectional profiles, it remains a complex process. Finite element analysis (FEA) can assist the designer to improve this process.
During the roll forming process the metal sheet undergoes various states of plastic deformation that need to be properly represented by the finite element model. Depending on the sheet thickness and the radii of the profile, an accurate analysis of the stress field through the thickness is inevitable, which requires suitable volume type finite elements. Some examples of typical roll formed profiles are shown in Fig. 2.

Isogeometric analysis (IGA) is a new finite element analysis method that uses mathematical geometry descriptions from computer aided design (CAD) tools, such as non-uniform rational B-splines (NURBS). Therefore, the standard piecewise continuous Lagrange polynomials are replaced with higher order spline basis functions, leading to higher continuity across finite element boundaries.
In recent years NURBS-based finite elements have been added to the commercial simulation software package LS-DYNA. This paper examines the usability of higher order NURBS-based solid elements in the context of roll forming applications.

Characterization of fragments induced by High Velocity Impacts and satellite additional shielding protective structures evaluation

Tess Legaud[1], Vincent Lapoujade[1],

[1]DynaS+, 5 avenue Didier Daurat 31400 TOULOUSE France, t.legaud@dynasplus.com

Abstract

A substantial number of debris coming from human production gravitates around the Earth. Their size, nature, orbit and velocity can extremely vary, but all these debris represent an increasing collision risk and a threat for the current and future spatial activity. The spatial industry is looking for solutions to limit this risk, by better controlling objects in orbit throughout their whole life and by improving their structures protective capabilities.

All debris can be classified depending on their size. The ReVuS European project highlighted that the most dangerous debris, according to the satellite mission failure probability, have a diameter from 1mm to 5mm.
Consistently with this statement, the aim of the ATIHS project, funded by the French Occitanie region, is to improve the satellite protection against millimetric debris impacts. Various solutions can be explored in order to do so, ATIHS focuses on the local shielding one.

As a consequence, the project global aims consist in:
- Improving the satellites resistance on strategic locations to prevent mission failure,
- Working on the secondary debris generation limitation during a non-lethal impact in order to minimize the satellite contribution to the debris increase.

The project is composed of three main tasks:
- Evaluating new material solutions potentially optimizing mass/resistance ratio,
- Designing, building and evaluating new disruptive hypervelocity testing devices enabling significant increase in impact velocities (goal: 10 to 12 km/s for millimetric to centimetric projectiles),
- Setting up predictive numerical methodologies in order to both increase the capabilities and the reliability of high velocity impact simulations.

This paper focuses on the hypervelocity impact response. It especially deals with the evaluation of new structures and their SPH modelling to better protect the satellite equipment. A method has also been developed in order to evaluate the size and density of the fragments ejected from a first impact. The results, as a function of impact velocity and impacted structure material, will be presented.

A study on blast-loaded aluminium plates with crack-like defects subjected to blast loading

Henrik Granum[1], David Morin[2,3], Tore Børvik[2,3], Odd Sture Hopperstad[2,3]

[1]Enodo AS, Trondheim, Norway
[2]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), Trondheim, Norway
[3]Centre for Advanced Structural Analysis (CASA), NTNU, Trondheim, Norway

1 Abstract

This paper presents a study on AA6016 plates in temper T4 subjected to blast loading. Four different crack-like defects have been introduced in the plates to facilitate crack propagation as the dominating failure mode. Uniaxial tensile specimens extracted from a plate are used in the calibration of the *MAT_258 material model available in LS-DYNA. This material model contains a non-quadratic yield surface, isotropic work hardening and a failure model where the onset of failure is dependent on the element size as well as its bending-to-membrane loading ratio. Four different element sizes are investigated to assess the ability of the model to predict the onset of failure and subsequent crack propagation in blast-loaded plates by comparison to experiments conducted in a shock tube facility.

Inconel 713 and TiAl turbine blade impact test validation with LS-Dyna, including Inconel 718 casing and failure models

Kevin Manzanera[1], Izei Catalina[1]

[1]ITP Aero, Industria de Turbo Propulsores, S.A.U. Zamudio – Spain

Abstract

Motivated by the necessity of validating new materials for future turbines, a set of Blade Crush Tests have been performed with Inconel 713 blades, TiAl blades, Inconel 718 casing material and steel plates. The objective of these tests is to study separately the deformation of a blade during a containment event (configuration 1 tests), and the damage of the casing caused by the impact of different blades (configuration 2 tests). The results of these tests are validated with LS-Dyna analysis, providing a reliable tool for predicting the containment capability of the casings and the out of balance progression in a blade off event. This will allow to assess the containment capability of future designs without the need of large and very costly test campaigns or service experience.

The results and analysis validation obtained from configuration 1 tests are representative of the initial impact sequence of a containment event with Inconel 713 and TiAl blades. Inconel 713 blades will bend after tip contact with the casing and the shroud is likely to break. TiAl blade will shatter after tip contact with the casing and cracks will propagate through the aerofoil. Configuration 2 does not fully represent the damage and impact sequence expected in a containment event. The damage observed in these tests is likely to be higher than the expected on a containment event. Cracks observed in the plates correlate well with the analyses. This suggests that a containment analysis would also correlate well with a real containment event, even though the impact sequence is slightly different.

The work presented in this paper shows the capability of ITP Aero to perform impact rig tests that represent the initial containment impact conditions. This will allow to test and compare the impact behaviour of different blade and casing materials in containment like conditions.

Predictive Engineering using Dfss of IBM Power9 System

Arshad Alfoqaha[1], Kevin O'Connell[1], Eric Campbell[1], Mehdi Hamid[1]

[1]International Business Machines Corporation, Rochester, MN, USA

Developing reliable and robust designs of servers and supercomputersis critical to IBM Systems‘ vision and continuing its global leadership. Shorter development cycle, increasingly dense product designs, along with advanced design features create real challenges in predicting mechanical performance of servers, such as IBM Cognitive Systems' POWER9 portfolio of servers. The 2U version has DDR4 RDIMM's, POWER9 HLGA processor modules, PCIe Gen3 and Gen4 slots, blowers, hard drives, and internal storage controller slots.

To meet IBM shipping standards to ensure POWER9 system's reliability and design robustness, ANSYS Mechanical and LS-DYNA FEA models were developed and physical tests were conducted. The ability to apply accurate predictive engineering methods using simulation early in the design cycle predicted the structural performance of the servers which shortened its development time significantly, reduced number of physical builds, and therefore, reduced development cost.

Also, FEA simulations, both deterministic and probabilistic, of the main printed circuit board (PCB) were performed to evaluate the effect of plugging and unplugging components on the robustness and the reliability of the main PCB. Components plugged into the main printed circuit board include both PCIe, and RDIMM. Manufacturing tolerances, and dimensional and materials variation are taken into account to ensure accuracy. Initially, the deterministic FEA approach is employed to select the worst case plugging scenario. Then, the probabilistic approach of the worst case plugging scenario is employed using Design for Six Sigma (DFSS) techniques to generate the full factorial Design of Experiment (DOE), regression analysis, Response Surface Modeling (RSM), and to obtain the probability density function to predict the probability of meeting design requirement.

***keywords Supercomputers, FEA, Mechanical Performance, Design for Six Sigma**

Fig.1: 2U POWER9 System (Cover off)

Numerical Modeling of Aluminum Forgings;

Issues of Material Failure and Element Formulation

Fatemeh Hekmat[1], Paul DuBois[2]

GM Canadian Technical Centre, Oshawa, Canada
[2]Consultant, Northville, USA

Abstract

The increased use of castings, forgings and thick extrusions in vehicle structure has led to the need for modeling certain parts with solid elements in crash simulations. Since the geometries of the considered parts are typically highly complex, using tetrahedral elements seem to be the practical solution. In the LS-DYNA simulation software, a multitude of different tetra element formulations are available however there exist the uncertainty whether these elements can be used with confidence in a simulation for accurate fracture prediction and with respect to the material failure model. It is well known that linear tetra elements suffer from locking phenomena, though this problem can be partially eliminated through averaging of the hydrostatic pressure term (the so-called element formulation 13). However, recent studies have shown that even in quadratic tetra elements the locking phenomenon is not fully eliminated and continues to be dependent on the specific element formulation. In addition, the number of integration points used in quadratic tetrahedral elements and their location which is potentially quite different from those in a regular hexahedral element contributes to the existing uncertainty.

As result engineers are justifiably concerned about the accuracy of the predictions made with models based on tetrahedral elements but are left without a practically viable alternative.

In this study we describe the development of a material and failure model for an aluminum forging with very high ductility. Whereas the high ductility of the material already makes this a non-trivial task as we need to consider mesh dependency, regularization and damage coupling, the task is made more challenging by the need of using tetra elements. While isotropic behavior was assumed for the material, specific attention was given to the yield curve in the post-necking region and a full 3D failure model with failure surface and instability surface in function of stress triaxiality and Lode parameter were developed. The instability surface corresponds to incipient diffuse necking (and thus mesh dependency) and is based on a 3D generalization of the Swift criterion. Through number of numerical solutions for a high number of test specimens a fracture surface has been developed. We selected the quadratic type 16 tetrahedral element with the Cosserat continuum formulation for this project believing this to be easily the most powerful tetra element formulation in the LS-DYNA code. Comparisons were done with the more traditional linear and locking corrected type 13 element as well as with models built from hexahedral elements, the latter serving a baseline comparison. It is found that for most of the specimens tests the response of the selected tetrahedral elements was well behaved, however locking phenomena were still observed in a small number of simulations leading in some cases to overestimated force levels and in some cases invalidating the failure surface. The conclusion of the authors is that numerical results from models based on tetrahedral meshes, although reliable in majority of cases, are not on par with results obtained with hexahedral models.

PDC electrical cable modeling using TRUSS elements

Bruno Pockszevnicki[1], Vinicius Carvalho[2], Raj Rajagopalan[1]

[1]bruno.pock@stellantis.com

[2]vinicius.carvalho@external.stellantis.com 1raj.rajagopalan@stellantis.com

1 Abstract

This study aims to present a proposal for finite element modeling for electrical cables of a PDC to improve the response of the virtual analysis during design phase, establishing a good interation of electrical cables during crash tests. The objective of this study is to present types of elements and contact pairs that are capable of predicting the response of electrical cables.

In current vehicles, the use of PDC units is an increasingly larger reality. In attempts for improvements they tend to occupy a larger space, which create an additional challenge to the vehicle architecture project in positioning it.

The positioning should be done in order to avoid cable ruptures and anchoring of the PDCs as much as possible. As the cables are energized there is always a risk of short circuit, for this reason, vehicle safety must be ensured by avoiding or minimizing the interaction between cables and components of the engine compartment.

In this way, a better definition of the form that the cables interact among themselves and between engine elements, led us to present contact formulations that can improve the understanding of the phenomenon.

In addition, when electrical cables interact with other components, they suffer tensile stress and must not be able to withstand compressive loads, for this, different formulations are presented as the type of finite element to be used, as in the case of TRUSS elements instead of BEAM which are able to withstand moments and compressive loads.

In order to guarantee and validate the modeling in the vehicle, a simplified model test was also carried out to assess the condition of the cable, as well as the physical tensile test.

KEYWORDS: **PDC electrical cables, Truss elements, Crash tests.**

Vortex: A New Structural Optimisation Solver for Light-weight Holistic Automotive Design

D. Russell (Vortex Engineering); N. Kalargeros (Jaguar Land Rover)

As the automotive industry heads towards electrification and autonomy, huge challenges have been placed upon OEMs and the supply chain to develop new vehicles, new architectures and new design philosophies. Big advances in automotive design have been seen over the past several decades, driven in part by enhanced numerical simulation capacity, where engineers and designers are presented with ever increasing datasets to better inform decisions. However, with modern OEMs now possessing the throughput to produce several thousand full vehicle simulations per day, engineers and designers are now increasingly becoming the limiting factor in the speed of design and in final performance. To address this, Vortex Engineering Group with support from Jaguar Land Rover have created a holistic topometry based iterative optimiser that works directly with LS-DYNA, with the intention to allow the user to optimise full-vehicle structures for a wide range of structural attributes including: crashworthiness, strength, stiffness and modal. Often attributes share deep fundamentals with regards to design and thus complement each other, a phenomenon that is often not fully exploited when disciplines are treated separately. A roof that is well designed to meet the requirements for roof-crush, will likely have a higher modal performance than a badly designed structure, as the badly designed structure will be heavier. The ability to work with multiple load-cases and multiple materials across several domains holistically, means that there is potential for very efficient and robust structures to be obtained. The code is designed to be run continuously throughout a vehicle design programme, where the analyst can interact, modify and introduce DFM constraints, parts or features on the fly to guide the solution from a fully generative design space driven optimisation to a feasible, tangible, pragmatic solution. For generative design the optimiser uses a novel body-centred cubic shell lattice as a design space surrogate. This topography-based methodology is sufficiently numerically lightweight in comparison to topology, allowing for detailed architecture development for complete vehicle systems. Alongside these and other methodologies we will be presenting several case-studies ranging from simple component level optimisation, through to full vehicle optimisation using open-source full vehicle crash models.

Determination of Material Modeling Parameters using LS-OPT Based Optimization Technique

Amritha U[1], Kottresh Kurudimath[1], Subhransu Mohapatra[1],
[1]SABIC

Abstract

Material models for high-performance thermoplastic polymers need to capture highly complex material characteristics accurately to enable lightweight and sustainable designs. Some thermoplastic polymers display dissimilar behavior under impact loading during Dynatup test compared to uniaxial tensile testing at high strain-rates. This necessitates modeling the tri-axial state of stress and damage characteristics with sufficient detail rather than relying on the uniaxial tensile data alone. This paper deals with predicting this complex state of stress and damage behavior using *MAT_187 and GISSMO in LS-DYNA® software, respectively. *MAT_187 predicts biaxial behavior by considering the stress-strain curves generated from uniaxial tension, uniaxial compression and pure shear tests. Performing these tests are well established and relatively robust for polymers in comparison to biaxial test. GISSMO model is adopted to predict damage and failure. Four key damage parameters, namely “critical plastic strain”, “damage exponent”, “fading exponent” and “biaxial plastic strain to failure” which are hard to determine through experiments, are optimized using meta-model based optimization technique in LS-OPT®. The finalized material card with optimized parameters are validated against physical test results and that accords confidence in using these cards in real-life application designs.

LS-OPT® Status Update

Nielen Stander[1], Anirban Basudhar[1], Imtiaz Gandikota[1],

Katharina Liebold[2], Åke Svedin[2], Charlotte Keisser[2]

[1]LST LLC, Ansys, Livermore, CA

[2]DYNAmore GmbH, Stuttgart-Vaihingen, BRD

1 Introduction

LS-OPT Version 7.0 is the last update of the original ‘free‘ version of LS-OPT developed since 1998. The development over the last year has emphasized integration with Ansys products. Among other developments, this involves the transfer of features to and from the optimization code optiSLang. Part of this integration involves the introduction of the Metamodel of Optimal Prognosis [1], an automated metamodel selection method introduced in optiSLang, into LS-OPT Pro. A vehicle crash example is presented in the paper to demonstrate some of the capabilities in comparison with LS-OPT metamodels. As part of the integration process, several LS-OPT features are also being transferred to optiSLang. The enhanced product, *LS-OPT Pro,* will be licensed as an *Ansys-LST* product.

2 *LS-OPT Pro* Release 2022 R1

The new version of LS-OPT will be released early 2022 as a licensed *LS-OPT Pro 2022-R1*. This means that LS-OPT becomes part of a 3-tier series of optimization programs featuring LS-OPT Pro – optiSLang Premium – optiSLang Enterprise. LS-OPT Pro has the following main new features:

- ***Point Mapping****:* In addition to quadrilateral shells, point-based mapping has been extended to triangular shell elements as well as most solid element types supported by LS-DYNA. This is a useful feature since most meshes are not uniform and automatic meshers often use tetrahedra (for solids) or triangles (for shells). This feature allows extraction of results at arbitrary spatial locations that need not coincide with nodal points.
- ***LS-OPT Extractor****:* The Extractor, a standalone executable, has allowed for easy integration of LS-DYNA into other Ansys products. In addition to direct extraction, crash criteria, mathematical expressions and other special functions such as similarity measures are made available. The Extractor is now supported with a GUI to define the extraction commands.
- ***LS-Reader*** integration: LS-Reader, a comprehensive LS-DYNA interface, has replaced the native LS-OPT/LS-DYNA interface. The issue of occasional extraction failures is now resolved in the new interface. Currently only the *d3plot* interface has been replaced, this being the major part of LS-Reader. LS-Reader is integrated with the aforementioned standalone Extractor, as well as *LS-TaSC*. The plan is to expand the current LS-DYNA interface in LS-OPT to support all the result types available in LS-Reader such as Acoustics, SPH, CFD, etc.
- ***A CORA interface*** has been added for seamless integration of ISO 18571 standard rating for dummies and barriers [2,3] into an LS-OPT process flow. The response interface allows the extraction of similarity measures for individual load cases, signals and injury criteria or for all of them together. The CORA interface is selected as a stage type.
- ***Metamodel of Optimal Prognosis (MOP)****.* This is a metamodel-based method, integrated from optiSLang (an Ansys product), which relies on the automatic selection of metamodels [1]. Future versions will integrate all the metamodels currently available in LS-OPT (Polynomials, Feedforward Neural Networks, Radial Basis Function Networks, Support Vector Regression) into an extended MOP feature (referred to as MOP-X). This automates the selection from a wide array of models.
-

An outlook on the following *LS-OPT Pro Version 2022 R2* is also described in the conference paper.

3 References

[1] Most, Thomas and Will, Johannes. (2008). Metamodel of Optimal Prognosis - An automatic approach for variable reduction and optimal meta-model selection. 10.13140/2.1.2194.4007.

[2] Gehre, C. et al; “Objective rating of signals using test and simulation responses”; *21st International Technical Conference on the Enhanced Safety of Vehicles Conference (ESV)*; Stuttgart; Germany; 2009; Paper 09-0407.

Topology optimization of an automotive hood for multiple load cases and disciplines

Imtiaz Gandikota, Willem Roux, Guilian Yi

Livermore Software Technology, an Ansys Company

Abstract

To reduce the head impact injuries in case of traffic accidents, the design of an automotive hood must consider many design requirements including impact of the head against the hood at different locations, lightweight but with enough stiffness to resist various loads imposed on the hood, and NVH characteristics such as the fundamental frequency. Methodologies to solve this type of design optimization problem that integrates multiple design criteria are rare to non-existent in the automotive design field. This paper shows how to conduct the worst-case design of the hood for multiple head impact locations, which is required by the pedestrian safety code. In addition, a topology optimization problem of the hood that combines statics, impact, and eigen frequency load cases is solved by using LS-TaSC to provide the optimal lightweight hood structure satisfying the design constraints. This is possibly the first demonstration of both the worst-case design and multi-disciplinary design optimization considering both impact and frequency load cases on an industrial problem.

Keywords: Automotive Hood Design, Head Injury Criteria, Worst-case Design, Multi-disciplinary Design

Modelling of Fracture Initiation and Post-Fracture Behaviour of Head Impact on Car Windshields

Karoline Osnes[1], Sebastian Kreissl[2], Jonas D'Haen[2], Tore Børvik[1]

[1]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, NTNU – Norwegian University of Science and Technology, Trondheim, Norway
[2]BMW Group Research and Innovation Center FIZ, Munich, Germany

1 Summary

More than half of all road fatalities involve vulnerable road users, such as pedestrians and cyclists. When involved in crashes with cars, the head is particularly susceptible to injuries, and especially if the road user hits the windshield of the car [1]. Impact tests, using a spherical headform, are often performed to estimate the risk of head injury during such an event. In this study, we aim to recreate the acceleration-time history of a headform during an impact test on a windshield through finite element (FE) simulations in LS-DYNA. The test was performed at BMW's research and innovation centre in Munich.

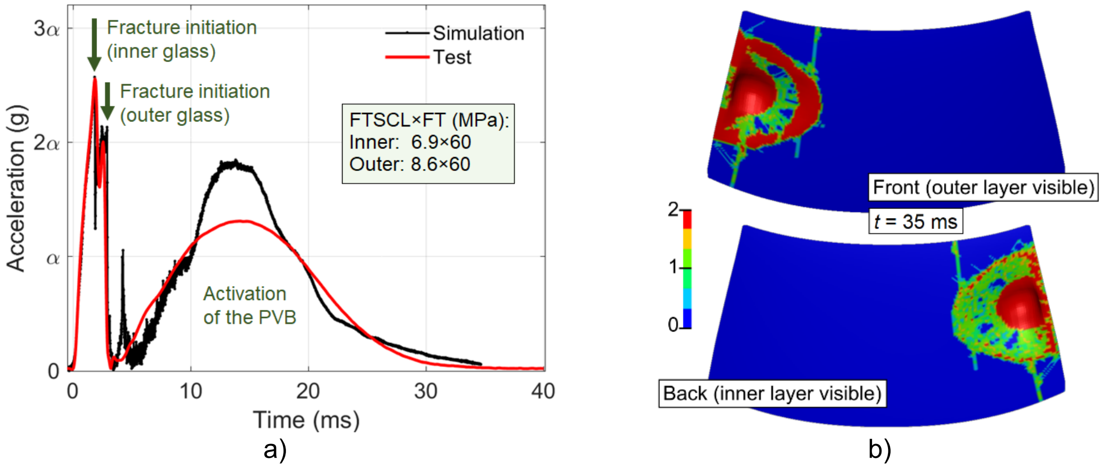


Fig.1: Simulation results: a) acceleration-time curve compared to the test, b) glass fracture pattern

In the impact test, the headform was launched towards the windshield at 35 km/h. The LS-DYNA model comprised of the windshield (glass + PVB+ glass) and a frame. The PVB was described by **`*MAT_24`**, while **`*MAT_280`** was used to model the glass. Thus, glass fracture initiation occurs at stress values of FTSCL×FT. After initiation, failure occurs at FT. FTSCL was used to capture the high stresses necessary to initiate fracture at the correct time, and the rapid crack growth in glass.

Figure 1a presents the acceleration-time curves in the simulation and the test. The initial acceleration tops with a subsequent drop are captured very well. However, in the stage where the PVB is activated, the accelerations are too large. This is most likely related to the PVB stiffness. Figure 1b shows the crack pattern, where the legend denotes the number of cracks.

Please note that FTSCL×FT were chosen to match the physical test, however, the fracture strength of glass is probabilistic [2]. Thus, the parameters used would not fit all tests. To estimate the variation in the fracture strength, we employed a "strength prediction model" [2]. The model results corresponded well with physical tests for the inner glass.

2 Literature

[1] Otte D., Jänsch M., Haasper C.: "Injury protection and accident causation parameters for vulnerable road users based on German In-Depth Accident Study GIDAS", Accident Analysis & Prevention, 44, 149-53, 2012.
[2] Osnes K., Hopperstad O.S., Børvik T.: "Rate dependent fracture of monolithic and laminated glass: Experiments and simulations", Engineering Structures, 212, 110516, 2020.

Experience a complete virtual crash and safety laboratory with the aid of the ANSA pre-processor

Athanasios Lioras[1], Athanasios Fokylidis[2], Emmanouil Dagdilelis[3], Stavros Porikis[4]

[1]BETA CAE Systems S.A.
[2]BETA CAE Systems S.A.
[3]BETA CAE Systems S.A.
[4]BETA CAE Systems S.A.

1 Abstract

With safety protocols and regulations becoming increasingly enhanced, safety analysts try to keep up replicating all possible crash event scenarios in laboratories using specifications that frequently change. In this pursuit, it is crucial for analysts to have at their disposal accurate and robust digital models that enable the tune and study of any real crash event parameter.
Through the ANSA pre-processor, BETA CAE Systems offers an extensive crash and safety portfolio of automated tools for simulation modelling, to create a complete “virtual crash and safety laboratory”. Such tools include this of the seat and the dummy guide, from the identification of HPOINT of the seat to the positioning of the coupled restrained seat-dummy system according to a regulation or a test data position, available not only for standard crash dummies but also for human body models. Pedestrian and Interior tools ensure the proper marking of the exterior and interior of the vehicle but also the accurate positioning of the headform to the desired targets. The Impactor tool enables the positioning of barriers/impactors according to all available regulations. Moreover, the Knee Mapping plugin helps the analyst avoid knee modifier, while Airbag stitching and folding tools set up the pre-crash simulations required for the proper treatment of the airbags.
The current paper presents all the afore mentioned tools and more handy features that crash and safety analysts need to set up detailed and accurate models for different regulations fast, and with the minimum human interaction.

CASE STUDY - Material models for depiction of unloading in low speed crash applications

B. Hirschmann[1], Y. Nakagawa[2], N. Matsuura[2], H. Pothukuchi[1], M. Schwab[1],

[1]4a engineering GmbH, Traboch, Austria
[2] Honda Motor Co., Ltd, Tochigi, Japan

Abstract

The knowledge of mechanical behavior of polymer materials is essential for the simulation of dynamic load cases. In the case of low speed crash in the automotive sector, the unloading behavior and prediction of plastic deformation also assumes a greater importance. The prediction of the unloading behavior of the plastic material helps in determining the structural integrity of the plastic part in consideration and thereby evaluate if the part needs to be changed or can be used further after a low speed impact. This paper deals with comparing the possibility of modelling this unloading behavior using existing models in LS-DYNA® i.e. models with linear viscoelasticity - Prony series (*MAT_124, *MAT_SAMP-1, *MAT_ADD_INELASTIC), models with additional unloading options (*MAT_101) and non-linear elasticity (typical models for foams or rubber). The definition of a suitable test plan at the coupon level with the IMPETUS® system that is representative of the real loading situation is also a part of this study. This initial pre-study not only focused on the depiction of the unloading behavior but also on the numerical costs involved in using the material models. After an initial pre-study, the *MAT_SAMP-1 model has been investigated in detail with the aid of the software solution VALIMAT® to explore its full potential in modelling the unloading behavior using the options of elastic damage and viscoelasticity. In this approach, a damage curve is added, and the hardening curves are compensated. With this approach, in 3-point-bending tests at the corresponding velocity representative of a low speed crash, the deformation behavior could be well captured. The next part of the study is using this developed material model in a full vehicle simulation in a low speed rear impact. Our partner Honda has conducted extensive tests to obtain information on the bumper deformation in this loading scenario. The results from the full vehicle simulation were compared to the test data by Honda and a good correlation to the bumper deformation was found. The simulative prediction of the return in the bumper deformation correlated well to the deformation observed from the tests.

Assembly of full-vehicle digital crash models using ANSA techniques

Alexandros Kaloudis[1]

[1]BETA CAE Systems International AG

Abstract

The assembly of full-vehicle digital models for various crash simulations is always a challenging task. The constant increasing level of detail and size of these models, along with the demand of building models for all variants of a vehicle, make it necessary to create tools that will automate this process and at the same time relieve the user of the difficult task to consider and manage many details during the assembly process. Using ANSA Subsystems and Assembly Points we have developed a process that : i) can handle and assemble multi-variant subsystems, such as the BiW, the engine, the corresponding subframes, etc. The assembly takes place in a lightweight manner, concentrating to the important areas of intermodular (between subsystems) connections, ii) can handle multi-instantiated subsystems, e.g. the wheels, the front seats, etc. iii) reads in a simulation with include-structure, and automatically transforms it into the appropriate entities, ready to be used by the tool in subsequent assemblies, iv) supports various types of intermodular connections, v) is able to handle multiple load cases applied on the same or multiple models, vi) provides a graph for the lifecycle and use of the various subsystems.

Coupling feedback control loop-based model in Simulink to finite element model in LS-Dyna: Application to reposition forward leaning occupant to upright posture

Stefan Schilling, Anurag Soni, Heiko Hinrichs, Christian Verheyen, Martin Grikschat, Burkhard Eickhoff, Andreas Lucht, Alexandru Cirstea

Autoliv B.V. & Co KG

1 Objective

In autonomous vehicles, the forward leaning posture of occupant is expected more frequently. A strategy to provide optimized protection to a forward leaning occupant using existing restrain systems is to bring the occupant back to the upright position before the crash by a mechatronic belt pre-pretensioner (MBPPT) system. Repositioning simulations can be carried out by modelling the MBPPT effectively using imposed force-time characteristics with assumed maximum force and ramp-up time in LS-Dyna. In a more advanced approach, a Simulink MBPPT feedback control loop model is coupled to interact with the occupant model in LS-Dyna.
The objective is to compare and rate two modelling methods against physical tests. First the defined force-time characteristics and second a coupled Simulink feedback control model. The belt forces, kinematics and repositioning time were compared.

2 Method

MBPPT mechatronic system properties are modelled in Simulink with a feedback control loop. The motor properties in the model correspond to the motor used in physical tests. Model.CONNECT is used to couple the Simulink model with a LS-Dyna environment which consists of a UN R16 Manikin in a 50° forward leaning position seated on a UN R16 steel seat. For evaluation, outcomes of both the methods are compared using CORA rating against the physical test results.

3 Results

Final position, belt force and repositioning time differed between both methods. The belt force in feedback control loop method reached 350N in 140ms with degressive characteristics afterwards. The manikin was brought to a nearly upright position at 600 ms. Belt force in force-time method followed the input curve and reached 350 N in 90 ms and then maintained constant level. The manikin was brought to full upright position in 420 ms.
The results from feedback control loop model achieved a higher CORA rating compared to test, where the force-time method achieved a lower CORA rating.

4 Discussion

The belt force in feedback control loop method peaked to overcome inertia of the manikin. The motor rotation speed increased which reduced the applied force on the seat belt until the manikin came to rest close to upright position. The force-time method applied a constant force on the seat belt and hence predicted lesser time to reposition.
The coupling offered the advantage of using the best modeling interfaces of each solver and averted the need to translate the models between software environments. This accelerates the development process and reduces the re-engineering work. However, a disadvantage is that two additional simulation tools, other than LS-Dyna, are required.

5 Conclusion

The feedback control loop method outperforms the force-time characteristic method as evidenced by CORA ratings. Therefore, feedback control loop modelling could be preferred for designing and evaluating repositioning functionalities of seat belt systems.

Human Body Model Positioning using Oasys PRIMER

Galal Mohamed, Gavin Newlands

Arup

Abstract

The LS-DYNA pre-processer Oasys PRIMER provides a comprehensive solution for occupant safety applications using LS-DYNA. This includes creating seat mechanisms which allows the seat to be moved into different positions. There are also tools for the creation and fitting of seatbelts, including fully fabric belts through explicitly meshed sliprings. PRIMER also contains functionality to set-up simulation-based occupant positioning and seat deformation, either as a single-stage or multi-stage analysis.

Oasys PRIMER's positioning capabilities have been recently extended to human body models (HBMs), which are detailed biofidelic finite element models of the human body. The positioning and seatbelt tools allow users to interactively achieve realistic articulation and positioning of HBMs to prepare them for simulation-based model positioning in LS-DYNA, covering a wide range of positions and postures.

Body positioning of HBMs involves using a positioning tree, which is a collection of ordinary LS-DYNA entities such as nodes, elements, materials etc. but with extra PRIMER-specific data to define body parts relationships. This metadata includes realistic stop angles to ensure appropriate relative movement between the various parts of the model, and representative biofidelic joint detailing, for example, bending and straightening of the spine, and realistic shoulder/clavicle movement. Once a target position has been defined, PRIMER automatically creates displacement-based cables to pull the HBM into position during an LS-DYNA analysis.

Using industry-leading HBMs, this paper describes the modelling workflow to position an occupant HBM in a front-sled model subject to a frontal impact load case.

Real Time Biofidelic positioning of Human Models with ANSA

Lambros Rorris[1], Athanasios Lioras[2]

[1]BETA CAE Systems AG
[2]BETA CAE Systems SA

Abstract

The advent of autonomous driving with its many out-of-position load cases, makes the use of HBMs in safety simulations a necessity. Moreover, safety simulations for other vulnerable road users such as, pedestrians and cyclists are increasingly needed, and HBMs can address this need too.

Positioning of an HBM though, has always been a challenge. BETA CAE Systems has always taken up the challenge to industrialize advanced methods and produce tools that push simulation technology to its next steps.

ANSA, in its latest version offers a novel solution to this complex problem, making the positioning and handling of an HBM, as easy as with an ATD model. Using an advanced integrated MBD solver in parallel with morphing algorithms, engineers are provided with real time articulation and positioning of a HBM within an easy user interface. While the user just articulates the human model with the mouse in a most direct way, the biofidelic joint modelling guarantees realistic model movements and the generation of a ready-to-run model without the need of pre-simulation. Of course, in case the user wishes to run a re-simulation this can also automatically be set up. Furthermore, all tools and procedures available for ATDs (restraints, coupled dummy-seat movement etc.) are also available for the HBM models. Thus, the positioning of HBMs and ATDs are treated in the same manner within the pre-processor offering the liberty to the engineer to perform the analysis he wishes. In this presentations, all these exciting developments and future plans are demonstrated.

A critique of the THUMS lower limb model for pedestrian impact applications

Thomas Cloake[1], Christophe Bastien[1], Joseph Hardwicke[2], Demetrios Venetsanos[1], Clive Neal-Sturgess[1]

[1]Institue for Future Transport and Cities, Coventry University, UK
[2]University Hospitals Coventry and Warwickshire, Coventry, UK

Abstract

The Total Human Model for Safety (THUMS) is widely used for biomechanics research and validated at the component and full-body levels. Nonetheless, some authors have reported differences in predictions between the model and real-life injuries, particularly in the lower limbs. This study aims to perform an extensive critique of the THUMS lower limb and identify areas for improvement. The THUMS model was assessed across quasi-static and dynamic validation tests to understand geometry, material properties and response to impact. The study has highlighted that the THUMS' geometry is comparable to published cadaveric data for bones and ligaments, but soft tissues (muscle, adipose and skin) and fascia have significant simplifications. The bones' material properties are evidence-based and vary appropriately according to anatomical site. Bone failure is permitted through element deletion; however, the unusually transverse fracture pattern predicted in THUMS is seldom seen in clinical practice. The simplified soft tissue model cannot fail, making it unable to replicate the extensive damage seen in high energy open fractures. Ligament injury is a frequent result of an impact to the pedestrian lower limb, often at the bone-tendon interface, yet the failure location seen in the THUMS model is mid-substance. In summary, THUMS makes an excellent attempt to model the lower limb; nonetheless, some work is still required to increase biofidelity. Improvements in soft tissue geometry and material properties and fracture pattern modelling represent apparent areas for development.

Definition of Peak Virtual Power Brain Trauma Variables for the use in the JSOL THUMS injury post-processor web-based estimator

C. Bastien[1], C. Neal-Sturgess[2], H. Davies[1], X. Cheng[1]

[1]Coventry University, Institute for Future Transports and Cities, Coventry, UK
[2]University of Birmingham, Department of Mechanical Engineering, Birmingham, UK

Road traffic accidents and falls are catastrophic events leading to serious injury and in some cases fatality. The dichotomy is that traumatic injuries are assessed using the Abbreviated Injury Scale (AIS), which is a measurement of the probability of death, whilst the engineering tools available to support the understanding of injury causation rely on engineering measurements of stress and strain. Further to this, the problem of ageing is not adequately dealt with using existing engineering tools. The research proposes the development of a generic mathematical injury severity model, based on Peak Virtual Power (PVP), which is using the Clausius-Duhem inequality from the rate dependent form of the 2nd Law of thermodynamics, relating to degeneration and decay, to establish relationships between AIS, age and collision speed. The paper has shown that PVP was generic (global), objective (theoretical), not subjective (empirical), dimensionally meaningful (not as HIC), unique (peak values are unique) and could predict soft tissue injury severity, illustrated in this research with the case of an occipital fall. This method, newly implemented JSOL THUMS injury post-processor web-based estimator, has the ability to calculate all AIS levels of any finite element model's white and grey matter, which are defined as a polynomial function. This paper explains the underpinning of the PVP theory, as well as provide the coefficients to calculate brain injury severity under blunt trauma impact of a THUMS4.01 head model in which material properties were modified to reflect THUMS 4.02's latest material brain enhancements.

On the Prediction of Process-Dependent Material Properties for Ti-6Al-4V with LS-DYNA

T. Klöppel (DYNAmore)

Simulating Resistive Heating of Ti-6Al-4V

M. Merten (DYNAmore)

FE analysis and parameter optimisation of anisotropic material models for metals using Full-Field Calibration

Johann Jung[1], Serkan Mouchtar[1], Julian Schlosser[1], Michael Schmiedt[1], Wolfgang Rimkus[1],
Hochschule Aalen

[1] Hochschule Aalen

Abstract

Forming simulation models and the associated material characterisation are important factors when representing the increasingly complex deep drawing operations. Especially in context of automotive components, the finite element method ensures producibility prior to pilot series and minimises the risk of wasting resources by mapping the plastic and anisotropic behaviour of the material. In order to identify material parameters and to calibrate complex material models, digital image correlation can be used. This method enables recording the three-dimensional deformation of components under different load conditions so that the entire strain fields can be evaluated and verified using FE analysis. Based on the experimental strain fields, material parameters in the simulation model can be optimised to match the component deformations both locally and globally. The calibrated material model can be used to simulate complex components and predict the close-to-reality component behaviour during forming. This paper demonstrates the calibration of an anisotropic material model for sheet material using Full-Field Calibration and the Hockett & Sherby yield curve rule as well as Lankford coefficients. The results of the Full-Field Calibration shows a high correspondence between the predicted strain field and the experimental data since the deviation of the true strain values is less than 3 %. Furthermore, different optimisation approaches and settings in LS-OPT were investigated and the formation of metamodels with Linear Polynomial, Fastforward Neural Network and Radial Basis Neural Network algorithms were compared with focus on accuracy and computation time. The Fastforward Neural Network algorithm achieves the best results in terms of precision, but this algorithm is slightly slower in terms of optimisation time compared to the Linear Polynomial algorithm.

Automation of LS-DYNA's Material Model Driver for Generation of Training Data for Machine Learning based Material Models

Daniel Sommer, Tarun Kumar Mitruka Vinod Kumar Mitruka, Peter Middendorf

Institute of Aircraft Design, University of Stuttgart
Pfaffenwaldring 31, 70569 Stuttgart, Germany

daniel.sommer@ifb.uni-stuttgart.de
mitruka.tarun@gmail.com
peter.middendorf@ifb.uni-stuttgart.de

1 Abstract

The substitution of classical constitutive material models with data-driven models supported by machine learning techniques could provide a leap in the modelling of materials. The most notable benefits are a faster description of new materials without a tedious manual parameter identification procedure, lower computational time for simulations due to efficient computation within the material model and a more efficient selection of the correct material model for the use-case. The base for any data-driven model is adequate amount and quality of training data. Based on this, machine learning techniques can be used to train neural networks such that they learn the relationship between given input and output. The mapping in the machine learning based material model will be the strain measures to the stresses, similar to classical models. In order to learn the stress-strain relationship for materials, training data is generated from existing material models in LS-DYNA, as the direct extraction from materials testing is not possible due to the impossibility of local stress measurements. To generate training data, an existing model could be implemented in own code, single-element simulations could be performed, or even data from component simulations could be extracted.

This study shows a way to generate training data for almost any material model available in LS-DYNA by using and automating the integrated Interactive Material Model Driver. This enables access to the unbiased stress response of the black-box material models, by providing an arbitrary progression of components of the displacement gradient in time as input, to be evaluated at a single integration point. Automation is added on top of the embedded Material Model Driver regarding the generation of strain paths for 2D and 3D cases. It is followed by automatic execution, data extraction and preparation of data for machine learning. Advantages and disadvantages over other strategies for training data generation will be highlighted, compared and finally an exemplary material model using neural networks trained on data generated with the Automated Material Model Driver is shown. The goal is that networks learnt through such a training data can also be used as a surrogate to develop any new material model.

Implementation of Deep Learning-Based Models for the Prediction of Elasto-Plastic Material Behavior

M. Burkhardt, D. Wiesner, Prof. U. Göhner (Univ. of Applied Sciences Kempten)

For the modeling of elasto-plastic material behavior, the development of the strains up to the current point in time must be taken into account. For the computational architectures of the machine-learning approaches, this means that correspondingly flexible input lengths must be taken into account. For the three basic architectures investigated, MLP, RNN, CNN, different approaches are necessary. The recurrent neural networks already fulfill the requirement of the material model due to their architecture. The transition from state to state, as well as the parameter sharing, helps. Simple MLPs and CNNs, on the other hand, required additional methods and approaches. For example, in the case of CNNs, global pooling or padding after the convolution layer can satisfy the requirements for flexible input sizes. In addition, different approaches to data injection can overcome the challenges of predicting elasto-plastic material behavior for MLPs and CNNs. Ultimately, a combination of CNNs and RNNs is also possible.

For the implementation and evaluation of the different methods, the Keras API is used in a Python environment. A test data set, unknown to the network, is used for evaluation. The average error over the test dataset shows the accuracy of the predictions of the different methods. In addition, a random sequence is plotted from the test data set with the corresponding predictions of the methods. The figures give further indications of the suitability. At the same time, the decision on suitability is not trivial. Architecture-specific factors, such as the hyperparameters, are relevant. Different learning times of the methods are another factor to be considered. In addition, there are specific properties of the data set. A classification of individual stress responses is conceivable.

Considering all factors, the results can be summarized as follows: The simple MLP architecture was unexpectedly able to provide good predictions, RNNs have achieved very good results with the use of different cell architectures, in the case of CNNs not all approaches achieve satisfactory results, but nevertheless some approaches could be suitable here. The CRNN architecture should also be highlighted as a promising approach.

The aim of this work is the investigation and implementation of suitable methods for the prediction of elasto-plastic material behavior. The results show that deep learning based methods are suitable for these predictions.

In the area of the finite element method, the performance (computing time, required memory) of the deep learning methods should also be compared with each other. Whether deep-learning based methods are suitable to replace existing models, however, must be shown by further investigations.

Isogeometric Analysis in LS-DYNA R13
–
key steps towards industrial applications

Stefan Hartmann[1], Lukas Leidinger[1], Dave Benson[2], Attila Nagy[2], Marco Pigazzini[2], Liping Li[2], Lam Nguyen[2]

[1]DYNAmore GmbH
Industriestr. 2, 70565 Stuttgart, Germany
e-mail: stefan.hartmann@dynamore.de

[2]ANSYS LST
7374 Las Positas Road, Livermore, CA 95411, USA

Abstract

The term "Isogeometric Analysis" (IGA) was first introduced by Hughes et al. [1] in 2005. Since then, numerous research papers have been devoted to IGA which have demonstrated beneficial analysis properties compared to standard finite element analysis (FEA). IGA is a finite element method that invokes the geometry descriptions used in computer-aided design (CAD) to perform numerical analysis. B-splines and non-uniform rational B-Splines (NURBS) are amongst the most widely used geometry descriptions in CAD and thus, NURBS-based finite elements have been developed and implemented into LS-DYNA over the recent years.
In this paper the analysis possibilities using IGA elements in LS-DYNA are introduced. Hereby the focus is on some major and crucial developments added to LS-DYNA in R13, which now allows the finite element analysis of large-scale industrial models of interest.
The following topics will be discussed in this paper:
- from boundary representation (B-Rep) in CAD to a suitable multi-patch analysis
  - the new **`*IGA_xxx`** keyword family
  - mechanical coupling of topologically connected trimmed patches [2], [3]
  - mechanical coupling of T-joints
- stabilization of heavily trimmed IGA-shell elements
- the idea of feature-based modeling
  - application of boundary conditions
  - spotweld modelling using **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** (former **`*X_SPR3`**)
  - definition of rigid bodies (**`*CONSTRAINED_NODAL_RIGID_BODY`**) linked to IGA
- hybrid analysis: using IGA together with standard FEA in the same model

Various examples will demonstrate the above mentioned enhancements. The paper closes with a conclusion about the current status of IGA in LS-DYNA R13 and presents an outlook and vision about future possibilities and trends around IGA in LS-DYNA.

Literature

[1] Hughes, T.J.R., Cottrell, J.A., Bazilevs, Y.: "Isogeometric Analysis: CAD, finite elements, NURBS, exact geometry, and mesh refinement", Computer Methods in Applied Mechanics and Engineering, Vol. 194, 2005, 4135-4195.

[2] Breitenberger, M., Apostolatos, A., Philipp, B., Wüchner, R., Bletzinger, K.-U.: "Analysis in computer aided design: Nonlinear isogeometric B-Rep analysis of shell structures", Computer Methods in Applied Mechanics and Engineering, Vol. 284, 2015, 401-457.

[3] Leidinger, L.F., Breitenberger, M., Bauer, A.M., Hartmann, S., Wüchner, R., Bletzinger, K.-U., Duddeck, F., Song, L.: "Explicit Dynamic Isogeometric B-Rep Analysis of Penalty-Coupled Trimmed NURBS Shells", Computer Methods in Applied Mechanics and Engineering, Vol. 351, 2019, 891–927.

Latest advancements for IGA model creation with ANSA

Lambros Rorris[1], Ioannis Chalkidis[2]

[1]BETA CAE Systems AG
[2]BETA CAE Systems SA

Abstract

Isogeometric Analysis (IGA) is maturing and becoming a promising alternative to FE for crash industrial applications. During the last years the capabilities of both LS-DYNA and ANSA, in the creation and analysis of IGA crash models, have been under continuous development.

The main focus has been the efficient generation of trimmed NURBS-based multi-patch surfaces that can represent parts of shell structures for the analysis in LS-DYNA. Nevertheless, the IGA community is always pushing the boundaries of the technology, so pre-processing support has been added for all IGA descriptions, both trimmed and untrimmed. Meanwhile co-operations with numerous universities and research groups makes sure that the latest technology advancements are swiftly available to engineers.

The basis for the communication between ANSA and LS-DYNA is the new, *IGA family of keywords, including topology information as in B-Rep CAD models. Along with that, comes support for better description of Boundary and Initial Conditions and Constraints directly on geometry entities, making modelling more straightforward and increasing transparency from CAD to Analysis.

Continuous integration with existing FE models and easy creation of Hybrid FE – IGA models has been one of the main goals. Thus, considerable effort has been put, so that both technologies are compatible within the pre-processor to ensure that IGA parts can be used interchangeably throughout the entire process from CAD translation to final model assembly.

This commitment to the technology has proved fruitful during the last two years and many examples of successful application cases already exist.

Hybrid IGA/FEA Vehicle Crash Simulations with Trimmed NURBS-based Shells in LS-DYNA

Lukas Leidinger[1], Stefan Hartmann[1], Dave Benson[2], Attila Nagy[2], Lambros Rorris[3], Ioannis Chalkidis[3], Frank Bauer[4]

[1]DYNAmore GmbH
Industriestr. 2, 70565 Stuttgart, Germany

[2]ANSYS LST
7374 Las Positas Road, Livermore, CA 95411, USA

[3]BETA CAE Systems International AG
Platz 4, 6039 Root D4, Switzerland

[4]BMW Group
Knorrstraße 147, 80788 Munich, Germany

Abstract

Isogeometric Analysis (IGA) [1] is a rather new approach to Finite Element Analysis (FEA), using spline basis functions known from Computer Aided Design (CAD) for describing both the geometry and the solution field. The main motivation for IGA is the integration of design and analysis. Achieving such a full integration requires a holistic approach with a fundamentally different modeling strategy and development process to exploit the full potential of IGA. Such changes certainly take time and cannot be achieved overnight. Fortunately, IGA with its higher-order and higher-continuity elements also offers several additional advantages such as an accurate geometry description, superior analysis qualities, a larger explicit time step size or smart modeling techniques. Thus, users may benefit from IGA immediately, even without a full paradigm shift.

Several recent developments made the application of IGA in an industrial context more attractive for users. First, the introduction of the new **`*IGA`** keywords in LS-DYNA R13, which are able to capture the data structure of common CAD models, so-called B-Rep (boundary representation) models, including geometry and topology information. This comes along with the capability to couple and stabilize the multiple trimmed NURBS (Non-Uniform Rational B-Splines) shell patches appearing in industrial models during analysis. Second, the increasing availability of preprocessing capabilities for IGA, especially in ANSA [2]. From version 22, ANSA enables an efficient generation of industrial NURBS shell models for LS-DYNA, based on the new **`*IGA`** keywords. A third reason for the attractiveness of IGA in LS-DYNA is the possibility to simulate hybrid models, that is, models consisting of both, IGA and conventional FEA components.

The focus of this paper is on hybrid IGA/FEA models with isogeometric shell components. In particular, Section 2 describes the generation of isogeometric NURBS shell models from current (not analysis-suitable) CAD models. Section 3 demonstrates the possibility to insert IGA components into an existing FEA (vehicle) model via a simple one-by-one component exchange. It furthermore shows that connection technology like spotwelds, bolts and rigid bodies can now be directly applied to isogeometric shells in LS-DYNA, without further model modifications. Section 4 provides two industrial crash examples of BMW models, dynamic buckling of a crashbox-type component and impact of a hybrid IGA/FEA vehicle front end structure against a rigid wall. Section 5 gives a brief summary and an outlook to future steps and developments.

Isogeometric Analysis on Trimmed Solids: A B-Spline-Based Approach Focusing on Explicit Dynamics

Manuel Meßmer[1], Lukas Leidinger[2], Stefan Hartmann[2], Frank Bauer[3], Fabian Duddeck[4], Roland Wüchner[1], Kai-Uwe Bletzinger[1]

[1]Technical University of Munich, Chair of Structural Analysis
Arcisstr. 21, D-80333 München, Germany
e-mail: manuel.messmer@tum.de, web page: www.st.bgu.tum.de

[2]DYNAmore GmbH, Industriestr. 2, D-70565 Stuttgart, Germany

[3]BMW Group, Knorrstr. 147, D-80788 Munich, Germany

[4]Technical University of Munich, Associate Professorship of Computational Mechanics,
Arcisstr. 21, D-80333 München, Germany

Engineering workflows are habitually split into a modelling phase and a consecutive analysis phase, which is primarily driven by the finite element method (FEM). However, bridging the gap between design and analysis remains a sophisticated problem and may consume a vast amount of computational as well as manual operations, especially in highly iterative development processes. To avoid this major bottleneck, isogeometric analysis (IGA) [1] and later isogeometric B-Rep analysis [2] were developed. They rely on the mathematical descriptions of computer aided design (CAD), such as NURBS- and B-Spline-based B-Rep models. Nevertheless, classical B-Rep formulations describe a solid only by its boundary faces and do neither provide any physical nor geometrical description of the interior. Therefore, the IGA concept cannot be directly applied to solid structures.

To overcome this issue and to facilitate the modelling with solid-based formulations, a design-through-analysis workflow is presented which avoids demanding boundary-fitted meshes. In a preprocessing step, the geometry is retrieved from the CAD system as a stereolithography (STL) representation and embedded into the parameter space of a regular grid composed out of trivariate B-Splines. The STL description defines the geometrical borders to construct efficient integration rules and additionally provides an intrinsic parametrization for contact formulations and visualization purposes. We employ $\boldsymbol{C^{p-1}}$ continuity across all knot spans to achieve practically feasible critical time steps even for arbitrarily small trimmed elements. The obtained results show similar behaviour of the critical time step as observed in explicit shell element-based IBRA [3] and thus demonstrate the same gain in efficiency regarding explicit time integration. We illustrate the effectiveness and applicability of the approach through explicit dynamic simulations of a BMW component including large deformations and contact. Finally, we discuss encountered open research questions and suggest possible next steps.

Literature

[1] Hughes, T. J.R. and Cottrell, J. A. and Bazilevs, Y.: "Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement", Computer Methods in Applied Mechanics and Engineering 194, 2005, 4135-4195.

[2] Breitenberger, M. and Apostolatos, A. and Philipp, B. and Wüchner, R. and Bletzinger, K. U.: "Analysis in computer aided design: Nonlinear isogeometric B-Rep analysis of shell structures", Computer Methods in Applied Mechanics and Engineering 284, 2015, 401-457.

[3] Leidinger L.: “Explicit Isogeometric B-Rep Analysis for nonlinear Crash Simulations”, Dissertation, Technical University of Munich, 2020.

Prediction of fatigue damage by random vibration using isogeometric and finite element analysis

Shubiao WANG[1], Renata_Troian[1], Leila Khalij [1]

[1]INSA Rouen Normandy, France

At present, the Finite Element Analysis (FEA) method is indispensable in the field of simulation technology, as this kind of numerical analysis method can assist engineers to predict results, which are often difficult to obtain from experimental tests. However, there exist some problems in terms of finite element mesh generation time and geometric representation. In this studying, we adopted a new numerical analysis method, Isogeometric Analysis (IGA) to develop static and dynamic analyses on two models, a notched plate and a wind turbine tower model in Ls Dyna software. From the static convergence analysis result, it is shown that IGA is more time-efficient compared with FEA. In terms of fatigue analysis results, IGA can predict the fatigue life corresponding very well to the fatigue life computed by FEA. It can be concluded that IGA is appropriate for the numerical analysis.

Keywords: Isogeometric analysis; Finite element method; random vibration fatigue analysis

Adaptation of a solid self-piercing rivet made of aluminum using numerical simulation to extend the application limits

Maximilian Schlicht[1], Dr.-Ing. Thomas Nehls[1], Pascal Froitzheim[1], Prof. Dr.-Ing. Wilko Flügge[1]

[1]Fraunhofer Institute for Large Structures in Production Engineering IGP, Rostock, Germany

Increasing resource efficiency, for example through the consistent application and further development of lightweight construction concepts, plays an important role in the development of the mobility sector. This requires a steadily increasing use of high-strength aluminum alloys in primary vehicle structures. A suitable and efficient process for joining high-strength aluminum alloys is solid self-piercing riveting (SSPR). A major advantage of this process is the elimination of time-consuming preparatory work such as pre-drilling, deburring and positioning of the components to be joined, as the rivet punches through these during the installation process. Due to the high stress on the rivet during the process and the lack of knowledge on the use of ultra-high-strength aluminum alloys as the rivet material, solid self-piercing rivets (SSP-rivets) made of steel are generally used. However, against the background of recyclability, thermal expansion and corrosion protection, the use of aluminum SSPR would be desirable.
To meet the challenges of the joining task, the SSPR can be adapted in terms of material and geometry. In this case, the tool of simulation is very well suited for testing a large number of possible combinations during geometry adaptation and for choosing preferred variants. When mapping the experimental tests and the joining process in the finite element software LS-DYNA®, it is of great importance to characterize the material behavior and failure in appropriate accuracy, in order to be able to physically correctly map the partial aspect of punching in particular. The GISSMO failure model, which is used in this work, is suitable for this purpose. To adapt the groove geometry of the SSP-rivet, simplified punching tests with different groove geometries are carried out first. The grooves are exposed to high shear forces during the stamping process, which they must withstand without undergoing significant deformation. The preferred variant of the preliminary investigations is then transferred to two SSP-rivets and the rivet concept is finally validated.
As a result, the simulation is able to highlight the suitability of different geometry variants and to correctly represent trends in the results of the joining process. This has made it possible to extend the application limits of an aluminum SSP-rivet with regard to the maximum total sheet thickness as well as the maximum strength of the materials that can be joined.

Forming and spring-back simulation of CF-PEEK tape preforms

S. Cassola, M. Duhovic, L. Münch, D. Schommer, J. Weber, J. Schlimbach, J. Hausmann

Leibniz-Institut für Verbundwerkstoffe GmbH, Technische Universität Kaiserslautern, Erwin-Schrödinger-Straße 58, 67663 Kaiserslautern, Germany

Abstract

A virtual process chain has been developed for the thermoforming of customized CF-PEEK tape-preforms into the final geometry of an aircraft fuselage integral frame. The goal of the simulation was to be able to predict the fiber orientation during forming along with the part warpage/distortion (spring-back/spring-in). During thermoforming, the CF-PEEK tape-preform is subjected to combined bending and shear stresses at elevated temperatures of up to 400°C. For the creation of the material model, it is therefore necessary to determine the temperature-dependent bending and shear behavior of the preform using material characterization tests. For this purpose, the bending behavior of 2 mm thick, 0° UD specimens (unidirectional tailored preform laminates) were investigated in both fully and partially consolidated forms using 3-point bending tests at temperatures of 360°C, 380°C and 400°C. In shear frame tests, the shear behavior of 0°/90° specimens laminates in fully and partially consolidated forms at elevated temperatures was also determined. The measured force-displacement and force-shear angle curves were then reproduced in simulations of the characterization tests to obtain calibrated material parameters for the forming simulations.

The material model developed for the simulation is based on a combined element unit-cell approach. To set-up the simulation, a layer of shell elements is first created based on the preform geometry and the radii and flanges of the mesh are refined based on the chosen mesh size. The beam elements and a second layer of shell elements, which provides the thermal contact, are generated in LS-DYNA® using the machine paths of the tape-laying robot. Together, the shell and beam elements form a single layer of the complete preform laminate, represented by a stack of several layers. According to the preform structure, the layers are linked together by means of contact definitions. The preform holding clamping-frame is realized in the simulation via beam elements. The complete forming simulation model of the demonstrator part contains approximately 1.2 million elements (see Fig. 1).

To carry out a warpage and/ simulation, the final state of the forming simulation is used as the basis for input. In addition to the fiber orientations, stresses and strains were also taken directly from the final state of the forming simulation model. The simulation model predicted a spring-back behavior of the part in-line with that observed in thermoforming experiments.

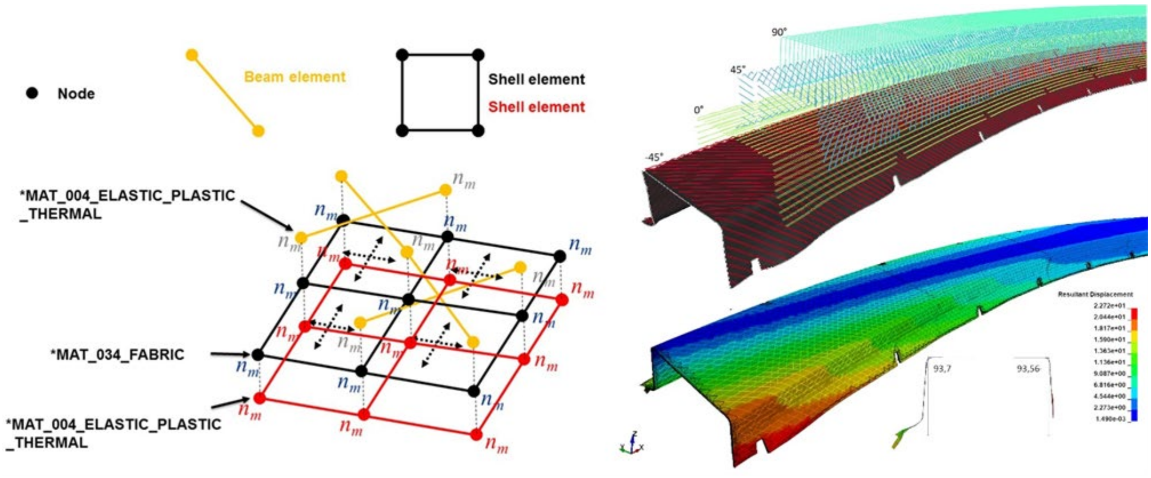


Fig.1: Process simulation of aircraft fuselage integral frame manufacturing: Schematic representation of the model's unit-cell (left) and exploded view of the resultant fiber orientations after the thermoforming simulation and the resultant warpage/distortion of the demonstrator part following the spring-in simulation.

Dynaform 6.2
New Features and Enhancements

J. Chen (eta)

Introducing Quickform Solver - A Fast Enhanced Incremental Solver for Early Die Face Development Phase

J. Chen (eta)

Simulation of the high velocity impact of railway ballast on thermoplastic train underbody structures

Mathieu Vinot[1], Dorothea Schlie[1], Tobias Behling[1], Martin Holzapfel[1]

[1] Institute of Structures and Design, German Aerospace Center, Stuttgart

1 Abstract

Keywords: high velocity impact, railway ballast, composite simulation, thermoplastics, DEM method

Railway transportation represents an environmentally friendly alternative to automotive transportation for long distance travel. In the project Next Generation Train of the DLR, new railway solutions are developed for passenger and freight transportation for a broad range of applications (intercity, cargo, long distance…). Specifically, the high-speed train NGT HST aims at reducing travel times and specific energy consumption with new technologies. At the maximal operating speed of 400 km/h, the coupling of mechanical and aerodynamical forces leads to increasing risks of ballast stone impact on the train structures (in particular underbody structures), thus threatening primary components underneath [1]. Through the repetition of stone impacts during the entire lifetime of a structure, critical damage can occur and reparation or replacement concepts are required. The present work aims at investigating an impact-resistant underbody structure made out thermoplastic composite materials for the HST train on numerical and experimental basis. By considering multiple impact scenario in the structure sizing process, the project intends to reduce the interval at which the structure has to be replaced and to lower security factors.

Starting with two combinations of thermoplastics and preforms, an experimental test campaign is performed on specimen scale to investigate their impact resistance at low velocities. The most promising material is then tested on component level under high-velocity impact at the gas gun DLR facility. Tests on single ballast stones are performed as well for validation of the numerical methodology.

As support to time and material expensive test methods, a numerical framework is developed within LS-DYNA at all levels from low to high impact velocities. The composite parts are automatically modelled with stacked TSHELL and cohesive-element interfaces via *PART_STACKED_ELEMENT. Composite materials and interface use the state-of-the-art material models *MAT_262 and *MAT_240 respectively. Random ballast stones geometries are automatically generated with the Voronoi tessellation and simulated with DEM particles (Figure 1).

The present paper illustrates key aspect of the modelling techniques and their influence on the numerical impact behavior. Furthermore, achieved results along the test pyramid are presented.

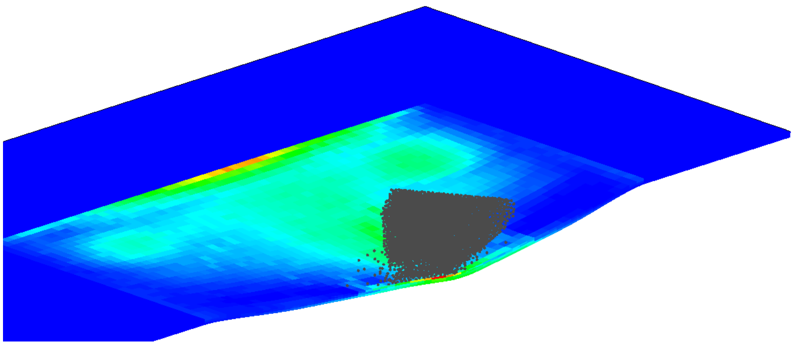


Figure 1: Simulated impact of a DEM-stone on a composite plate

[1] U.S. Department of Transportation, Identification of High-Speed Rail Ballast Flight Risk Factors and Risk Mitigation Strategies – Final Report, 2015, Washington DC.

Numerical Analysis of Impact Tests on Bending Failure of Reinforced Concrete Slabs Subjected to Inclined Soft Missile Impact

Christian Heckötter, Jürgen Sievers

Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Cologne, Germany

1 Introduction

Impact loading is a safety relevant loading case for reinforced concrete structures used to protect vital parts of nuclear facilities. Numerical methods used for the assessment are validated on the basis of impact tests. Even though normal impacts are the most common item of analysis, there is a certain interest to study scenarios with inclined impacts. It is for instance not obvious, that normal impact is the most unfavorable loading case regarding floor response spectra. Further, protective structures may prevent normal impact at critical locations, which may however be subjected to inclined impact.

Effects of inclined impact include slipping and rotation of the missile, motion of the impact point and effects of tangential forces. Recently, an experimental program dealing with bending failure of reinforced concrete slabs subjected to inclined soft missile impact was carried out at Technical Research Centre of Finland (VTT) in the frame of phase IV of the international research project IMPACT. This paper reports on simulation results on these tests using LS-DYNA.

2 Numerical model

The numerical modelling (Fig. 1) based on explicit missile target interaction is capable to reproduce the sliding of the missile above a critical angle of inclination and the utilized concrete models (**`*MAT_WINFRITH`** and **`*MAT_RHT`**) are well suited to simulate the maximum deflections of the slab at several sensor locations. Regarding post impact vibration frequencies some differences occurred for **`*MAT_RHT`**. Concerning strain measurement on reinforcement steel differences occurred at locations of some strain gauges. Simulated spatial strain distribution tends to be more localized than the measured one.

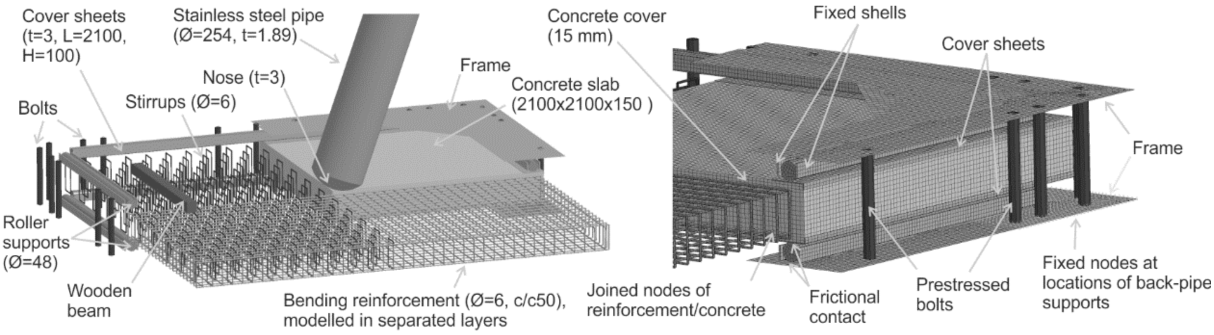


Fig.1: Details of the LS-DYNA model used for inclined soft impact simulations (dimensions in mm).

3 Acknowledgement

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

Hourglass reduction measures in hard turbine missile impact into concrete protective barrier

Milko Miloshev[1], Alexander Iliev[2]

[1]Mott MacDonald, Principal Structural Engineer
[2]Mott MacDonald, Senior Structural Engineer

1 Abstract

Failure of rotating machinery in nuclear power stations may result in ejection of fragments with high kinetic energy. Pieces of the disk or blades of a damaged turbine may cause failure of surrounding systems, structures, and components. The current paper presents study of effect of turbine disk fragment ejected as hard missile on the capacity of the protective walls of safety-related nuclear building. The investigation is performed by the missile-target interaction method, i.e. impact simulation of the model of the missile (i.e. the disk fragment) into the model of the protective barrier. Detailed model of the target is generated utilising non-linear material models for rebar steel and concrete. Different scenarios are investigated to define the most unfavourable impact with respect to the protective capacity of the safety barrier. The acceptance criteria are adopted form the relevant international regulatory documents. Special emphasis is given to the hourglass reduction. Various options are studied such as the hourglass control keywords, mesh refinement and different element formulation. The outcome in terms of CPU time and fulfilment of the structural acceptance criteria is compared and conclusions are drawn.

2 Literature

[1] LSTC, LS-Dyna Keyword User's Manual Volume I, 2018
[2] I. Klecherov, M. Deyanova, A. Andonov, Turbine Missile Impact Assessment- Comparison of Results from Empirical Formular & Finite Element Analysis.

Cardiac Electrophysiology using LS-DYNA®

Pierre L'Eplattenier[1], Inaki Caldichoury[1], Karim El Houari[2]

[1] Ansys - Livermore Software Technology LLC
[2] Ansys Immeuble le Patio, 35-37 Rue Louis Guérin, 69100 Villeurbanne, France

1 Abstract

Heart disease is among the leading causes of death in the Western world; hence, a deeper understanding of cardiac functioning will provide important insights for engineers and clinicians in treating cardiac pathologies. However, the heart also offers a significant set of unique challenges due to its extraordinary complexity. In this respect, some recent efforts have been made to be able to model the multiphysics of the heart using LS-DYNA.

The model starts with electrophysiology (EP) which simulates the propagation of the cell transmembrane potential in the heart. This electrical potential triggers the onset of cardiac muscle contraction, which then results in the pumping of the blood to the various organs in the body. The EP/mechanical model can be coupled with a Fluid Structure Interaction (FSI) model to study the clinically relevant blood flow parameters as well as valves or cardiac devices. This paper concentrates on the EP part of the model.

Different propagation models, called "mono-domain" or "bi-domain", which couple the diffusion of the potential along the walls of the heart with ionic equations describing the exchanges between the inner and the outer parts of the cells have been implemented.
Other features of the EP solver will also be presented such as the coupling of the mono/bi domain models with a Purkinje Network, the automatic generation of fiber orientations, the computation of EKG, and the coupling of the EP with the mechanics and FSI.

Introduction of ISPG Method and Geometric Multiscale Modeling for Electronics Solder Reflow and Shock Wave Analysis

Dandan Lyu, Wei Hu, Xiaofei Pan, C. T. Wu

Computational and Multiscale Mechanics Group, Livermore Software Technology, An Ansys Company, Livermore, USA

Abstract

Solder joints have become the main mechanical and electrical connections in modern microelectronics packaging for most consumer electronics products and they are typically observed to be the weakest links in terms for structural strength in the drop shock event. A drop shock simulation involves modeling the shock wave effect on mesoscale solder joints and macroscale chip packages concurrently, which is a typical multi-scale problem. Conventional finite element approaches using beam elements for the representation of the solders and the one-way sub-modeling technique cannot offer a high-fidelity solution. In addition, the shape of the solder ball is a very important contributory factor in determining the local stress levels and it is impractical to obtain all solder ball geometries by experimental measurement. Therefore, an effective simulation tool for the prediction of solder ball shape in the solider joint design as well as for the drop shock analysis is required in electronics industry.

In this presentation, a multiscale computational approach for linking the information of mesoscale dissimilar solder ball geometries to the macroscale drop shock of a printed circuit board (PCB) [1] is developed. To begin, a full-implicit incompressible smoothed particle Galerkin (ISPG) [2] method is introduced to model the free-surface solder reflow process and predict the solder ball shapes. The formulation considers the surface tension of molten solder and the wall adhesion between the solder and the substrate. Subsequently, the predicted solder ball shapes from the reflow analysis are used in a chip package model for the drop shock analysis. The mesoscale solder joint model is coupled concurrently with the macroscale chip package model using an explicit-explicit non-intrusive two-scale coupling method via the co-simulation technique. An algorithm that handles properly the load-balancing, heterogeneity of processors and memory also has been developed to achieve the practical non-intrusive scale-bridging effect.

Numerical examples firstly demonstrate that the proposed implicit ISPG formation is able to accurately and efficiently predict the solder reflow profile. Then a multiscale drop shock example utilizing the reflowed solder ball shapes for a chip-scale package is performed, which indicates that the present approach is capable of delivering an accurate and efficient solution by comparing to the (direct numerical simulation) DNS results.

Literature

[1] Hu, W., Pan X., Lyu D., Srivastava A., Shah S., Wu, C.T.: "A Two-scale approach for the drop shock simulation for a printed circuit board package considering reflowed solder ball geometries", International Journal for Multiscale Computational Engineering, 18(4), 2020, 455-476.

[2] Pan, X., Wu, C. T. and Hu, W.: "A semi-implicit stabilized particle Galerkin method for incompressible free surface simulations", International Journal for Numerical Methods in Enginering, 121(17), 2020, 3979-4002.

Comparative Evaluation of Sound Absorption Performance of Various Types of Core Panels

Sunao Tokura[1]

[1]Tokura Simulation Research Corporation (TSR)

1 Introduction

Core materials such as honeycomb panels are used in various industrial products as lightweight and high stiffness materials. As one of the characteristics of the core panel, the sound absorption/insulation effect due to the core structure is expected. By using a large core panel for a soundproof wall such as a highway, the possibility of producing a soundproof wall that is lighter than the conventional soundproof wall and has a high sound insulation/absorption effect is being studied. In addition, with the electrification of automobiles, further quietness in the passenger compartment is required, and the use of core panels is being considered for improving the quietness of various transportation machines such as automobiles. Therefore, in this research, we assume several types of core shapes and compare the sound absorption/insulation effect of the core panel by modeling the interaction between the core panel and sound using ALE-based FEM in LS-DYNA®.

2 Analysis model

Figure 1 shows an example of an acoustic analysis model. A diaphragm that serves as a sound source is installed in front of the core panel. The air is modeled with the ALE element to compute the sound field surrounding the core panel and diaphragm. Basically, this layout is used, and the core panel is replaced with various types to perform calculations. As a material model, the core panel is made of steel and isotropic elastic. All nodes on the edges of the panel are fully constrained. The diaphragm is modeled as rigid, and five degrees of freedom other than the vibration direction (normal direction of the surface) are constrained. For the air, the pressure is computed using the equation of state (*Eq.1*). Where, E_0 is the initial internal energy per unit volume, v_0 and ρ_0 are the volume and density in the initial state, respectively. C_p is the specific heat at constant pressure, C_v is the specific heat at constant volume.

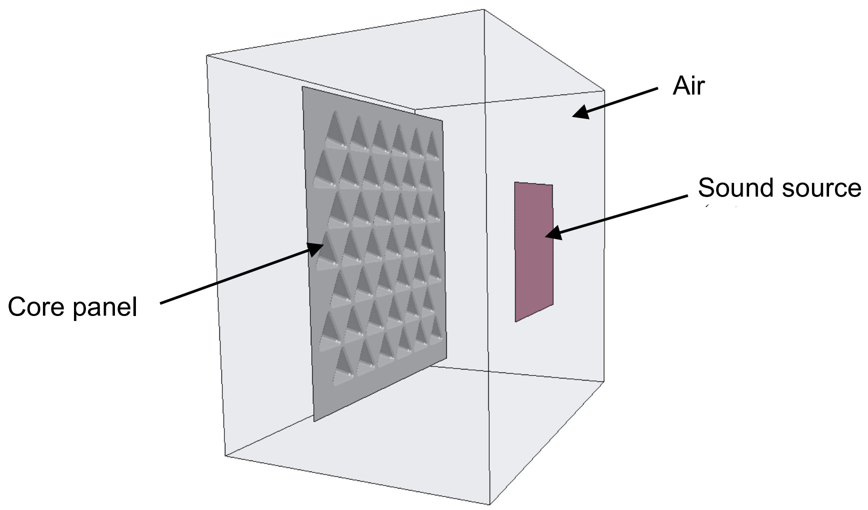


Fig.1: Model configuration for acoustic analysis of core panel

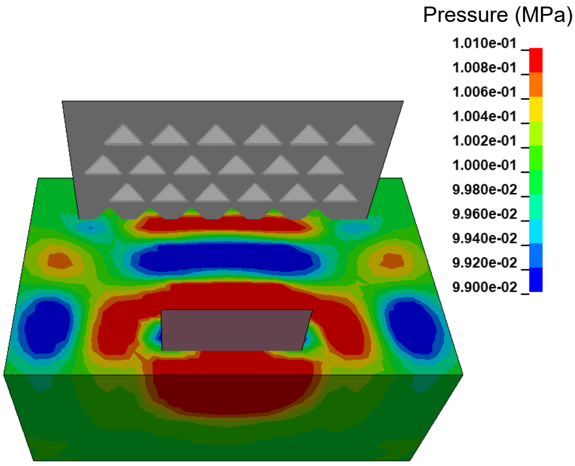
$$p(\mu, E_0) = (\gamma - 1)(\mu + 1)E_0 = (\gamma - 1)\frac{\rho}{\rho_0}E_0$$
$$\mu = \frac{v_0 - v}{v} = \frac{\rho}{\rho_0} - 1, \quad \gamma = \frac{C_p}{C_v} \tag{1}$$

3 Results and summary

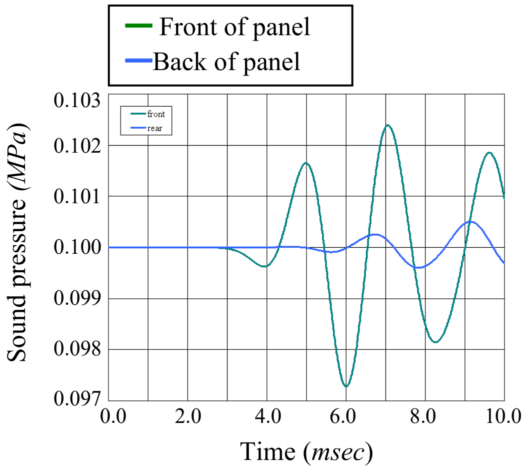
Figure 2(a) shows an example of the result of computation showing the sound pressure distribution of sound waves propagating in the air. This figure shows the cross section of the air layer and visualizes the sound pressure distribution. Sound waves are reflected, absorbed or transmitted after reaching the core panel. Figure 2(b) shows the history of sound pressure calculated by the ALE elements in front of and behind the core panel. Since the dynamic explicit method is used, the sound pressure is computed in the time domain. Therefore, in order to compare with the effective sound pressure measured in the experiment, the effective sound pressure is obtained by *Eq.2* from the pressure history from the simulation. Where p_e is the effective sound pressure, p is the calculated sound pressure, and t_1 and t_2 are the start and end times of the time when the sound pressure is in the steady state, and $T = t_2 - t_1$. In this paper, the results of sound absorption performance are compared and discussed for various types of core panels. The transmition loss of each core panel for various frequency was computed using Eq.3 and compared.

$$p_e = \sqrt{\frac{1}{T}\int_{t_1}^{t_2} p^2(t)dt} \quad (2)$$

$$TL = 20\log_{10}\frac{p_i^e}{p_t^e} \quad (3)$$



(a) Pressure distribution in the air



(b) Pressure history

Fig.2: Example of sound pressure computation

Magnets dynamics using LS-DYNA®

Trang Nguyen[1], Inaki Caldichoury[1], Pierre L'Eplattenier[1], Lars Kielhorn[2], Thomas Rüberg[2], Jürgen Zechner[2]
[1]Ansys - Livermore Software Technology LLC, 7374 Las Positas Road, Livermore, CA 94551, USA
[2]Tailsit, Nikolaipl. 4, 8020 Graz, Austria

1 Abstract

The LS-DYNA® Electromagnetic solver (EM) has recently integrated a new monolithic FEM (Finite Element Method) – BEM (Boundary Element Method) solver along with an AMS (Auxiliary Maxwell Space) preconditioner. Eddy-Current and Magnetostatic - including linear or non-linear magnetic materials - analysis can be done thanks to these new implementations. On top of this, the capability to have permanent magnets has been introduced.
We will start by showing a benchmark between LS-DYNA® and ANSYS Maxwell on the force calculation between two magnets in different conditions.
The model consists of 2-cylinder magnets with distance d (in mm). Each magnet is surrounded by an insulator. In ANSYS Maxwell, the 2 magnets are put in an air box. But no air mesh is needed in the Electromagnetic solver in LS-DYNA®. The configuration of the model is presented in Figure 1.

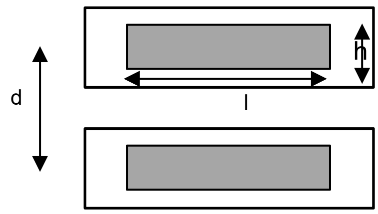


Fig 1: 2 magnets surrounded by an insulator at distance d.

The magnet is a Neodymium Iron Boron magnet with a magnetic coercivity of -890 kA/m. In the first comparison, a linear magnetic characteristic of the magnet is considered. Then a non-linear BH curve is introduced in the next comparison. The insulator is a linear material with no conductivity. Maxwell generates a tetrahedral mesh while LS-DYNA® generates hexahedral mesh.
Maxwell uses the virtual work principle to calculate the global force in the magnet, the expression of the force is derived from the classical magnetic energy (or co-energy). LS-DYNA® uses the Maxwell stress tensor derived directly from the magnetomechanical energy [1].
In Fig 2, the force in case of linear characteristics of magnet is plotted as a function of distance d between two magnets. The results obtained by Maxwell and LS-DYNA® are very close.
In the second part of this paper, different simulations of magnets in movement (and possibly coming in contact) due to attraction or repulsion will be presented.

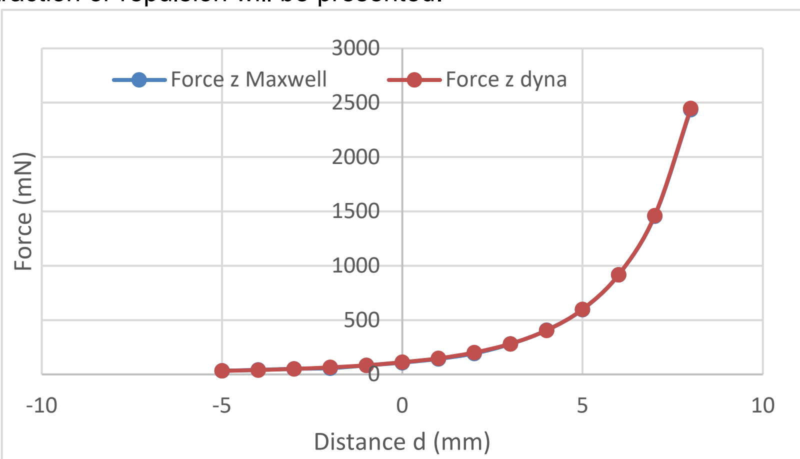


Fig 2: Force calculation by Maxwell and Dyna (linear characteristics)

[1] François Henrotte and Kay Hameyer "A theory for Electromagnetic Force Formulas in Continuous Media", IEEE Transactions on Magnetics, Vol 43, No. 4, 04/2007

Validation of the CHEMISTRY Solver in LS-DYNA

Hamid Rokhy[1], M.Sc., Reza Nasouri[2], Ph.D., Arturo Montaya[2], Ph.D., Adolfo Matamoros[2], Ph.D., Roya Backzadeh[3], M.Sc.

[1]Department of Aerospace Engineering, Amir Kabir University of Technology, Tehran, Iran
[2]Department of Civil and Environmental Engineering, University of Texas at San Antonio, TX, USA
[3]Department of Mining Engineering, Urmia University, Iran

1 Abstract

Recently the CHEMISTRY solver was added to the LS-DYNA software enabling users to model and predict accidental gas explosions in refinery plants, pipeline, and coal mines to reduce human casualties and the release of greenhouse gases such as NOx. Although this new solver has shown valuable potentials in chemical, oil and gas refineries, there are limited studies implementing such capabilities provided in CHEMISTRY solver. In this study, a series of fundamental chemistry problems were simulated using this solver to compare the numerical results with existing experimental data. This study shows that there is an excellent agreement between solver results and experimental proving the high level of precision obtained through the CHEMISTRY solver.

Calibration of Six Constitutive Material Models for Geomaterial

Hamid Rokhy[1], M.Sc., Reza Nasouri[2], Ph.D., Adolfo Matamoros[2], Ph.D., Arturo Montaya[2], Ph.D., Roya Backzadeh[3], M.Sc.

1 Department of Aerospace Engineering, Amir Kabir University of Technology, Tehran, Iran

2 Department of Civil and Environmental Engineering, University of Texas at San Antonio, TX, USA

3 Department of Mining Engineering, Urmia University, Iran

1 Abstract

Recently several constitutive material models have been developed and added to LS-DYNA library to predict concrete geomaterial behavior. These developments were established on merely using concrete compressive strength which limit the level of robustness in capturing actual concrete behavior. This study focuses on developing a simplified approach to calibrate six constitutive material models including Soil and Foam, Pseudo Tensor, Geological Cap Model, Concrete Damage Model Rel 3, Johnson Holmquist, and Continues Surface Cap Model against Triaxial and Hydrostatic Compression Tests (TXC and HCT) data. Comparison between individual numerical results were performed to evaluate accuracy can be offered through corresponded constitutive material model. The presented calibration method can also be applied to different geomaterials such as rock and soil.

Numerical Simulation of Rock Cutting Mechanism of Tunnel Boring Machine

Hamid Rokhy[1], M.Sc., Reza Nasouri[2], Ph.D., Adolfo Matamoros[2], Ph.D., Arturo Montaya[2], Ph.D., Roya Backzadeh[3], M.Sc.

1 Department of Aerospace Engineering, Amir Kabir University of Technology, Tehran, Iran

2 Department of Civil and Environmental Engineering, University of Texas at San Antonio, TX, USA

3 Department of Mining Engineering, Urmia University, Iran

1 Abstract

Forces applied to different region of disc cutter head are crucial design parameters for tunnel boring machine used for rock cutting purposes. These forces comprise of normal, rolling, and side forces which are used to determine required parameters for evaluating cutting geometries tool (e.g., cutter spacing and penetration). Existing models developed to predict these forces are based on linear cutting test experiments, while in practice rock is cut in a rotary path. This may result in a different set of outputs for the cutting forces. This paper presents a numerical approach simulating the Rotary Cutting Mechanism (RCM). Since experimental data for RM are limited, linear cutting experimental test results were used to validate the numerical simulation. To extend the level of verification, a 3-D model of a selected experiments was developed to compare the forces generated on disc cutter. Results showed that the developed RCM model predicted required force fairly close to what observed in experimental tests.

Benchmarking concrete material models using the 2D and 3D SPH formulations in LS-DYNA

Brian Terranova[1], Andrew Whittaker[2], Len Schwer[3]

[1] Research Engineer, University at Buffalo, 232 Ketter Hall, Buffalo, New York, 14260, U.S. (brt2@buffalo.edu)
[2] SUNY Distinguished Professor, Department of Civil, Structural, and Environmental Engineering, University at Buffalo, 212 Ketter Hall, Buffalo, New York, 14260, U.S. (awhittak@buffalo.edu)
[3] Schwer Engineering & Consulting Services, 6122 Aaron Court, Windsor, California, 95492, U.S. (Len@schwer.net)

Abstract

The unconfined, quasi-static behavior of three concrete material models available in LS-DYNA and compatible with the SPH formulation (i.e., MAT016, MAT072R3, and MAT159) were investigated by analysis of a 2D and 3D SPH cylinder with a diameter and height of 400 mm. Models were also prepared using axisymmetric (2D) and 3D Lagrangian (LAG) solid elements and analyzed to generate benchmark stress-strain data. Mesh refinement studies on the 2D and 3D LAG and 2D SPH cylinders were performed to identify a converged mesh size (particle spacing for the SPH models) for each concrete material model. The results highlight the importance of mesh refinement studies as the element size (or particle spacing) can have a significant effect on predictions of elastic modulus, peak average axial stress, and post-peak behavior. Analysis of the 2D and 3D LAG models with a converged mesh showed post-peak softening for MAT072R3 and MAT159 and non-softening (i.e., perfectly plastic) behavior for MAT016. The 2D and 3D SPH cylinders with a converged particle spacing reasonably recovered the elastic modulus, peak average axial stress, and post-peak behavior predicted using the LAG cylinder for all three material models. Additional numerical studies were conducted to compare the converged LAG and SPH model predictions with experimental data and identify the effect of cylinder aspect ratio on stress-strain behavior; results are presented in an appendix.

Analyzing bicycle accidents with human body models

Victor Alvarez[1], Heino Wendelrup[1], Hövding Karin Brolin[1],

[1]Lightness by Design

1 Abstract

Single-bicycle accidents are the most common cycling accidents in Sweden (more than 70% of all injuries) [1-4] and other countries where many people use bikes as means of transportation [5-6]. Vehicles are involved in a majority of the lethal bicycle accidents. Neck injuries are a small portion of all cycling injuries but are associate with a large risk of permanent medical impairment. Therefore, it is interesting to explore if head protective safety devices can provide safety benefits for the neck as well. Hövding is a head protective device, that is worn as a scarf around the neck, with sensors that trigger inflation of an airbag in case of an accident. Theoretically, the portion of the airbag that surrounds the neck could protect from neck injuries. To study this, numerical simulations with human body models developed for automotive safety is a tempting option. The aim of this study was to evaluate the suitability of two human body models (HBMs) to predict neck injuries in simulations of bicycle accidents. Therefore, this is a methodological study generating knowledge that eventually will enable systematic simulation-based neck injury risk evaluation of Hövding, and other neck protective devises, using HBMs. Included in the study where a model representing an average sized female and a model representing an average size male. Chalmers University of Technology have developed the open-source ViVA average size female HBM with an anatomically detailed model of neck validated on component, segment and full body levels [7]. The Global Human Body Models Consortium's (GHBMC) simplified average sized male model was chosen as it is relatively easy to position, is computationally efficient and has been developed with systematic validation [8] (Elemance Ltd, Clemmons, North Carolina, USA.) LS-DYNA (Ansys Inc. / LST, Canonsburg, Pennsylvania, USA) was chosen for all simulations. The two HBM's were positioned on a finite element model of a bicycle [9]. Simplified models of the Hövding airbag and a conventional cycling helmet were developed and adjusted to each HBM. Two impact scenarios were simulated with both HBM's without protection, with conventional helmet and with Hövding. Results were compared to experimental data where a stunt person crashed into a concrete road barrier. Neck injury criteria and thresholds were reviewed to analyze the simulation outputs. The results indicated that the methodology was promising. Both HBMs could be combined with relevant neck injury criteria, either global dummy type criteria or tissue level criteria. The ViVA model had issues with numerical instability due to low stiffness of soft tissues and required manual improvement to run through the whole accident scenario. The simplified GHBMC model was numerically robust but seemed a bit stiff, which needs to be studied further.

2 References

1. Kjeldgård L, Ohlin M, Elreud R, et al. (2019) Bicycle crashes and sickness absence - a population-based Swedish register study of all individuals of working ages. BMC Public Health 19:943. https://doi.org/10.1186/s12889-019-7284-1
2. Trafikverket (2014). Safer cycling – a common strategy for the period 2014–2020, Version 1.0. Trafikverket, rapport 2014:35. https://trafikverket.ineko.se/se/tv17634
3. Niska A & Eriksson J (2013) Statistik över cyklisters olyckor - Faktaunderlag till gemensam strategi för säker cykling. VTI rapport 801. https://www.vti.se/sv/publikationer/publikation/statistik-over-cyklisters-olyckor_694821
4. Rizzi M, Stigson H, Krafft M. (2013) Cyclist Injuries Leading to Permanent Medical Impairment in Sweden and the Effect of Bicycle Helmets. Proceedings of IRCOBI Conference, Gothenburg, Sweden. IRC-13-46. http://www.ircobi.org/wordpress/downloads/irc13/pdf_files/46.pdf
5. Amoros, E., Chiron, M., Martin, J.L., Thélot, B., Laumon, B., 2012. Bicycle helmet wearing and the risk of head, face, and neck injury: A French caseecontrol study based on a road trauma registry. Inj. Prev. 18, 27–32. http://dx.doi.org/10.1136/ip.2011.031815
6. Schepers P, Klein Wolt K (2012) Single-bicycle types and characteristics. Cycling Research International, 2:119 – 135

7. Östh J. Mendoza-Vazquez M, Linder A. et al. (2017) The VIVA OpenHBM Finite Element 50th Percentile Female Occupant Model: Whole Body Model Development and Kinematic Validation. Proceedings of IRCOBI Conference, Antwerp, IRC-17-60. http://www.ircobi.org/wordpress/downloads/irc17/pdf-files/60.pdf
8. Decker, W., Koya, B., Gayzik, F.S., 2018. Validation of detailed organ modularity in a simplified human body model. Proceedings of IRCOBI Conference, Athens, IRC-18-59. http://www.ircobi.org/wordpress/downloads/irc18/pdf-files/59.pdf
9. Hardy, R.N., Watson, J.W., Carter, E., Neal-Sturgess, C., Joonekindt, S., Yang, J., Herman, K., Baumgartner, D., Guerra, L.J., Martinez, L., 2004. Impact conditions for pedestrians and cyclists - APROSYS Deliverable D3.2.3. https://trimis.ec.europa.eu/sites/default/files/project/documents/20120313_143640_26037_AP-SP32-009R_D323.pdf

Predicting the results of the finite element simulation of a snowboarding backward fall with ODYSSEE

Wei wei[1], Dorian Salin[2], Nicolas Bailly[1]

[1]Aix Marseille Univ, Univ Gustave Eiffel, LBA, Marseille, France
[2] CADLM part of Hexagon, Wissous, France

1 Introduction

Snowboarding backward fall is the most common crash scenario leading to brain and spinal injury in the ski slopes [1]. In order to design effective protection against those injuries, it is critical to understand the injury mechanism. Previous work reproduced hundreds of crashes scenarios using the fast and robust multibody simulation method, highlighting a large range of head impact conditions but unable to provide information on local brain and spine loadings [1], [2]. The use of human finite element (FE) model would enable to access that information but is too costly and time consuming to reproduce hundreds of impact scenarios. The purpose of this study was therefore to propose a method coupling a FE Human model (THUMS) with a Reduced Order Modelling (ROM) technique from ODYSSEE [3] to provide real-time analysis of the snowboarding backward fall.

2 Method

The range of normal (Vn) and tangential (Vt) impact velocity and of head impact angle (α) relative to the ground, were extracted from the 324 multibody reconstructions of snowboarding backward fall [1]. A Montecarlo method was used to define a design of experiments (DOE) with 9 impact conditions covering the range of those 3 parameters. These impact conditions were reproduced using the THUMS v5.03 M-50 occupant model [4] impacting a snow ground on the LS-DYNA 971 R11.1 solver (LSTC, Livermore, CA, USA). The snow ground was model as viscous foam with the material properties extracted from a previous work [1]. Three additional impact scenarios, randomly chosen within the range of the three parameters were also reproduced using the Thums Model for validation. The maximal principal strain (MPS) in the brain as well as the loading force in the cervical vertebrae of the first 9 simulations were extracted and used in ODYSSEE to predict those of the three other simulations. The MPS, and vertebral loading curves predicted by lunar were compared to those obtained by the FEM simulations.

3 Results

Twelve impact scenarios were simulated (9 for the model and 3 for validations). The time of 1 simulation was approximatel 10 h on an Intel Xeon (2.20 GHz) workstation with 24 processors. All three parameters had a significant effect on the force in the C1 vertebrae, but this force was most affected by Vn and α. Figure 1 presents comparison between the force measure in the C1 vertebra of the THUMS model and the force in C1 predicted by ODYSSEE in the 3 tested impact conditions: Lunar closely predict in real time the force curve with an R2 of 79.8, 88 and 82 for the cases 1, 2 and 3 respectively.

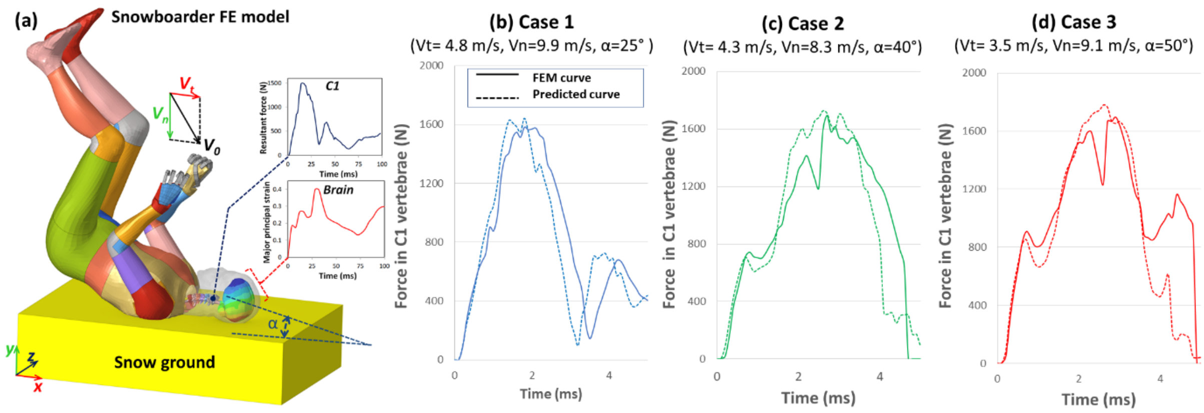


Fig.1: (a) FE model setup and result analysis for snowboarder falling on the snow ground and (b)-(d) Comparison between the force measurement in the C1 vertebra of the THUMS model and the force in C1 predicted by ODYSSEE.

4 Conclusion

The use of ROM methods was adequate to predict tissue-level loadings of the THUMS model during a snowboarding backward fall. The method can highly reduce the calculation time as it gives results in real time and requires only a small sample of simulations for learning. This work is a proof of concept that the methodology could be used on detailed biomechanical model and open up numerical design and evaluation of protective devices.

5 Literature

[1] N. Bailly, M. Llari, T. Donnadieu, C. Masson, and P.-J. Arnoux, "Head impact in a snowboarding accident," *Scand. J. Med. Sci. Sports*, vol. 27, no. 9, pp. 964–974, 2017.

[2] W. Wei, M. Evin, N. Bailly, M. Llari, J. Laporte, and P. Arnoux, "Spinal injury analysis for typical snowboarding backward falls," *Scand. J. Med. Sci. Sports*, vol. 29, no. 3, pp. 450–459, 2019.

[3] Kayvantash, K., 2019, "Model Order Reduction Techniques For Real-Time Parametric Crash and Safety Simulations," CARHS Automotive CAE companion, pp. 76–79.

[4] M. Iwamoto, Y. Nakahira, H. Kimpara. "Development and validation of the total human model for safety (THUMS) toward further understanding of occupant injury mechanisms in precrash and during crash," Traffic injury prevention, vol. 16(sup1), pp. S36-S48, 2015.

Comparing the Frontal Impact Responses of the VIVA+ Average Female and SAFER Average Male Human Body Models in a Generic Seat

Erik Svenning[1], Krystoffer Mroz[2], Torbjörn Johansen[1], Nils Lubbe[2], Johan Iraeus[3]

[1]DYNAmore Nordic AB
[2]Autoliv Research
[3]Chalmers University of Technology

1 Abstract

The VIVA+ 50F average female Human Body Model (HBM), currently in early beta status, was compared to the SAFER average male HBM Version 9 with the aim of investigating differences between females and males in terms of kinematics and injury assessment in frontal impacts. The VIVA+ HBM is under development within the research project VIRTUAL and will be released as open source during the summer 2022.

The comparison between the HBMs was carried out using LS-DYNA version R9.3.1 in a generic sled test interior consisting of a semi-rigid seat, a footwell and a pretensioned three-point belt system with 3.5 kN load limiter. The HBMs were positioned in a pre-simulation using spring and damper elements attached to target points estimated using an automobile driving posture prediction model from the literature. Subsequently, the HBMs were subjected to a frontal crash corresponding to an initial velocity of 50 km/h.

Occupant kinematics were analyzed by comparing head, chest, and pelvis kinematics between the two models. Additionally, HIC15, rib peak strains, and upper neck, lumbar spine and pelvis anterior superior iliac spine (ASIS) resultant forces were compared between the two models. The largest differences between SAFER HBM and the beta version of VIVA+ were found for rib peak strains, where VIVA+ predicts higher strain than SAFER HBM, and lumbar spine forces, where VIVA+ predicts lower forces than SAFER HBM. Furthermore, higher neck forces and ASIS forces were predicted by SAFER HBM compared to VIVA+.

Characterisation of an Energy Absorbing Foam for Motorcycle Rider Protection in LS-DYNA

Steffen Maier[1], Martin Helbig[2], Holger Hertneck[3], Jörg Fehr[1]

[1]Institute of Engineering and Computational Mechanics (ITM), University of Stuttgart
[2]DYNAmore GmbH
[2]SAS-TEC GmbH

Abstract

Energy absorbing foam elements are an essential safety aspect for new mobility concepts. For example, in a novel motorcycle safety concept that restrains the rider to the motorcycle in accidents, energy absorbing foams protect the rider from hard impacts. To improve and speedup the development process adequate crash-material models are needed to represent such density-dependent foams.

Under rapid impact, the proposed foam type reacts with high resistance, while it yields gently under slow impact, i.e., similar to a non-Newtonian fluid. To model such behaviour, the LS-DYNA material model **MAT_FU_CHANG_FOAM** is used. To parametrise the material model, dynamic and quasi-static experiments were conducted. The stress-strain data is surface fitted using bipolynomial regression. The polynomial coefficients are determined by minimising a quadratic objective function, with linear equality and inequality constraints incorporating material card requirements and surface topology knowledge. Parameter variations are performed to characterise the unloading behaviour. Drop tower experiments are performed, to verify accuracy for impactors with different geometries and impactors smaller than the respective foam samples.

The experimentally gathered force response in the drop tower tests shows good agreement with equivalent simulations of an energy absorbing foam with a density of 425 g/l. The impactor rebound, tracked with a motion capture setup, shows high sensibility for the chosen unloading parameters. In simulations of the novel motorcycle safety concept, the foam shows good performance in protecting a motorcyclist's lower extremities. The demonstrated procedure of fitting a polynomial representation of dynamic and quasi-static tests combined with a material model that supports near test conditioning is very efficient and fast. The parameterisation of the unloading behaviour, on the other hand, is much less automated.

The novelty of the demonstrated procedure is the polynomial representation of experimental test data incorporating material card requirements and material behaviour knowledge to characterise the **MAT_FU_CHANG_FOAM** material cards.

Numerical and experimental investigations of thick-wall composite hydrogen tanks

Mathieu Vinot[1], Tobias Behling[1], Martin Holzapfel[1], David Moncayo[2]

[1] Institute of Structures and Design, German Aerospace Center, Stuttgart
[2] Daimler AG, Research and Development, Sindelfingen, Germany

1 Abstract

Keywords: hydrogen tanks, thick-wall laminates, crash application, composite simulation

Hydrogen-powered fuel cells have established as one of the main emission-free alternatives to combustion engines in a broad range of transportation systems. This technology is however limited by the lower energy content of hydrogen on a volume basis compared to fossil fuels. For the mass market, high-pressure tanks have to be further developed so that costs fall while maintaining high performances.

Type IV composite pressure vessels are submitted to high internal mechanical loading at 700 bars and require the manufacturing of thick-wall laminates. To achieve optimal sizing and save weight, predictive numerical solutions need to be developed to simulate the complex failure mechanisms in thick-walled laminates under various loading scenarios. Moreover, the in-situ laminate layup and quality are strongly dependent on the manufacturing processes (i.e. winding technology) and can influence the mechanical potential of the vessel. It is therefore necessary to first investigate local fibre architecture in pressure vessels and determine the local material properties.

In the presented work, carbon fibre reinforced thermoset pressure vessels have been provided by Daimler AG to be investigated at the DLR test facility. Local fibre layups are first investigated with the Computer Tomography (CT) technology on small in-situ cylindrical specimens and used as input for the simulation. In a second step, failure mechanisms are investigated with the Digital Image Correlation (DIC) technique with special focus on delamination effects. Impact tests are finally performed in the DLR drop tower facility in Stuttgart on several energy levels to estimate the impact performances of tank segments.

Simulating the behaviour of thick-wall composites necessitates the use of special modelling methods compared to classical thin-walled laminates. In this work, a stacked-layer approach with TSHELL elements and a cohesive contact formulation is investigated. Digital twins of every experimental tests with their precise boundary conditions are built up with this approach and first ran in a predictive manner. With the developed numerical approach, the simulated mechanical behaviour is in good agreement with the experimentally observed failure patterns and load curves (Figure 1). Furthermore, delamination effects have been accurately reproduced in simulation. Prediction capabilities still have to be improved through the consideration of strain-rate effects in the layers and at the interfaces.

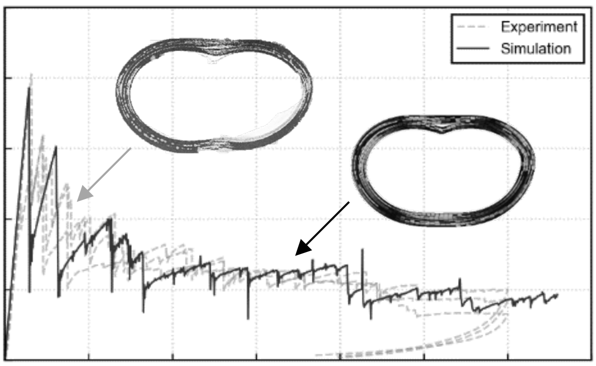


Figure 1: Experimentally and numerically measured force-displacement curves and delamination pattern in a ring compression test

Thermo-mechanical homogenization of composite materials

Shadi Alameddin[1], Felix Fritzen[2]

[1] Data Analytics in Engineering, Institute of Applied Mechanics, University of Stuttgart
[2] Data Analytics in Engineering, Institute of Applied Mechanics, University of Stuttgart

This work presents a multiscale simulation framework that will be used for the simulation and experimental validation of eigenstresses in composite materials generated via laser-dispersion. These materials are obtained by adding tungsten carbide particles into the melt pool of a base metal to generate surface coatings. Such coatings are used to boost wear-resistance, more precisely to protect metallic surfaces against abrasion, erosion or corrosion. The coating significantly extends the part's lifetime due to the outstanding material characteristics of the locally produced metal matrix composite (MMC). Eigenstresses, which are the residual stresses left in the MMC material after the coating process, shall be investigated and predicted within the framework of this project and their effect on the lifetime shall be estimated.

Computational homogenization is employed to predict the thermo-mechanical response of heterogeneous structures. In such schemes, the constitutive response of every integration point in the macro scale is obtained by solving a boundary value problem on a representative volume element (RVE) with boundary conditions from the corresponding integration point. Direct numerical simulation (DNS) of RVE response, which can be seen in FE^2 context, is computationally demanding. In contrast, reduced order models (ROM) and data-driven surrogate models provide an appealing and efficient alternative to DNS. In this work, an interface to LS-DYNA is developed to allow for:

1. implementation possibilities of user-defined material routines (UMATs) in Python and C++
2. easy integration of temperature-dependent thermo-mechanical material parameter in LS-DYNA
3. direct integration of data-driven models and ROM written in Python and C++
4. the usage LS-DYNA or self-written software as the DNS solver to compute RVE responses

Simple examples and an open-source code are provided to allow a straightforward deployment of UMATs and two-scale simulations in LS-DYNA.

Numerical Investigation of the Forming Behavior of Polymer Composite-Metal Hybrids using Fiber Reinforced Thermoplastic Tapes with Discontinuous Layup

Philipp Kabala[1], David Trudel-Boucher[2], Ingo Karb[3], Ric Meissner[3], Tim Ossowski[1], Klaus Dröder[1], André Hürkamp[1]

[1]Institute of machine tools and production technology (IWF), Technische Universität Braunschweig
[2]National Research Council Canada
[3]Compositence GmbH

Abstract

Fibre-reinforced plastic composites offer great potential to fulfil the increasing requirements for environmental protection and crash safety in the automotive industry due to their high specific strength and stiffness as well as energy absorption capacity. Fibre reinforced thermoplastics (FRTP) in particular are suitable for large-scale production owing to their good recyclability and short processing cycle times. However, FRTPs such as organic sheets or laminates consisting of tapes have a limited formability. This is primarily due to the lack of plasticity of the fibres as well as to the limited shearing of the weft and warp threads in fibre fabrics, which can lead to fibre displacement, fibre breakage, or wrinkling for complex-shaped components. In order to overcome the higher cost of FRTPs, the fabrication of polymer composite-metal hybrid structures is now currently being considered in the industry. To ease manufacturability of such hybrid materials using a one-step forming processes, an approach consisting of overlapping composite tape sections that can slide relative to off each was developed. This allow composite materials to display a global ductility similar to the one of metals during forming. However, such an approach has yet to be fully characterized.

In this paper, a one-step forming process of an intrusion beam consisting of steel sheet and FRTP is modelled using LS-Dyna. For this, the material models MAT_037 "TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_NLP_FAILURE" and MAT_249 "REINFORCED THERMOPLATIC" are applied. First, conventional FRTPs laminates made of continuous tapes are used. Subsequently, the FRTP is replaced by discontinuous tape sections. The single tape layers of the FRTP are modelled considering interlaminar shearing. For this purpose, a velocity- and pressure-dependent coefficient of friction under isothermal conditions is defined. Thereby, the single layers with discontinuous layup are combined in one section consisting of several sub-components to reduce the modelling and computing effort for the contact.

By comparing the results of both simulations, the potential to improve the formability of the discontinuous FRTP will be underlined. Furthermore, it is shown that the length of the tape sections has a significant influence on the stress distribution in the FRTP laminate. This offers the potential of optimising the FRTP consisting of discontinuous layups to minimise stresses induced during forming.

Simulating the induction heating behavior of CFRTPC laminates

Thomas Hoffmann, Stephan Becker, Miro Duhovic, Peter Mitschang

Leibniz-Institut für Verbundwerkstoffe GmbH, Technische Universität Kaiserslautern, Erwin-Schrödinger-Straße 58, 67663 Kaiserslautern, Germany

Abstract

In the goal towards zero-emission aircraft design, weight efficiency is the decisive parameter. In combination with the challenges of new fuel storage systems and improved aircraft aerodynamics, novel aircraft geometries play a key role. A particular challenge is the shift from cylindrical fuselage designs to complexly curved structures utilizing integral, thermoplastic part design and manufacturing methodologies as well as the connection of these parts with one another. The continuous induction welding method fits well into this requirement profile due to its high degree of automation and energy efficiency. In addition, next generation manufacturing and computer aided engineering (CAE) technologies for the design of carbon fiber reinforced thermoplastic composite (CFRTPC) induction welding/joining processes for commercial aircraft structures also play a pivotal role.

In the process of continuous induction welding, the adherends are heated by electromagnetic induction and joined by applying pressure via a cylindrical consolidation roller. In this process, the laminate configuration has a major influence on the induction heating behavior of the laminate and thus on the quality of the resulting joint. Modeling the induction heating process not only serves to optimize laminate structures with respect to heating behavior, but also represents an important step towards designing efficient process control. Current FE models for the induction heating process in CFRTPC materials are mostly limited to simple isotropic material models describing the electromagnetic heating behavior of the laminate.

The objective of this work is to create an FEM-based model for the inductive heating of CFRTPC laminates. A macroscale simulation model was created using the multi-physics capabilities of LS-DYNA®. Material model parameters were largely determined by micromechanical considerations. In order to further increase the accuracy of the FEM model, DSC measurements were also carried out to determine the temperature dependence of the heat capacity of the laminates investigated. The model was then validated for laminates reinforced by non-crimped fabrics (NCF) with fiber volume contents of 32%, 47% and 60% via induction heating tests. In general, the heating experiments could be approximated well both qualitatively and quantitatively. Furthermore, analyses were carried out in order to investigate the influence of individual ply orientations in the laminate on one another as well as the influence of the layer thickness on the resulting heating behavior (see Fig. 1).

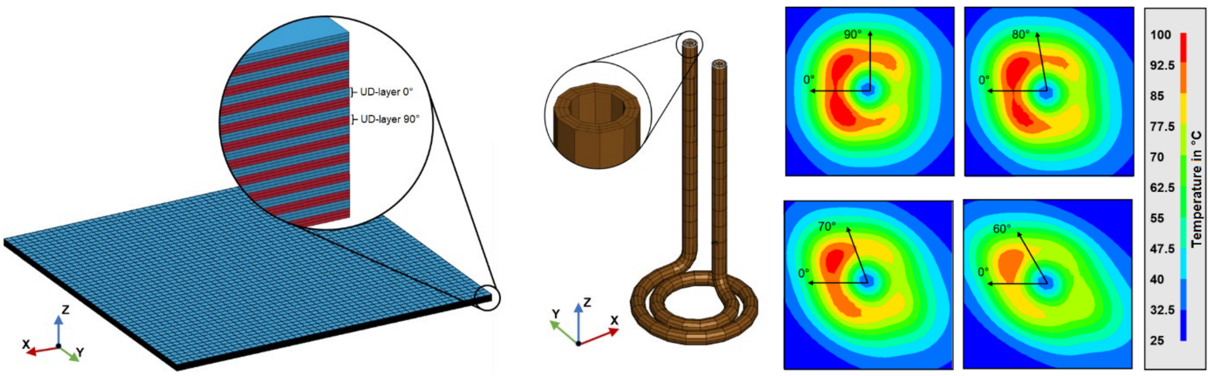


Fig.1: Mesh geometry of CFRTPC laminate (left) induction coil (middle) and temperature contours showing the influence of ply orientation in the laminate on the resulting heating pattern (right).

Experimental-numerical determination of the Taylor-Quinney coefficient

J. Johnsen[3], L.E.B. Dæhli[1], T. Børvik[1,2], O.S. Hopperstad[1,2]

[1]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), Trondheim, Norway
[2]Centre for Advanced Structural Analysis (SFI CASA), NTNU, Trondheim, Norway
[3]Enodo AS, Trondheim, Norway

Abstract

During plastic deformation of a metal, a part of the plastic work is stored in the material due to local distortion of the crystal lattice, while the remainder is dissipated as heat. The part of the plastic work dissipated as heat can be observed on a macroscopic scale through thermal measurements in high strain rate experiments. Typically, this fraction of plastic work converted into heat is assumed to be constant and around 90%. In this study, we have performed tension tests at a constant crosshead velocity of 0.6 mm/s on flat notched specimens from a DP600 material. Digital image correlation (DIC) was used to apply virtual extensometers spanning the length of the notched area. Furthermore, an infrared camera was used to measure the temperature increase over the same area as monitored by DIC, enabling correlation between temperature and displacement. These temperature-displacement curves were used as the target curves in thermomechanical simulations to obtain the Taylor-Quinney coefficient as a function of equivalent plastic strain. It was found that the Taylor-Quinney coefficient exhibits quite large variations during the experiment, ranging from a minimum of about 0.5 in the beginning of the test, to about 0.95 at the end of test.

Calibration of *MAT_258 with a Lode dependent fracture surface and its application in bending of high-strength steel

J.K. Holmen[1,2], J. Johnsen[1,2], D. Morin[2,3], T. Børvik[2,3], M. Langseth[2,3]

[1]Enodo AS, Trondheim, Norway (mail@enodo.no)
[2]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, Norwegian University of Science and Technology (NTNU), Trondheim, Norway
[3]Centre for Advanced Structural Analysis (SFI CASA), Department of Structural Engineering, NTNU, Trondheim, Norway

Abstract

`*MAT_258` (**`*MAT_NON_QUADRATIC_FAILURE`**) is a through-thickness failure regularization model for shells in LS-DYNA. In this model the failure indicator is computed as a function of both the size of the element and its bending-to-membrane loading ratio. The constitutive behavior and fracture surface in **`*MAT_258`** are represented by well-known analytical expressions which simplify calibration. We present the calibration process for **`*MAT_258`** with a three-parameter Extended Cockcroft-Latham fracture surface for the high strength steel Docol 1500M. The material card is applied in shell element simulations of three-point bend tests.

Dynamic behaviour study of a satellite propellant tank using numerical and experimental vibratory tests

Thomas Pierrot[1], Antoine Guilpin[1], Tess Legaud[1], Vincent Lapoujade[1],

Jean-Emmanuel Chambe[2], Miguel Charlotte[2], Yves Gourinat[2],

Mathieu Delorme[3]

[1]DynaS+, 5 avenue Didier Daurat 31400 TOULOUSE France, t.pierrot@dynasplus.com

[2]Université de Toulouse, CNRS, ISAE-SUPAERO, Institut Clément Ader, Laboratory

[3]ATECA, 4 chemin du quart-Verlhaguet 82000 MONTAUBAN, France

Abstract

The ecological transition necessity makes the use of cryogenic fluids more and more relevant. However, experimental tests and associated modelling of those liquids dynamic vibratory behaviour remain extremely challenging. Indeed, security, control and conditioning are critical issues due to the intrinsic fluid instabilities. Among those critical fluids, Liquid Hydrogen and Xenon are both highly used in the spatial propulsion domain. Because of their hazardous behaviour, only few experimental dynamic tests have been performed to improve the knowledge of their behaviour inside a vibrating tank.

Following the EASYNOV TANKYOU project, the READYNOV DANKE project, also funded by the French Occitanie region, aims at finding a safe substitute metamaterial that would be able to represent the liquid xenon vibratory behaviour in a fully filled tank. The main objective is to find the granular medium properties that enable to match the modal shapes and frequencies of the tank filled with this granular medium with the one filled with liquid xenon. The generalisation of this work will lead to a methodology combining numerical predictions, experimental validations and dimensional parametrization which should enable its uses to any other cryogenic fluid and larger applications.
The project combines analytical, numerical and experimental approaches, that are strongly linked to each other as part of a material by design study. To this end, the project has been conducted in two main parts:

- Design and validate, using numerical simulation, an experimental device highlighting the structural modes of the fluid and tank assembly,
- Evaluate the abilities of the numerical approach, developed in the TANKYOU project and based on the Discrete Element Method, to reproduce the dynamic behaviour of a fluid inside a vibrating tank.

This paper especially focuses on the numerical designing step of the experimental device and on the tank structural modes identification. The experiments use a simplified closed cylinder filled with spheres of various materials. The combination of those different approaches is the guiding thread leading to disruptive innovative research opportunities.

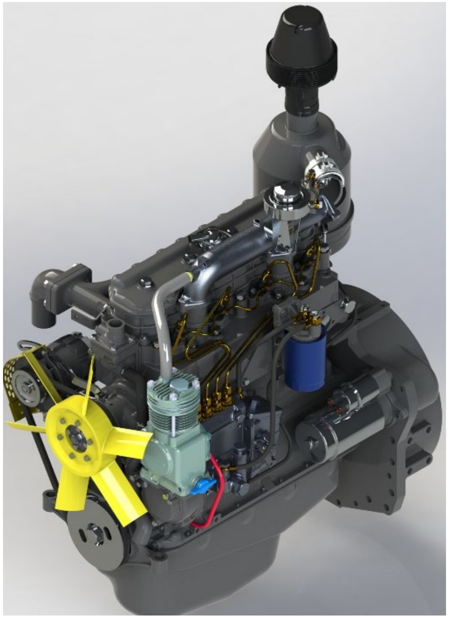
Combustion Engine Analyses Using New and Extended Features in LS-DYNA

Fredrik Bengzon[1], Anders Jonsson[1], Thomas Borrvall[1]

[1]Dynamore Nordic AB, Ansys LST

1 Abstract

Recently, many enhancements have been made to LS-DYNA's implicit solver with regard to improved robustness and general functionality. New material models, element formulations, and features for convenient load history management all contribute to making the implicit solver more versatile and user friendly. The new features are relevant to general analyses, but here a case study of a combustion engine (see Fig.1:) analysis is presented to illustrate and promote these new features. Focus will be on thermal and mechanical analyses. The built-in fatigue analysis capabilities of LS-DYNA will also be demonstrated, since fatigue life is often the design target in structural analyses of combustion engines.



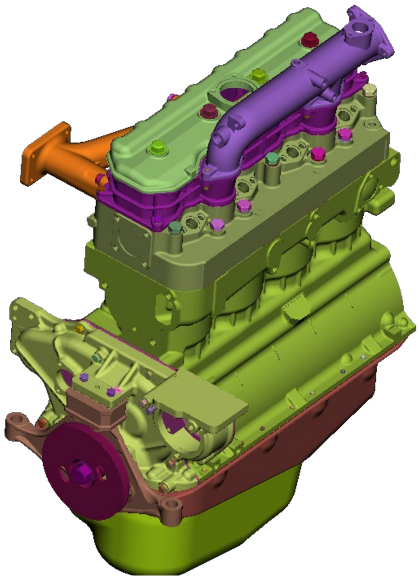


Fig.1: The left image shows the original CAD model. The CAD modelling work of Mr. Artem Slyusarev is gratefully acknowledged. The right image shows the parts kept for the structural analyses.

The new features that will be illustrated include:

- Gasket elements with corresponding material model (**`*MAT_COHESIVE_GASKET`**) allowing for a meso-level description of gaskets and seals, using pressure vs. closure curves for loading and unloading, see Fig.2:.

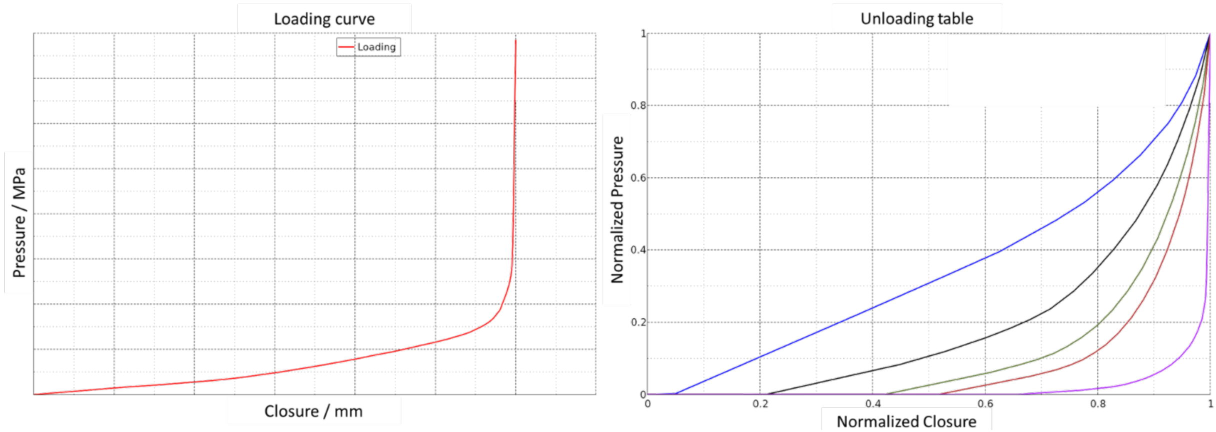


Fig.2: The pressure-closure relation for the gasket elements is defined by a loading curve (left image) and an unloading table (right image), using ****MAT_COHESIVE_GASKET****.*

- Enhanced strain hexahedral elements with incompatible modes (formulation -18) are used in for example cylinder liners.
- Temperature dependence of material properties is added to the non-linear mixed hardening material model ***MAT_DAMAGE_3**. This type of material model is often applied for modelling thermo-mechanical fatigue and cyclic plasticity, see Fig.3:. In this demonstration, ***MAT_DAMAGE_3** is used in the engine block, cylinder head and exhaust manifold.

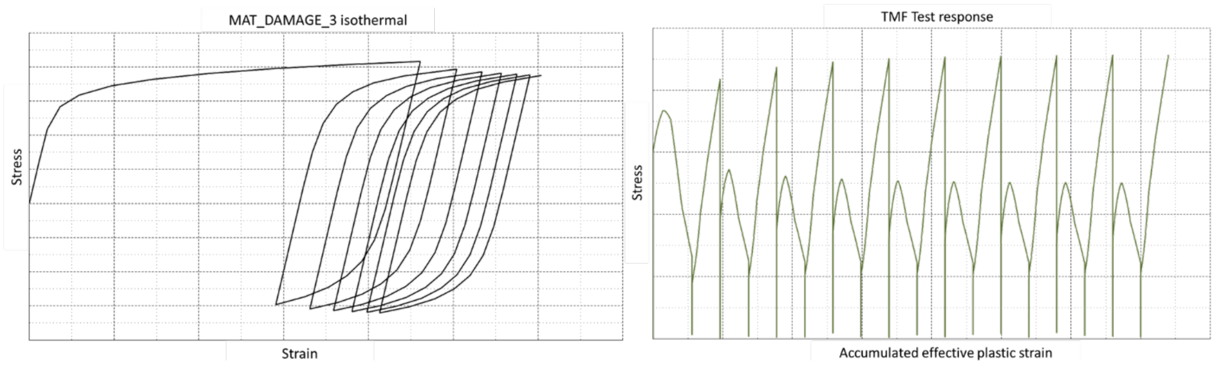


Fig.3: Typical behavior for the non-linear kinematic hardening model of ****MAT_DAMAGE_3****. The left image shows strain cycling at constant temperature. The right image shows the response under a TMF test, where both strain and prescribed temperature is varied simultaneously.*

- A new material model for polymer materials, including tension/compression asymmetry and a variable plastic Poisson's ratio (***MAT_SAMP_LIGHT**) is applied for modelling plastic parts of the oil pan.

The analysis illustrates the multistage concept, by combining ***CASE** and **dynain.lsda** to describe the loading sequence as a series of stages.

LS-DYNA holds a strong position within the automotive industry, as the market leading software for crash and safety analyses, using the explicit solver. Hopefully, this contribution can demonstrate the capabilities of the implicit solver within engine/drivetrain analyses which is quite new to LS-DYNA.

Enabling the *CONSTRAINED_INTERPOLATION_SPOTWELD (in detail SPR3) as a general-purpose fastening element

Tobias Erhart[1], Michal Styrnik[2]

[1]Dynamore GmbH
[2]BMW AG

It is known that the simulation of fastening elements can be carried out by using different approaches. One common way is the use a force-displacement based approach for single-point-connections. While enabling LS-Dyna to calculate crash simulations in our crash simulation tool chain it was necessary to make several adjustments to the standard **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** keyword to ensure that the currently available data can be used almost completely. The main improvements on the SPR3 element were accuracy features such as exponential decay, different stiffness factors for different loading conditions, the introduction of torsional moments, mixed-mode dependent hardening, the ability to connect with beam elements and the possibility to switch between different coordinate systems while calculating the rotational behavior of the element. Additionally, we were able to improve the overall stability of the penalty formulation by introducing a new approach to distribute forces and moments on arbitrary meshes, by monitoring the deletion of parent elements and by checking if a proper geometry is defined to proceed with the calculation (**`*CONTROL_CONSTRAINED`**). Considering the evaluation of the elements we introduced the "SPR state variable" to make the whole "life cycle" of a spotweld visible to the user. Due to all these measures the prediction quality of this approach has improved significantly. In order to add the possibility to define the material independently of the element **`*MAT_265`** was introduced. As future challenges we would also like to implement the proper identification of peel moments to switch between different degradation behaviors. And we would like to include the possibility to couple SPR elements properly with solid meshed parts.

Development of a finite element model of high energy laser-material interaction

Matthew Ross and Dan Pope

Dstl Porton Down, Salisbury, SP4 0JQ, UK

A thermomechanical modelling technique was developed using LS-DYNA for simulating the heating and the subsequent erosion of metallic elements by a continuous wave laser beam. Accurate representation of the laser-material interaction requires inclusion of several physical phenomena; heating via absorption of the laser beam, radiative cooling, convective cooling, thermal conduction and mass loss by phase change. To model the heating via absorption of the laser beam the use of the recently implemented LS-DYNA keyword, **`*BOUNDARY_FLUX_TRAJECTORY`**, was required. When modelling the phenomenon of "burn-through", this keyword was applied along with **`*BOUNDARY_CONVECTION`** and **`*BOUNDARY_RADIATION`** to update the transient surface heat flux condition upon element erosion (facilitated by **`*MAT_ADD_EROSION`**). Accurate material properties were required up to very high temperatures, and these properties were sourced via open literature, collaborative agreement and commercial material databases.

The modelling technique was validated against test data supplied by commercial partners. These experiments focused on the thermal loading of flat steel plates with differing laser beam powers. The simulations predicted very similar rates of in-plane temperature development and through-thickness "burn-through" to those measured in the equivalent experiments. The modelling technique can be further developed for simulating more complex geometries containing different materials, and can be expanded to incorporate additional physical phenomena such as the effect of cross-surface airflows and surface-to-surface radiative heat exchange.

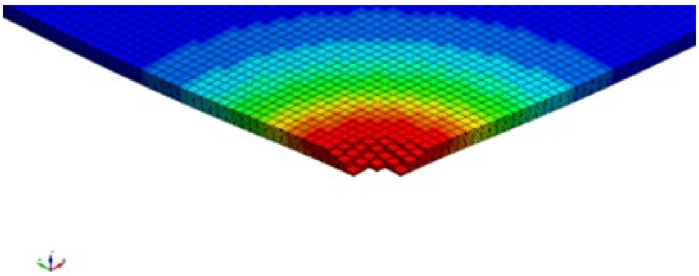


Fig. 1: Example LS-DYNA Mesh at laser simulation endpoint

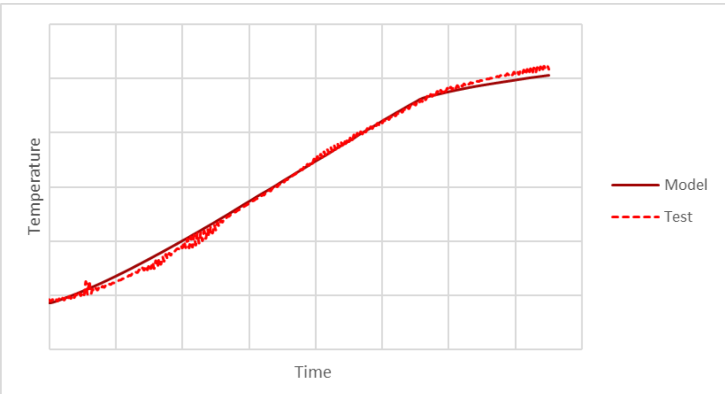


Fig. 2: Temperature comparison plot of model and test

Using MAT_ADD_INELASTICITY for Modelling of Polymeric Networks

Fredrik Bengzon[1], Thomas Borrvall[1], Anders Jonsson[1], Magnus Lindvall[2]

[1]DYNAmore Nordic AB
[2]IKEA

Abstract

Thermoplastics are found in many products that surround us in everyday life, e.g., automotive interiors, packaging containers, bottles, toys and medical equipment. The modelling of their material behavior is a challenging task, in part due to the inhomogeneous micromechanical composition of a matrix and various fiber inclusions. In addition, the interaction between linear polymer molecules results in a strong time and temperature dependent response, which is macromechanically manifested as viscoelasticity. It is therefore important to incorporate both short and long term loading characteristics into the constitutive modelling framework, ultimately leading to universal models that can handle the full life cycle of the product (impact, cyclic loading, recovery, stress relaxation, creep etc). The keyword MAT_ADD_INELASTICITY in LS-DYNA can be used with any standard material model to incorporate a parallel network of linear or nonlinear creep models to predict the viscous response for a large range of time and strain scales. The purpose of this paper is to present this modular concept, including usage and examples with results, and then discuss potential future enhancements.

Modelling and Simulation of the long-term Behavior of Thermoplastics in LS-DYNA

M. Morak[1], S. Seichter[2], I. Sladan2, R. Steinberger[2], W. Hahn[3], M. Göttlinger[3], H. Pothukuchi[4], P. Reithofer[4], M. Schwab[4]

[1]Polymer Competence Center Leoben GmbH
[2]Hirtenberger Automotive Safety GmbH
[3]Hilti AG 44a engineering GmbH

Abstract

There is a need for transitioning to an energy system with less greenhouse gas emissions and more sustainable energy production and consumption. A long-term structural change in energy systems is needed. Germany and France, among other countries, have decided to scale up the green hydrogen sector, with fundings of 9 billion and 7 billion euros respectively in the next 10 years.

Hydrogen as a new energy vector has many advantages over traditional hydrocarbon-based fuels. It is energy-efficient and can be environmentally friendly if it is be obtained from renewable sources. Potentially, in the future, it can solve many ecological and energy security issues. For more than a century, hydrogen has been produced and used for commercial and industrial purposes with a high safety record. However, the wider use of Fuel Cells and Hydrogen (FCH) technologies by the public (and not only by trained professionals) will require a new safety culture, innovative safety strategies, and breakthrough engineering solutions.

Injection molded energy absorber (Ultramid® PA-GF30) in the front end of Daimler S-Class MY2020

Andreas Wüst[1], Levente Juhasz[2]

Stefan Glaser[1], Sebastian Ebli[1], Torsten Hensel[1], Gerhard Summ[2], Marco Herok[2], Gernot Jäger[2]

[1]BASF SE
[2]Daimler AG

Energy management in passenger cars has traditionally been dominated by metal structures due to their high energy absorption efficiency. Due to changing legislations and increasing requirements there is a trend in car development towards spatially more distributed energy management concepts resulting in more and new loadpaths. This allows the use of plastic structures for moderate energies which are designed for a robust and controlled failure under variable ambient conditions. These structures are based on a controlled total destruction of the part and though need numerical material models which allow a precise description of the failure process. Used in the right way, the parts allow a high degree of functional integration by injection molding process and thus reduce the system cost. The new part concept which will be presented and explained during the session was jointly developed between Daimler AG, BASF SE and HBPO.

BASF's software Ultrasim® in combination with LS-Dyna was used for material modelling / numerical simulation. The early virtual part development was done based on former BASF crash concepts and adapted, newly developed structural geometries. Physical droptower tests allowed to monitor the critical aspects and showed the robustness of the concept under different ambient conditions. Although energy absorption involves many successive material fractures with subsequent self-contact of the fragments, the test/simulation comparison shows a wide agreement.

Based on this initial work, the first serial application took place in the MY2020 Daimler S-Class. Further platforms with the same concept will follow.

During the presentation the early virtual development steps, the Ultrasim® material modelling as well as physical tests and the performance of the first serial application will be shown.

Andreas Wüst, Ludwigshafen, 14.6.2021

Ceramic-Rubber Hybrid Materials – A Way to Sustain Abrasive Heavy Impact Applications

M. Herr[1], M. Varga[1], L. Widder[1], J. Mermagen[2], S. Rodinger[2], W. Harwick[2]

[1]AC2T research GmbH, Wr. Neustadt, Austria
[2] Fraunhofer EMI, Freiburg, Germany

1 Introduction

Many industrial applications require the transport of raw material which in general is a very abrasive situation. At certain occasions, e.g. transfer between two conveyer belts, additional impact cannot be avoided. For these purposes it is difficult to find wear resistant solutions. For pure impact applications often hyperelastic materials, e.g. rubber, are used, which can tolerate impact without damage. Those materials usually have the drawback of a poor abrasion resistance, as they do not contain protective hard phases. Abrasion resistance is mostly achieved by hard materials, which usually correlates with high brittleness. This, however, is detrimental for impact resistance. [1-3]
In this work we present an approach to combine the benefits of hyperelastic materials for impact resistance and hard ceramics for abrasion resistance. With those ceramic-rubber hybrid materials the lifetime of components in the aggressive environment of raw material handling and processing can be increased multifold. The utilized materials are characterized at high strain rates and numerical simulations aid the optimization of the geometries.

2 Experimental

Preliminary abrasion experiments were carried out to select an abrasion resistant alumina ceramic. Subsequently, ceramic-rubber hybrid plates were built and evaluated in a dropping hammer test: A sphere, backed by a defined weight, is dropped onto the ceramic. The impact load is steadily increased until the segment breaks. Thereby, a first indication is given, on how the ceramic size and the rubber influences the impact resistance.

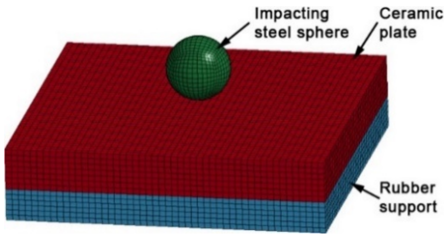


Fig.1: Numerical setup.

In a second step high-speed mechanical characterization of the rubber and ceramic was done by means of tension, compression and bending tests up to strain rates of 100 s^{-1}. These data were then used to tune a numerical model of the hybrid plates in LS-Dyna in order to calculate the limits of certain hybrid setups. The size of the ceramic (area and thickness) and the rubber thickness were varied (Fig. 1) to calculate a design for optimal hybrid plates at a given application.

3 Results

When using ceramics, much thicker than wide, impact energies up to 500 J could be sustained. When exceeding the maximum energy, a damage like shown in Fig. 2 can occur.
Tailored impact resistance could be calculated with the simulation model. For a given impact load we can calculate an optimal ceramic-rubber setup, to minimize material input and production costs.

Fig.2: Impact damage example: 48×48×65 mm ceramic on 9 mm rubber after 500 J impact

4 Acknowledgments

This work was funded by the Austrian COMET Program (InTribology, no. 872176) and carried out at the "Excellence Centre of Tribology". Kalenborn International GmbH & Co. KG and voestalpine Stahl GmbH are acknowledge for active research coopertion.

5 References

[1] M. Varga et al.: Ceramic-rubber hybrid materials – A knowledge-based design concept, Wear (2021) in press.
[2] P. Kulu, I. Hussainova, R. Veinthal: Solid particle erosion of thermal sprayed coatings, Wear 258 (2005) 488-496.
[3] H. Winkelmann et al.: Influence of momentum and energy on materials, Steel Research International 88/10 (2017) 1-15.

Modeling and Simulation of Hypervelocity Impacts on Spacecraft in Low Earth Orbit

Rannveig M. Færgestad[1], Jens K. Holmen[1,2], Torodd Berstad[1], Tiziana Cardone[3], Kevin A. Ford[4], Tore Børvik[1]

[1]Structural Impact Laboratory (SIMLab), Department of Structural Engineering, NTNU – Norwegian University of Science and Technology, Trondheim, Norway
[2]Enodo AS, Trondheim, Norway
[3]European Space Agency, ESTEC, Noordwijk, The Netherlands
[4]NASA, Johnson Space Center, Houston, USA

1 Summary

During the last decade, the low Earth orbit (LEO) has become extremely populated. In addition, several catastrophic impact events have contributed to the generation of thousands of debris in that region. This situation constitutes a continuous threat to our space missions. Thus, we need to understand what happens after our spacecraft is hit by a debris flying at hypervelocity and what is the probability that the elements (units, payloads, etc.) behind the spacecraft external panels, solar arrays, and other possible appendages, are damaged.

Hypervelocity impact experiments require highly specialized equipment, and all the relevant variations of impactor mass and velocity cannot routinely be reached in a laboratory setting. This makes numerical simulations essential for efficient design of debris shields. In this study, we have used a combined finite element-discrete particle method available in LS-DYNA [1] to simulate the response of Whipple shields [2] struck by space debris travelling at velocities between 1 and 10 km/s. We have also implemented a user-defined material model (UMAT), combining the modified Johnson-Cook material model [3] (or *MAT_107) with a non-linear Mie-Grüneisen equation of state (or *EOS_ GRUNEISEN). The numerical results are evaluated against hypervelocity impact experiments from the literature [4].

The proposed methodology predicts in general the shape of the debris cloud as a function of impactor shape, impactor velocity, angle of incidence, and the thickness of the Whipple shield. We conducted parametric studies to investigate the sensitivity of the material parameters, and the effect of using a non-linear EOS versus a linear EOS. A comparison is also made between simulations using the particle method and smoothed-particle hydrodynamics (SPH). Ballistic limit curves are determined for a single-layered Whipple shield configuration (see an example in Figure 1) and a corresponding monolithic configuration of equal areal mass. The effect of layering is captured by the simulations in the entire velocity range of interest and the flexibility of the computational method is illustrated by altering the standoff distance and shield thickness.

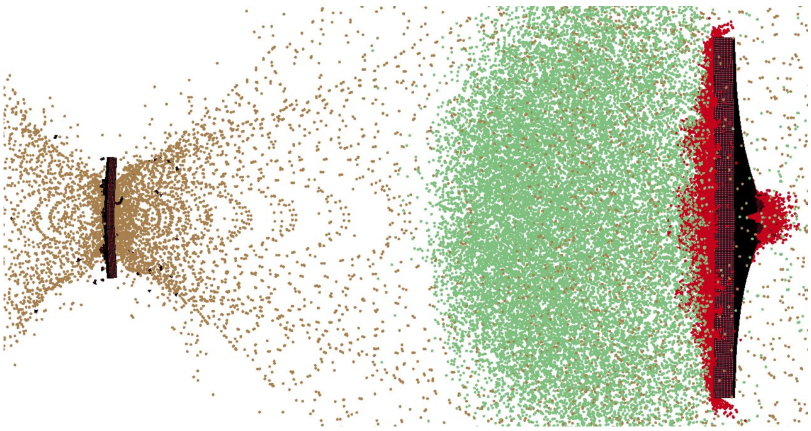


Figure 1: Example of a layered (1.2 mm + 100 mm + 3.2 mm) Whipple shield configuration perforated by a 6.5 mm diameter sphere at an impact velocity of 7 km/s. All materials are made of AA6061-T6.

2 Literature

[1] Livermore Software Technology (LST), An Ansys Company. LS-DYNA Keyword User's Manual, LS-DYNA R12, 2020.
[2] https://en.wikipedia.org/wiki/Whipple_shield.
[3] Børvik, T., Hopperstad, O.S., Berstad, T., Langseth, M.: A computational model of viscoplasticity and ductile damage for impact and penetration. European Journal of Mechanics – A/Solids, 20, 685-712, 2001.
[4] Piekutowski, A.J.: Formation and description of debris cloud produced by hypervelocity impact NASA contractor report 4707, 1996.

Meso-scale modeling of hypervelocity impact on spacecraft foam-core sandwich panels

Aleksandr Cherniaev[1]

[1]Department of Mechanical, Automotive and Materials Engineering, University of Windsor, 401 Sunset Ave., Windsor N9B 3P4, Canada

Abstract

Sandwich panels are widely used in the design of unmanned satellites and in addition to having a structural function, can often serve as orbital debris shielding. In this application, sandwich panels with open-cell foam cores have a significant advantage, enabling intensive interaction between the impactor fragments and the foam ligaments which enhances the fragments' breakdown and reduces their damaging potential. Modeling of this process requires an explicit meso-scale representation of the core material. In this study, realistic geometry models were obtained for 10 ppi and 20 ppi 8% aluminum open-cell foams using X-ray Computed Tomography imaging. They were then converted into LS-DYNA meshless simulation models and used in modelling of 6.9 km/s particles impact on foam core sandwich panels. Results of the numerical simulations were compared with experimental data reported by NASA in terms of the panels' core and rear facesheet damage.

Modeling of hypervelocity impact on spacecraft honeycomb-core sandwich panels: investigation of projectile shape and honeycomb-core effects

Reihaneh Aslebagh[1], Aleksandr Cherniaev[1]

[1] Department of Mechanical, Automotive and Materials Engineering, University of Windsor, 401 Sunset Ave., Windsor N9B 3P4, Canada

Abstract

Honeycomb-core sandwich structures are commonly utilized as orbital debris shielding in unmanned satellites. This study investigates the effects of projectile shape on the ballistic performance of aluminum honeycomb-core sandwich panels subjected to 7 km/s (hypervelocity) impacts at normal incidence. The shape of the reference projectile was a sphere, and other projectiles had a disk topology, including simple disks and disks with a central hole (ring projectiles), with different aspect ratios. To facilitate the investigation, a numerical simulation model was developed and verified against experimental data and predictions from an empirical ballistic limit equation. The verified model was then used to investigate hypervelocity impact scenarios involving different projectile shapes, honeycomb grades with different cell sizes, and different projectile/honeycomb cell alignments. It was found that of the shapes considered here, ring projectiles were of the highest concern: the volume of a 7 km/s ring projectile that could be resisted by a honeycomb-core sandwich panel without perforation of the rear facesheet was 1.65 times lower than that of a spherical projectile. In contrast, simulations with simple disks (without a central hole) did not show any significant change in the ballistic performance of the panel compared to impacts with a spherical projectile. Additional simulations conducted with ring projectiles demonstrated the strong influence of the honeycomb cell size and the projectile/honeycomb cell alignment on the damage to the rear facesheet of the panel.

Analysis of Partially Confined Blast Experiments and Simulations

Len Schwer

Schwer Engineering & Consulting Services

1 Abstract

Data and simulation results presented by Teland et al. (2018) for incident and reflected pressure histories indicated the simulation times of arrival and pressure magnitude did not agree well with the data for the reflected shock. They posited three possibilities for the differences: (1) Charge load (explosive mass per chamber volume), (2) Afterburning and (3) Variable Gamma for the gas mixture.

The present analysis dismisses charge load and afterburning as likely causes of the simulation result differences. The focus is placed on variable gamma as the most likely cause of the observed time differences. While additional evidence is presented in support of this possibility, the lack of a variable gamma equation of state computational model for the air and detonation product mixture limits certainty.

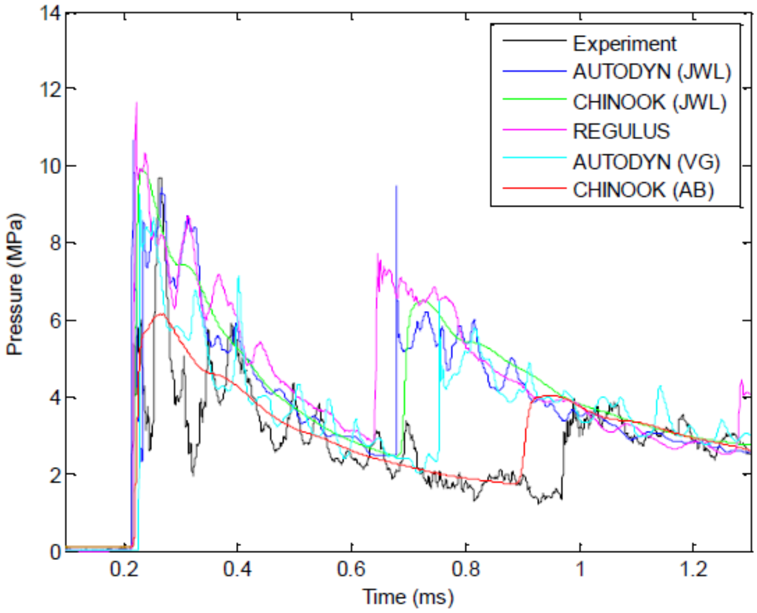


Figure 1 Pressure histories comparisons at gauge P2 for 82.5gram charge.

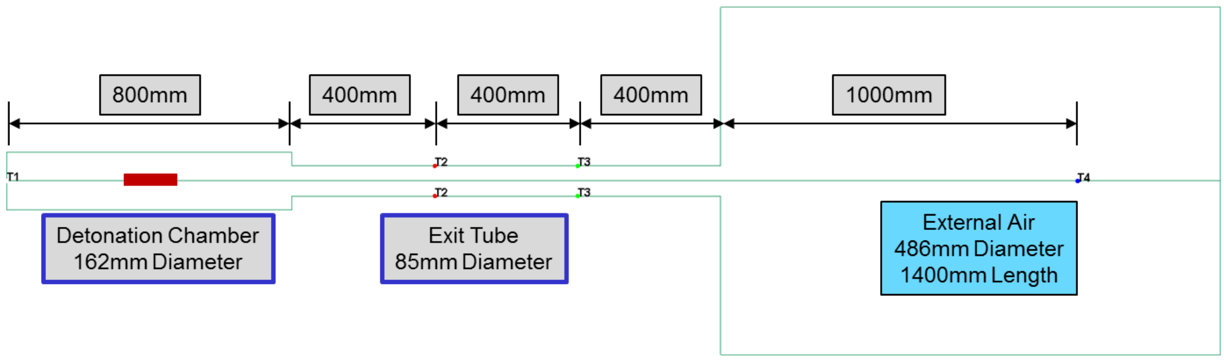


Figure 1 Schematic of test setup with location of tracer particles corresponding to pressure gauge locations.

Battery Simulation in the Crash Load Case

S. Rybak (EDAG)

Two Modelling Approaches of Lithium-Ion Pouch Cells for Simulating the Mechanical Behaviour Fast and Detailed

A. Schmid (TU Graz)

Modeling the Mechanical Behavior of a Li-Ion Pouch Cell under Three-Point Bending

Anja Altes, Benjamin Schaufelberger, Pascal Matura

Fraunhofer Institute for High-Speed Dynamics, Ernst-Mach-Institut, EMI, Ernst-Zermelo-Straße 4
79104 Freiburg, Germany

Abstract

Short-circuits caused by external forces, as they occur in crash situations, may lead to uncontrolled discharge of battery cells. As a consequence, the battery heats up locally, which, if it comes to the worst, results in an explosive reaction of the cell. However, the detection of critical deformations, for example in car crash simulations is very challenging: On the one hand, local indentations in the range of a few millimeters often result in a breakup of the inner structure and consequently in a short circuit. On the other hand, battery cells can also withstand surprisingly large deformations with the internal structure remaining intact. Thus, a reliable battery cell model has to capture a variety of different deformation modes.

Simulation models with different degrees of complexity are available for predicting and evaluating local indentations (e.g. [1]-[4]). Though, up to now more structure-dominated deformations of pouch cells like buckling or bending are not yet fully understood, what makes the derivation of sound modeling approaches for theses deformation scenarios especially challenging (e.g. [5]-[7]).

Within the present work a FE-model for simulating the bending behavior of a Li-ion pouch cell is developed based on the layered structure of the cell and validated against experimental results. Modeling assumptions necessary to consider the different involved length scales are discussed and the impact on the deformation behavior is evaluated. The developed numerical model provides a detailed insight into the cell behavior and may help to improve battery cell models in the future, to better predict their behavior in bending dominated deformation scenarios.

Acknowledgment

The research received funding from the State Ministry of Economic Affairs, Labor and Tourism Baden-Württemberg (Ministerium für Wirtschaft, Arbeit und Tourismus Baden-Württemberg) through the project BATTmobil (a cooperative project between Fraunhofer EMI and Fraunhofer IWM), grant number 3-4332.62-EMI/3. The project was part of the Tech Center i-protect.

Literature

[1] Greve, L.; Fehrenbach, C.: “Mechanical testing and macro-mechanical finite element simulation of the deformation, fracture, and short circuit initiation of cylindrical Lithium ion battery cells”, Journal of Power Sources 214, 2012, 377-385

[2] Sahraei, E.; Hill, R.; Wierzbicki, T.: “Calibration and finite element simulation of pouch lithium-ion batteries for mechanical integrity”, Journal of Power Sources 201, 2012, 307-321

[3] Sahraei, E.; Bosco, E.; Dixon, B.; Lai, B.: “Microscale failure mechanisms leading to internal short circuit in Li-ion batteries under complex loading scenarios”, Journal of Power Sources 319, 2016, 56-65

[4] Zhu, J.; Li, W.; Wierzbicki, T.; Xia, Y.; Harding, J.: “Deformation and failure of lithium-ion batteries treated as a discrete layered structure”, International Journal of Plasticity 121, 2019, 293-311

[5] Breitfuss, C.; Sinz, W.; Feist, F.; Gstrein, G. et al.: “A ‘Microscopic’ Structural Mechanics FE Model of a Lithium-Ion Pouch Cell for Quasi-Static Load Cases”, SAE Int. J. Passeng. Cars – Mech. Syst. 6(2), 2013

[6] Ali, M.Y.; Lai, W.-J.; Pan, J.: “Computational models for simulations of lithium-ion battery cells under constrained compression tests”, Journal of Power Sources 242, 2013, 325-340

[7] Lian, J.; Koch, M.; Li, W.; Wierzbicki, T.; Zhu, J.: “Mechanical Deformation of Lithium-Ion Pouch Cells under in-plane Loads – Part II: Computational Modeling”, Journal of the Electrochemical Society 167, 2020

Simplified modeling of pouch cells under different loadings

Andreas Trondl[1], Dong-Zhi Sun[1], Silke Sommer[1]

[1]Fraunhofer Institute for Mechanics of Materials IWM, Wöhlerstraße 11, 79108 Freiburg, Germany
(contact: andreas.trondl@iwm.fraunhofer.de)

Abstract

Due to increasing requirement on the reduction of CO2-Emissions, the meaning of E-Mobility becomes more and more important. The related development of efficient Li-ions with high charge densities has also a direct impact on the automotive industry. This applies in particular to the crash safety of Li-ion-battery-powered vehicles. The structure of Li-ion batteries is in principle a repetitive layered system. One cell unit consists of two very thin metal foils (typically Al and Cu), which coated with active materials (lithium-metal-oxide and graphite). These coated layers represent the anode and the cathode of one unit-cell which are separated by thin polymeric films (separator-foils). A typical pouch cell consists of hundred and more layers which are embedded in an electrolyte within a bag (pouch, coffee bag). Local deformations resulting from uncontrolled crash loads can lead to critical damage in the separator, which causes an intrinsic short circuit and subsequently an unstable state (thermal runaway) that can result in battery explosions. Therefore, the simulations of intrinsic mechanical deformation states are crucial to predict electrical short circuits. However, in a full crash simulation of an electrical vehicle it is impossible to model the intrinsic layered system of each pouch cell, because there are hundreds of cells installed and each cell consists of more than hundred layers. Due to these facts, it is necessary to develop and apply simplified models, which use a homogenization approach based on the strong repetitive intrinsic layered cell structure [1]. For this modelling approach, anisotropic compressible plasticity models should be well suited, because it is to be expected, that the deformation behavior of the cells is different in the layered thickness direction in relation to the in-plane loadings of the parallel layers. In the presented work, different anisotropic and isotropic plasticity models in LS-Dyna were used to describe the mechanical behavior of pouch cells in a simplified manner and were compared to each other. The calculated results are compared to experimental investigations of representative crash loading scenarios, e.g. bending, indentation [2], intrusion and compression in different cell directions. Furthermore, the evaluated intrinsic mechanical quantities (volumetric strain) are presented to investigate the possibility of formulating a mechanical based electrical short circuit criterion for all experimental tested loading cases. The result suggests the conclusion for a development of a user defined material model to simulate the whole testing regime consistent with the experimental observations for the mechanical and electrical behavior of the pouch cell.

Acknowledgement

The research received funding from the State Ministry of Baden-Wuerttemberg for Economic Affairs, Labour and Housing Construction through the project BATTmobil (a cooperative project between Fraunhofer EMI and Fraunhofer IWM), grant number 3-4332.62-EMI/3. The project was part of the Tech Center i-protect.

Literature

[1] Greve, L.; Fehrenbach, C.: “Mechanical testing and macro-mechanical finite element simulation of the deformation, fracture, and short circuit initiation of cylindrical Lithium ion battery cells”, Journal of Power Sources 214, 2012, 377-385

[2] Zhu, J.; Li, W.; Wierzbicki, T.; Xia, Y.; Harding, J.: “Deformation and failure of lithium-ion batteries treated as a discrete layered structure”, International Journal of Plasticity 121, 2019, 293-311

An Integrated Modeling Scheme for Sensor Embedded Woven Composite Structures in Manufacturing Simulation

T. Usta, C. Liebold (DYNAmore), M. Vinot (DLR)

Axial Crushing of an Aluminum-CFRP Hybrid Component: FE-Modelling, Simulation and Experimental Validation

Sheikh Enamul Hoque[1], Alexander Rauscher[2], Matthias Hartmann[1]

[1] AIT Austrian Institute of Technology GmbH, Ranshofen, Austria
[2] University of Applied Sciences Upper Austria, Wels, Austria

Abstract

The crushing behavior of an aluminum-CFRP hybrid generic crash component under axial crushing load was experimentally investigated. Parallelly, finite element simulations of the crush tests were carried out in LS-DYNA. The extended 3-parameter Barlat model (MAT36E) was used to characterize the anisotropic elastic-plastic behavior aluminum sheet. The Chang-Chang failure criteria (MAT54) was used to characterize the fracture behavior of the CFRP laminate. Both material models were calibrated based on coupon tests. The aluminum-CFRP interface was modeled using tied contact with cohesive mixed mode failure criteria to capture the delamination behavior. Good agreement was found between experiment and simulation in terms of Specific Energy Absorption (SEA) as well as deformation behavior.

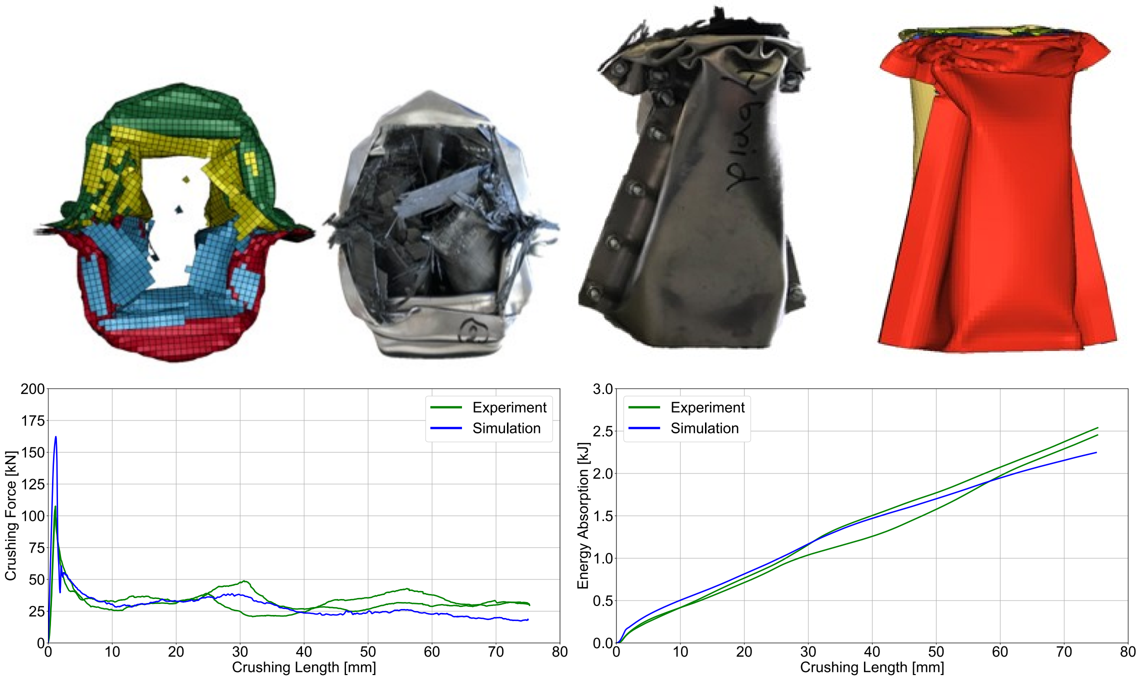


Figure: Comparison between experiment and simulation.

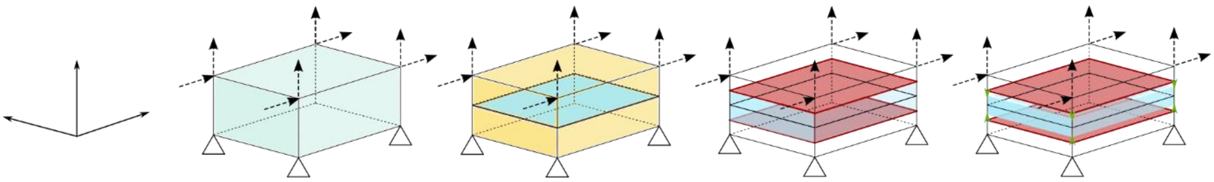
Delamination and Fracture Modeling Techniques for Shell Composite Structures in LS-DYNA®

Alessandro Polla[1], Enrico Cestino[1], Giacomo Frulla[1], Paolo Piana

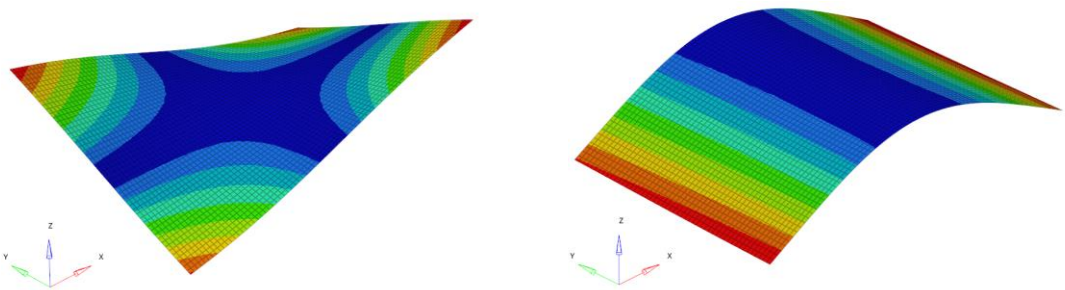
[1]Department of Mechanical and Aerospace Engineering (DIMEAS), Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129 Torino, Italy
Email: alessandro.polla@polito.it

1 Abstract

Over the last decade, increased demands of laminated composite in advanced structural applications have pushed forward the necessity to develop a broad range of modelling strategies. Cohesive zone modelling (CZM) is a powerful technique useful to characterize and reproduce the interlaminar & intralaminar behaviour of composite structures when subjected to static and dynamic loads. CZM efficiently replicates phenomenologies such as debonding, fracture propagation and delaminations between adherents or inside laminated components. Their use within numerical models is still limited due to the lack of guidelines about their appropriate implementation. Today industrial practices push forward the application of shell meshes to minimize the computational cost and to improve the realization of full and complex assemblies. For these purposes, the adoption of CZM with shell elements needs model techniques that allow the definition of the correct strength and stiffness comparable to the physical properties of the real specimen. Several approaches to predicting and simulate the initiation and propagation of delamination in shell composite laminates have been investigated. An appropriate methodology to establish and evaluate the cohesive constitutive stiffness parameters for the characterization of progressive delamination with shell adherents is proposed. Different shell-cohesive models were evaluated to outline the suitable methodology that defines the correct compliance and strength of a composite structure. Different practices were being analyzed with recent explicit and implicit LS-DYNA solvers. Particularly, modal analysis and quasi-static loading conditions were performed to evaluate and verify that the cohesive stiffness inserted in the model effectively describes the right compliance of the structural assembly. Furthermore, the results obtained and the procedures detailed were inspected through dynamic standard problems like DCB, ENF and LVI simulations to ensure that the refined proceeding describes the physical dynamical behaviour of the entire assembly. The methodology detailed tries to solve specific problems that arise with the adoption of cohesive elements in shell structure and support the application of CZM for the definition of damageable assembly. The techniques detailed demonstrates that shell-cohesive models with the correct compliance functionally reduce the computational cost associated and ensure accuracy of the results.



Picture 1 - Schematic representation of basic shell-cohesive methodology evaluated



Picture 2 - Modal comparison between standard FEM Software and LS-DYNA shell-cohesive models: First & Second structural Modes representation

2 Literature

[1] A. Turon, C.G. Dávila, P.P. Camanho, J. Costa: " An engineering solution for mesh size effects in the simulation of delamination using cohesive zone models", Engineering Fracture Mechanics, 2007, 1665:1682-74

[2] S.A. Muflahi, G. Mohamed, S.R. Hallett: " Investigation of Delamination Modeling Capabilities for Thin Composite Structures in LS-DYNA", 13th International LS-DYNA Users Conference, 2014

[3] Devon C. Hartlen*, John Montesano, Duane S. Cronin: " Cohesive Zone Modeling of Adhesively Bonded Interfaces: The Effect of Adherend Geometry, Element Selection, and Loading Condition", 16th International LS-DYNA® Users Conference, 2020

[4] Paul W. Harper, Stephen R. Hallett: " Cohesive zone length in numerical simulations of composite delamination", Engineering Fracture Mechanics, 2008, 4774:4792-75

Laser impact modelling in order to assess composites bonding on aeronautical structures

Charlotte Michel[1], Vincent Lapoujade[1], Teddy Maillot[1], Jérémie Grassy[1]

[1]DynaS+, 5 avenue Didier Daurat 31400 TOULOUSE France, c.michel@dynasplus.com

Abstract

Massively used in aeronautical structures, composites are nowadays essential in the search for a more ecological and successful industry. Their low density enables weight reduction and then decreases airplanes consumption. However, the current composites assembly process represents a limitation in their use. In fact, we do not have any reliable, industrialized and non-destructive technology to control the adhesive quality. Then composites are also riveted which adds weight and drilling process during which fibres can be locally damaged. For about 10 years, the LASAT (Laser adhesion test) technology appears to be a promising alternative. The laser impact creates a plasma that induces shock waves propagation in the structure. If well calibrated the laser does not damage the structure as long as the glue respect its nominal strength. The LASAT technology can also be used to generate damage anywhere in the assembly thickness. The experimental technology is mature but is lacking a numerical tool so to calibrate the input laser parameters depending on the targeted results.

DynaS+ is working on the VANESSES project, funded by the French Ministry of Defence, in order to:

- Create reliable and validated numerical models representing laser impacts and shock waves propagation on specific assemblies,
- Develop an automatized and numerical calibration tool to determine laser platforms input parameters depending on applications objectives (geometry, materials, targeted stress state)

This paper focuses on the numerical approach of this project and how LS-DYNA can be used to represent such phenomena. Analysis of laser induced shock waves propagation in representative structures will be presented in different configurations.

Benchmark as decision support for cloudification: Moving CAE and HPC to the cloud increases quality and efficiency of simulations

Christopher Woll
GNS Systems GmbH

1 Summary

A decision in favor of the cloud and the associated organizational changes are of central strategic importance for automotive OEMs and their suppliers. In advance, it is important to clarify whether existing organizational structures and processes can actually be optimized with the help of the cloud. To this end, we regularly carry out benchmarks for well-known customers as a decision-making aid. In the presentation, we will present a benchmark with real productive jobs based on existing data in the customer's specific environment. Mapping the workflow with selected CAE applications in the cloud provided a realistic evaluation basis for their benefits and costs. In addition to comparing runtimes and results of selected benchmark jobs, our experts tested the quality of remote work in a workstation scenario for selected finite element analysis (FEA).

2 Increased performance for simulations in the cloud

The benchmark on the High Performance Computing (HPC) reference architecture was based entirely on a cloud environment from GNS Systems. Within the available subscription, separate virtual networks were created for the scalable HPC cluster, the license server and the rest of the infrastructure. The core was formed by an orchestration system and a master node. Necessary machines for the simulation jobs started flexibly via predefined images and so-called "scale sets". Demand was determined by the number and size of jobs in the queue. Unneeded nodes were shut down to optimize costs.

The benchmark results demonstrate the high efficiency for simulations in the cloud. Engineers also achieve higher quality simulation results with better computing power. The simulation jobs benefit from the increased number of CPU cores per node and the demand-oriented scaling of computing performance. The implementation was functional and error-free with a selected toolset. The engineers experienced the interactive work sessions on remote desktops with Linux and Windows as smooth and trouble-free. The benchmark clearly proves that performance on modern CPUs in the cloud does not lag behind conventional on-premises solutions. The results for this specific business case provided a reliable strategic decision-making basis for the cloudification of HPC and CAE processes.

Virtual product development in the Digital Engineering Center: Greater innovative capacity through interdisciplinary organisation and automation

Christopher Woll
GNS Systems GmbH

1 Summary

While the traditional development cycle "design-build-test" often lasted several years, today it is a matter of bringing innovations to market readiness with simulation-driven design and digital twin in the shortest possible time. Virtual tests on vehicles make a significant impact in effectively reducing development times and costs. Companies must master two fundamental challenges for their efficient use: on the one hand, complex and time-intensive engineering tasks have to be mastered quickly. On the other hand, solutions are increasingly required that organise the complex product development across units. The Digital Engineering Centre, an engineering workplace in the cloud, solves these challenges.

2 Constant environment for virtual product development

HPC cloud solutions meet the demand for massive computing power quickly and reliably. However, on-demand access to " unlimited" resources in the cloud is often both a curse and a blessing. With cloud deployment, positive effects such as pay-per-use, reduced capital expenditure (CAPEX), greater business agility and higher quality results can be easily realised. Using efficient crash analysis tools like LS-DYNA in the cloud, on the other hand, is not trivial. Major bottlenecks are hidden in the planning as well as administrative activities - especially the lack of compatibility with existing IT structures and workflows.

Through its platform architecture, the Digital Engineering Centre creates suitable interfaces to relevant applications in virtual product development such as LS-DYNA, pre- and post-processing tools and data management. As a continuous development environment, the Engineering Workplace ensures strong flexibility, low costs and high reliability in interdisciplinary product development. With the support of the platform, engineers collect simulation data, process it in a structured manner and evaluate it as required with the help of suitable AI tools. User-friendly workflows for each user group make simulation-based product development a reality in a few simple steps. This is possible through a complete automation of the processes and the convenient integration of cloud-based or hybrid IT infrastructures into the entire development cycle. In the presentation, the experts from GNS Systems will show how the Engineering Workplace supports engineers with smart workflows and optimal computing performance to significantly accelerate simulation-based product development.

Transitioning LS-DYNA workloads to the Cloud on the path to Digital Maturity

Daniel Dorribo[1], Iago Fernández[1]

[1]Gompute, Gridcore AB (Gothenburg, Sweden)

1 Abstract

Industries worldwide are going through a digitalization process towards "Industry 4.0," where cloud resources play a key role. Forcing a transition for CAE engineers from traditional, in-house HPC to more flexible solutions in the cloud.

On the journey to digital maturity there are different dimensions to consider when transitioning a CAE team to a permanent cloud solution. The needs and requirements of different industries are not always covered with a single solution. What is required is an analysis of the different layers involved (IT/Network, Licensing, Security, Engineering).

Firstly, the Gompute team will cover an overview of the different options available in the market, and how those can be deployed for LS-DYNA users based on their needs. Secondly, we outline the most important dimensions that any corporation will require on this transition: Performance and Security. Gompute will show the scalability results of LS-DYNA on different hardware types, and the most common set-ups to ensure data protection both from on-premise and home offices.

LS-DYNA Extend in the Cloud

M. Schenke (DYNAmore)

A Meta-model based Approach to implement Variation Simulation for Sheet Metal Parts using Mesh Morphing Method

Hanchen Zheng[1], Kushagra Upadhyay[1], Frank Litwa[1], Kristin Paetzold[2]

[1]Mercedes-Benz AG, Sindelfingen, Germany
[2]Bundeswehr University Munich, Neubiberg, Germany

1 Abstract

The virtual process chain is an essential step for the sustainable digital transformation in the manufacturing industry. For the Body-In-White (BIW) sheet metal parts, the manufacturing joining simulation based on finite element method is used to simulate the joining processes in the body shop. The target is to predict the dimensional accuracy of assemblies after using different types of joining technologies. However, the assembly deviation is not only affected by the joining operations, but also by the initial part deviations. Therefore, an integration of geometrical variations in the joining simulation model is necessary to improve the prediction accuracy. The statistical analysis for the geometrical variations also enhances the applicability of the joining simulation, for example in the tolerance analysis. In this paper, a meta-model based approach is developed to implement the variation simulation for the sheet metal parts. The geometrical variations of parts are governed by the tolerance specification and modelled through a mesh morphing method. The nominal FE mesh of the toleranced surface is morphed by using scattered data interpolation followed by direct stiffness method. Joining simulations are implemented with the morphed meshes to build up the meta-model. Afterwards, Monte Carlo (MC) simulations are applied for an efficient statistical analysis. The proposed approach is illustrated along with a prototype part in the body in white. LS-Opt is used to build up the simulation model and LS-Dyna is used to implement the joining simulations. The simulation results are compared with the tolerance simulation results as well as the measurement data.
The proposed simulation model integrates the geometrical variations in the joining simulation model. It enables a statistical tolerance analysis considering the influence of manufacturing process in an early development stage. The sensitivity analysis helps the user to identify the importance of part tolerances, which contribute to the optimization of the product and manufacturing process.

Keywords: Manufacturing joining simulation, variation modelling, tolerance analysis, meta-model based analysis

From Time Delayed MRI to Patient-specific computational modeling of scar-related ventricular Tachycardia

Karim El Houari[1], Pierre L'Eplattenier[2], Clémentine Shao[1], Iñaki Caldichoury[2], Sophie Collin[1], Xavi Planes[3], Martin Steghöfer[3], Rosa M. Figueras[3], Luis Serra[3] and Michel Rochette[1].
[1]Ansys - 35-37 Rue Louis Guérin, 69100 Villeurbanne, France
[2]Ansys - Livermore Software Technology LLC, 7374 Las Positas Road, Livermore, CA 94551, USA
[3]ADAS3D MEDICAL SL - Carrer de París, 179, 08036 Barcelona, Spain

Abstract: Sudden cardiac death commonly occurs due to heart rhythm disorders called arrhythmia. Although recognized as the most efficient treatment options, Cardioverter Defibrillator implantation and tissue ablation are still not used to their full potential. Recently, advances in computational modeling and the increasing use of imaging tools have proven that patients' digital twins can play a role in addressing these limitations. This paper presents such an approach using the industrial software ADAS-3D and LS-DYNA. The workflow starts from Late Gadolinium Enhanced-Magnetic Resonance Imaging (LGE-MRI) data from a patient with structural heart disease. The left ventricle and fibrotic substrate were analyzed using ADAS-3D software, which enables to distinguish between tissue that is healthy, scarred, and intermediate, and to extract topological information. This segmentation and tissue classification are used to build, using LS-DYNA, a detailed electrophysiology model containing the relevant features for simulating arrythmia. Using LS-DYNA, this model is then used to simulate a normal heartbeat and a clinical pacing protocol for inducing arrhythmia.

Sideways launching process of a ship using the Arbitrary-Lagrangian-Eulerian approach

Albert Ulbertus[1], Martin Schöttelndreyer[1], Sören Ehlers[2]

[1]thyssenkrupp Marine Systems GmbH, Operating Unit Surface Vessels, 20457 Hamburg, Germany
[2]Hamburg University of Technology, Institute for Ship Structural Design and Analysis, 21073 Hamburg, Germany

Abstract

The launching process of ship is always a critical event during its construction. Especially a sideways launching process can be challenging. Besides high loads on the ship's hull structure at the impact with the water surface, the stability has to be checked carefully to prevent capsizing of the ship. In the past two decades, simulation techniques utilizing fluid-structure interaction (FSI) were continuously advanced to a point, where even complex systems with sophisticated simulation models can be simulated in a feasible timeframe.

However, for simulating such a sideways launching process different complex phenomena are to be covered by a suitable FSI-approach. The non-linear response of the ship (complete stability range) is relevant. A free 6-DOF movement of the ship with according equation of motions including added masses and hydrodynamic damping is given. Further, free water surfaces with spray have to be considered. Regarding the ship's hull structure the resulting dynamic loads at the impact of the ship hull with the water surface are important. High stresses at highly loaded areas must be captured correctly for assessing non-linear behavior of the hull structure if necessary (buckling or even plastic deformation).

Within this paper, the sideways launching process of a special purpose ship is investigated using the Arbitrary-Lagrangian-Eulerian (ALE) approach in LS-DYNA®. The focus lies on the verification of the ALE-approach for this intended use case and the different underlying phenomena. The verification is based primarily on model tests of the sideways launching process carried out at a ship model test facility. In addition to the movement of the ship, the pressure at different points of the ship hull was measured especially for verification purposes.

Furthermore, a parametric study regarding the different settings of the card *CONSTRAINED_ LAGRANGE_IN_SOLID is presented. Different settings of the FSI-algorithm are varied and the influence regarding the results (movement as wells as loads at impact) are evaluated. Moreover, different approaches for decreasing computational time are investigated and discussed.

At last an outlook is presented, how the ALE-approach is used to assess the resulting loads within the ship's hull structure during the sideways launching process of the special purpose ship.

Multiphysics SPH simulation of flow drilling process

Anthony Journaux[1], Tess Legaud[1], Vincent Lapoujade[1]

[1]DynaS+, 5 avenue Didier Daurat 31400 TOULOUSE France, a.journaux@dynasplus.com

Abstract

Flow drilling is an alternative drilling solution for metal plates up to several centimetres. Using a conical tool, the process combines high rotation speed and high pressure to initiate friction and heat up the plate material locally in contact with the tool. As a consequence, the heated material has its mechanical characteristics reduced and is subjected to very large plastic deformation. The surplus of matter is not wasted but is shaped into a collar above the metal plate and a socket below. These bulges induce a local additional thickness enabling a direct threading without added parts (bolt...).

Flow drilling technology has emerged since 1923 but has been little used due to the inherent difficulty to accurately predict the collar size and extent. The process itself is mature and used sparsely in the industry but is lacking a numerical tool able to simulate accurately the thermal and mechanical effects in order to ensure a reliable procedure.

Flow drilling is by nature a multiphysics process which also leads to locally high deformation. Such complex phenomena can be addressed by LS-DYNA which offers the possibility to couple Multiphysics solvers and the use of SPH method to deal with large deformation in continuous material.

The work undertaken and presented in this paper follows three steps of increasing complexity:
- Build a SPH model of the process using a purely mechanical solver with a pseudo-thermal model using steel materials with an idealized thermal gradient
- Evaluate the compatibility between SPH modelling and the LS DYNA thermal solver and ensure the friction, dilatation and diffusion works properly together with a SPH model.
- Set up a coupled mechanical-thermal model able to simulate the complete process accurately

Applications of the new magnetostatic solver/ AMS preconditioner in LS-DYNA®

Miro Duhovic[1], Inaki Caldichoury[2], Pierre L'Eplattenier[2], Trang Nguyen[2], Joachim Hausmann[1], Lars Kielhorn[3], Thomas Rüberg[3], Jürgen Zechner[3]

[1]Leibniz-Institut für Verbundwerkstoffe GmbH, Technische Universität Kaiserslautern, Erwin-Schrödinger-Straße 58, 67663 Kaiserslautern, Germany
[2]Ansys - Livermore Software Technology LLC, 7374 Las Positas Road, Livermore, CA 94551, USA
[3]Tailsit, Nikolaipl. 4, 8020 Graz, Austria

Abstract

Previous implementations of LS-DYNA's Electromagnetic (EM) module have relied on a decoupled FEM-BEM solve based on a Richardson method, requiring small time-step sizes and therefore long simulation times. Recently, a new Monolithic FEM+BEM solver, along with an auxiliary-space Maxwell solver (AMS) preconditioner for magnetostatic (small or zero electrical conductivity) has been developed in LS-DYNA®. Unlike the current implementation, the new solver is unconditionally stable with respect to the time-step size and allows for the handling of materials with high permeability and low electrical conductivity. In this paper, the capabilities of the new solver is tested on the use-case of carbon fiber reinforced thermoplastic (CFRTP) laminate induction heating using magnetic flux concentrators/field formers (see Fig. 1). A flux concentrator is a ferrous material, with a very high magnetic permeability and very low electrical conductivity, used to direct or intensify the magnetic flux of an induction coil towards a desired location. Such materials are typically used in induction heating systems in order to improve energy efficiency of the process. In the current work, induction heating characterization experiments were performed on carbon fiber poly(ether ether ketone) (CF/PEEK) laminates using two different coil geometries and close to constant induction generator settings (power, current, voltage and frequency). The testing program involved single plate specimens 100 x 100 x 2.1 mm in size. Temperature measurements were recorded using two laser guided non-contact pyrometers on both sides of the plate specimens at specific locations depending on the coil geometry. In addition, full infrared thermal imagery was recorded for the non-coil side of the specimen. The overall temperature information was captured at 50Hz for both the heating and cooling cycles and was the same for both the pyrometers and thermal camera equipment. The influence of the coil's magnetic flux concentrators was also studied in order to see if the effect of magnetic flux concentration could be simulated in LS-DYNA®. Two different shaped induction coils (a figure eight and a single turn square section coil) with and without magnetic flux concentrators have been used to demonstrate the enhanced functionality of the new solver.

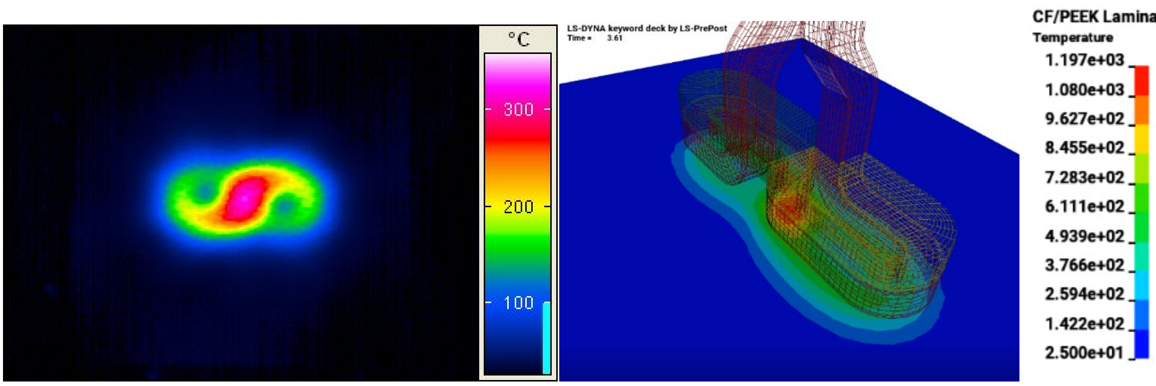


Fig.1: Infrared thermal image of induction heating experiment (left) and simulation with the magnetostatic solver considering magnetic flux concentrators (right).

Smoothed Particle Hydrodynamics Modeling of Granular Column Collapse

Yucheng Li[1], Ningning Zhang[1], Raul Fuentes[1]

[1]Institute of Geomechanics and Underground Technology, RWTH Aachen University, Germany

ABSTRACT

Granular column collapse is a commonly studied granular flow problem, where an initially cylindrical column of dry granular materials collapses onto a flat surface under gravity. In this study, the meshless method Smoothed Particle Hydrodynamics (SPH) is used to model this phenomenon examining in particular the effect of aspect ratio, defined as the ratio of the initial height h0 and radius r0 of granular column. The numerical results are consistent with experimental results in terms of three aspects: (1) description of flow shapes; (2) runout distance and (3) final deposit height. Further observations and measurements are obtained to explore the collapse.

KEYWORDS: granular column collapse, aspect ratio, Smoothed Particle Hydrodynamics

Improvement in predictive capability of small overlap crash simulation with emphasis on GISSMO material model, weld rupture and detailed modeling

Janaki Sholingar[1], Eric Stahmer[1], Anantharam Belur Sheshadri[1], Milind Shivaji Parab[1]

[1] FCA US LLC, Auburn Hills, Michigan USA

Abstract

CAE tools are one of the best techniques in the auto industry to drive design and help product development with minimal physical tests. Physical tests are very time consuming and expensive which is driving the Auto industry towards virtual simulations to replace physical tests. CAE has become an integral part of product development to accurately predict physical testing and drive design direction. For CAE to accurately predict the physical test, it depends on details captured in the full vehicle model. In the small overlap load case it's necessary to capture as much detail as possible for components engaged during the impact event. However, capturing too much detail leads to prohibitively large models with excessive computational time. So it is important to understand the load path to decide the critical vehicle components which play a vital role in the crash event. This includes the sheet steel/aluminum stamped parts, aluminum extrusion and also the fasteners and welds. In this paper an attempt is made to revisit the modeling of these critical vehicle components and later confirm the performance with respect to the physical test. The sheet steel/aluminum stamped parts and also the aluminum extrusions are finely meshed and GISSMO material models are implemented to define their rupture. The fasteners (bolts) are modeled using solid elements. Spot welds are modeled as solid nuggets with damage material model MAT_SPOTWELD_DIAMLERCHRYSLER and a simple elegant technique is used to define the aluminum MIG welds. The MIG welds are joining thick Aluminum parts in the cradle. MIG welds are represented by discrete beams with MAT119 material model. The stiffness, loads and rupture displacement parameters are adjusted to component tests and an envelope of rupture is created. This is carried on to the full vehicle as a predictive model and the designs are iterated. All of the above modeling methods and techniques helped to accurately predict velocities, intrusion, wheel kinematics and a good correlation to the physical test was achieved.

Using JFOLD and LS-DYNA to Study the Effects of Passenger Airbag Folding on Occupant Injury

Richard Taylor[1], Shinya Hayashi[2], Mayumi Murase[2]

[1]Arup
[2]JSOL Corporation

1 Abstract

JFOLD is a software tool for simulation-based airbag folding in LS-DYNA®. This paper presents how JFOLD and LS-DYNA can be used effectively to research how slight changes in automotive passenger airbag folding can lead to significant changes in occupant injury prediction.

The demands placed on today's occupant safety teams continue to increase, driving up the need for airbag complexity and simulation accuracy whist driving down the time to deliver. Accurate airbag simulation is critical to improve occupant safety in an increasing number of crash scenarios and out-of-position cases, including passengers of autonomous vehicles. In addition, airbag simulation is now being used to assess the performance of interior trim components during early break-out and deployment.

However, accurately simulating the deployment of frontal passenger airbags (PABs) is still a challenging task for many involved in automotive safety. Not only is it hard to replicate the flattened 3D shape and hard to simulate the complex gas dynamics during deployment, but there is also a degree of natural variation in the fold pattern of real airbags due to manual folding processes which can lead to a variation in deployment behaviour and occupant injury. In addition, even well-made computer models of PABs can demonstrate sensitivities to initial conditions due to the high complexity of simulating gas flow though folded fabric.

As part of ongoing airbag simulation research at JSOL Corporation in Japan we present some recent work into the detailed modelling, folding and deployment simulation of a production passenger airbag, to share our solutions to some of the above challenges. We demonstrate how JFOLD's flow-chart folding process can quickly generate several variations of fold pattern and investigate the effect on occupant injury in commonly used loadcases. The latest features of JFOLD and airbag simulation in LS-DYNA will also be discussed.

Reconstruction of Trimmed and Faceted Vehicle Models for Isogeometric Analysis in LS-DYNA

Kendrick M. Shepherd[1], Xianfeng David Gu[2], Thomas J. R. Hughes[3]

[1]Department of Civil and Environmental Engineering, Brigham Young University, Engineering Building 430, Provo, UT, 84602, USA
[2]Department of Computer Science, Stony Brook University, New Computer Science Building, Stony Brook, NY, 11794-242, USA
[3]Oden Institute for Computational Engineering and Sciences, The University of Texas at Austin, 201 East 24th Street, C0200, Austin, TX, 78712-1229, USA

1 Abstract

In this work, a theory-based computational framework is presented that is used to rebuild two industrial vehicles—a trimmed spline representation of the US Army's Unclassified DEVCOM Generic Hull [1] and parts of the body-in-white of a mixed finite element model of a 1996 Dodge Neon from the National Crash Analysis Center of George Washington University (GWU) [2]—into sets of trim-free analysis-suitable spline patches. The framework is implemented to interact directly with existing CAD and analysis software: a CAD interface is developed in Rhinoceros 3D, from which a keyword file is created for direct input into LS-DYNA for isogeometric analysis (IGA).

To date, a variety of techniques have been proposed to rebuild trimmed models into watertight splines. Many of these employ expensive optimization techniques or use spline definitions that are not supported by most commercial analysis packages, including LS-DYNA. In this work, a surface is rebuilt using discrete surface Ricci flow to compute a flat metric with cone singularities [3], after which a cut version of the surface is immersed into the plane and minimized against a quadrilateral layout-inducing energy [4]. A quadrilateral layout on the surface is then extracted and the geometry is rebuilt into a watertight, boundary-aligned set of NURBS patches. This technique is both grounded in theory and robust, as demonstrated through its application to industrially relevant geometries.

The work culminates by reperforming GWU's frontal crash analysis in LS-DYNA using the original LS-DYNA keyword files of [2] with certain piecewise (bi)-linear finite element parts of the vehicle substituted out for high-continuity isogeometric spline patches. This demonstrates that the framework can be used to not only rebuild and analyze CAD models, but that it can also be used to reevaluate existing finite element models with the high-order accuracy and smoothness of isogeometric techniques.

These tools bring a new level of semi-automation for constructing and reconstructing isogeometric-analysis suitable spline spaces for use in LS-DYNA. Ultimately, they are a step towards the realization of IGA's goal: a streamlined design through analysis process.

2 Literature

[1] Dooge, D., Dwarampudi, R., Schaffner, G., Miller, A., Thyagarajan, R., Madanmohan, V., Babu V.: "Evolution of occupant survivability simulation framework using FEM-SPH coupling," 2011 NDIA Ground Vehicle Systems Engineering and Technology Symposium (GVSETS), 2011, DTIC Report #ADA547566, TARDEC Registration #22044.

[2] George Washington University, "Finite element model of a Dodge Neon," Technical Report of George Washington University's Virginia Campus, 2006.

[3] Chen, W., Zheng, X., Ke, J., Lei, N., Luo, Z., Gu, X.: "Quadrilateral mesh generation I: Metric based method," Computer Methods in Applied Mechanics and Engineering, Vol. 356, 2019, 652-688.

[4] Shepherd, K.M., Gu, X., Hughes, T.J.R.: "Isogeometric Model Construction of Open Shells Via Ricci Flow and Quadrilateral Layout-Inducing Energies," Submitted to Engineering Structures.

Numerical investigation of the flow through foldcores with LSDyna ICFD Solver

Fabian Muhs[1], Ralf Walter[1]

[1]Institute of Aircraft Design, University of Stuttgart

1 Abstract

Foldcores used as sandwich core material offer a promising alternative compared to conventionally used honeycombs or foams. The production of the foldcores is inspired by the technique of origami and generates a three-dimensional core structure by folding a flat semi-finished product. Classical as well as technical papers, metal or plastic foils but also fiber composite semi-finished products can be used as starting material. The geometric design of the foldcores offers an almost infinite design space, which is only limited by the mathematical rules of folding. [1]
The decisive advantage of foldcores over classical core types is the generation of cores with an open structure, as shown in Fig. 1. This results in numerous possibilities for functional integration, such as the insertion of power lines or pipes for media transportation, the storage of media within the cavities, and the equipping of the core with active sensor technology. In addition to these functionalities, the channels within the foldcore can also be used directly for media guidance and thus heated/cooled sandwich elements can be realized. This aspect is investigated in more detail in the present work using the flow solver ICFD. For this purpose, different foldcore geometries are numerically evaluated and characteristic quantities such as the pressure loss are determined. The results are compared with the established flow solver OpenFoam as well as experimental data.
After successful validation of the simulation results, application-related problems of heat transfer are considered. This is done on the one hand by activating the energy equation but also by using conjugate heat transfers. The basic suitability of foldcores as heat exchangers is investigated and the potential of different foldcore geometries is evaluated.

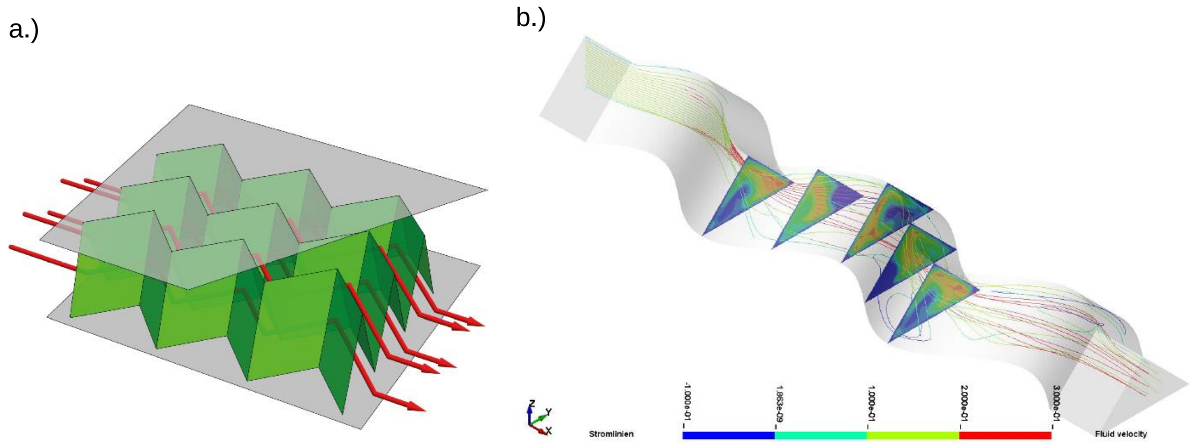


Fig.1: a.) Sandwich component with open foldcore structure. b.) Results of numerical flow simulation on a foldcore channel [2].

2 Literature

[1] Muhs, F., Thissen, S., and Middendorf, P.: „Virtual process chain for optimization of sandwich foldcores under flatwise compression “, Thin-Walled Structures 157, 2020.

[2] Walter, R.: "Numerische Untersuchung der Durchströmung von Faltkernen mit LS-Dyna", Master Thesis, Institut für Flugzeugbau, 2021

Impingement jet flows for cooling using LS-DYNA®: an introduction to ISPH and ICFD approaches

Edouard Yreux[1], Iñaki Çaldichoury[2]

[1]Ansys
[2]Ansys

Abstract

Cooling jet flows are commonly encountered in many industrial applications where fast and strong heat dissipation is required such as in pistons, gears, electrical engines and so forth. With the rapid growth and acceptance of simulation as a companion tool intervening directly in the design process, there is a need to provide fast and robust numerical solutions that can provide information on flow patterns and cooling efficiency. LS-DYNA includes several solvers capable of representing fluids and solve Multiphysics simulations. This presentation will focus on the ICFD solver and the ISPH solver. The ICFD solver is a finite element incompressible CFD solver, which includes a robust and accurate monolithic approach for conjugate heat transfer applications. The ISPH solver is based on LS-DYNA's SPH solver which has been recently extended to handle incompressible flows for fluid injection, wading and similar splashing applications. Heat transfer coefficient can be displayed on wetted surfaces thanks to the use of empirical laws based on flow patterns. In the presentation, the two solvers will be introduced. A Jet impingement model where an oblique jet impacts a hot wall and causes cooling to occur will be used as reference to compare and discuss the two approaches. Numerical techniques and workflows will also be described.

Trailing edge failure analysis of a friction pad in a clutch using thermal fluid structure interaction with LS-DYNA ICFD solver

Amit Nair[1], Inaki Caldichoury[1]

[1]Ansys

Abstract

A Clutch is a mechanical link used to transmit torque from engine to transmission and typically rotate at very high RPMs. They continuously engage with friction pads to transmit power for motion and only disengage when a gear ratio change is required. During this process of engaging and disengaing the clutch go from stationary to moving instantaneosly. A combination of friction pads and disks are used to transmit the power. There is significant increase in temperature due to friction between the pads and plate at transition and during rotation. This temperature increase leads to thermal expansion of parts and can cause uneven shape changes. This deformation leads to increase in frictional energy and eventual rise in heat generation. Friction and temperature along with pressure applied during the high rpm rotation leads to high probability of failure at the leading edge of the pads. To alleviate the effect of temperature, lubricating oil is injected via channels in the friction pad. Uneven distribution of heat can cause failure in the friction pads

To study this complex physics a 2-way coupled FSI solution with LS-Dyna can provide design guidance in reducing damage and failure. This paper will describe the steps involved in the approach and show the reliability of the approach.

Fully Coupled LS-DYNA FSI Simulation of an Automotive Painting Process (RoDip)

S. Adya, I. Caldichoury, F. Del Pin, D. Bhalsod (Ansys/LST)

A good quality paint protects the car body from corrosion and mechanical damage and enhances the external appearance. There are several pretreatment stages in the painting process namely, cleaning, degreasing, activation, phosphating, passivation and electrocoating. During every stage, the car body is submerged in a process tank and rotated to ensure a homogenous coating. ABB's Roll Over Dipping (RoDip) is found to be an efficient painting process and is gaining popularity with the Auto OEMs. The parts being dipped are subjected to fluid pressure in the tanks during the dipping and rolling maneuvers which could deform the part. Virtual simulations are critical to optimize the process parameters to get a homogenous coat. In this paper we demonstrate how LSDyna can be used to simulate the motion of the Tilting device, the fluid structure interaction and finally the structural deformations, if any, of the components being dipped. To predict the response of the structure interacting with the paint, the ICFD module is coupled to the implicit structural solver. The paint is simulated as an incompressible flow with a free surface. The coupling is performed using strong FSI, resulting in an accurate solution of the dipping process with a single model.

Dynamic Testing with Automated Local Strain Measurement using IMPETUS

S. Riemelmoser, M. Schwab, M. Rollant (4a engineering)

Thermoplastic and lightweight materials have taken an essential position in nearly all applications nowadays. Cheap mass production, optimizable physical behavior and durability helped extending its field of use immensely. Due to the huge variety of plastics and compounds, testing its properties has become crucial to many industries. Over the last decade, we have developed the IMPETUS®, an easy to use test platform with multiple interchangeable setups, ranging from the simple 3-point-bending load case over the high-speed tensile testing up to component testing. In this contribution we will focus on the dynamic tensile test and have a deeper look at the measurement technique and the challenges that come in with the high strain rates up to 400 1/s. Basically, there are at least 3 measured variables that we have to record in the test. On the one hand, this would be the measured force [N] and global stroke [mm] and on the other hand the locally measured strain [%] measured with an automated high-speed DIC system, which must all be recorded absolutely time synchronously. In order to cope with the considerable time effort that comes along with optical strain measurement, an automated post-processing technique for 2D DIC will be discussed based on the example of a dynamic tensile test. While the 2D DIC measurement works very well to correlate experimental data with simulation models of a tensile test. The next step to increase the level of correlation is to use a 3D DIC system in order to be able to measure more complex load cases and compare the entire deformation behavior with the simulation to get even closer to the real test. In addition to the challenge of local strain measurement, there are certain dynamic effects such as vibrations, natural frequencies and filter influences in all high-speed tests which have to be taken into account as well to get meaningful experimental data. Therefore possible mitigation measures as well as a methodology on how to assess experimental conditions with respect to expected results quality are discussed.

Thermo-Mechanical Characterization and Modelling of Battery Cell Components with IMPETUS and VALIMAT

M. Schwab, H. Pothukuchi, M. Rollant (4a engineering)

State-of-the-art Automotive Li-Ion battery cells consist of many layers of thin foils (electrodes, separators) flooded within liquid electrolyte and surrounded by a tight casing. The overall mechanical behavior of a complete battery cell is determined by the mechanical behavior of its individual components. Within automotive applications, three different cell types with respect to their shape and housing have evolved, which are pouch, cylindrical and prismatic. Cylindrical and prismatic cells usually feature a metal casing, whereas pouch type cells feature a laminated film consisting of a polymer-aluminum-polymer layup as outer cell housing. Pouch type cells are frequently used in battery electric vehicles (BEVs) due to their advantage in terms of energy density. However, their housing (i.e. pouch foil) is especially prone to mechanical loads, which are typically introduced through the cell fixation and arise either due to cell ageing (swelling, degassing) or external mechanical loads (i.e. vibration, crash). A rupture of the pouch foil leads to leakage of electrolyte and, eventually, loss of electrical function and needs to be avoided during normal operation. Additionally, a controlled rupture behavior during a thermal runaway event is required in order to guide hot vent gas into a desired direction. Hence, there is a need for the prediction and virtual validation of the mechanical behavior of pouch foils. This paper deals with the mechanical characterization and the generation of validated material cards of pouch foils at various temperatures and strain rates. Therefore, two different test setups are considered. Static and dynamic tensile tests utilizing high speed cameras and local optical strain measurement (DIC) are performed. Additionally, puncture tests are conducted for studying the influence of biaxial stress states on the hardening and failure behavior. In this case, a 3D DIC technique is applied for local strain measurement. Based on the experimental results, suitable material models within LS-DYNA are investigated and parametrized using the software VALIMAT®. The versatility of the VALIMAT® software solution will be demonstrated with the aid of user defined input decks to simulate the different battery components under different loading conditions. Similar characterization and modelling approaches can be applied to current collectors and separators of battery cells with the aid of the IMPETUS® test system and the software solution VALIMAT®.

Numerical Simulation of Cell Venting within a Simplified 18650 Li-Ion Battery Pack

Daniel Grimmeisen[1], Marc S. Schneider[1]

[1]Cascate GmbH

Abstract

Violation of nominal operating conditions in Li-ion batteries can lead to internal damage and failure of the cells. This usually triggers chemical reactions that produce a large volume of hot gas. As a safety feature, 18650 battery cells are equipped with a safety vent. Once an internal pressure threshold is exceeded, the vent opens, and the gas escapes the cell at a high velocity to prevent uncontrolled structural failure. Within a battery pack, the hot gas needs to be guided to exit the pack while at the same time keeping neighbouring battery cells cool enough to stay within the safe temperature range. CFD simulation offers the capabilities to explore the mechanism of battery cell venting and flow guidance. This paper describes how such a simulation can be set up and run. Several steps are necessary to achieve this. First, Simcenter Battery Design Studio is used to model the 18650 battery cells. However, it is also described how this step can be avoided if certain prior knowledge about the process is available. Simcenter STAR-CCM+ is used for that and all subsequent steps. The cells are then assembled to a battery module and placed within a simplified battery pack housing.

Simulating Thermal Runaway of Batteries

Nils Karajan[1], Skylar Sible[2]

[1]DYNAmore GmbH
Industriestr. 2, 70565 Stuttgart, Germany
e-mail: nils.karajan@dynamore.de

[2]DYNAmore Corporation
565 Metro Place South, Suite 300, Dublin, OH 43017, USA
e-mail: skylar.sible@dynamore.com

Keywords: *Battery Abuse, Thermal Runaway, Multiphysics, Thermo-Mechanical Coupling.*

1 Abstract

Driven by the development of electric vehicles, the need for simulation models of batteries on cell, module and pack level has increased tremendously. While models for thermal management during normal use of batteries are already at a very satisfactory level, the misuse cases of batteries as well as the associated thermal runaway remain a challenging field.
This presentation will cover the multi physical model building process using LS-DYNA as a solver. Herein, all necessary assumptions and model building techniques will be discussed to setup and calibrate a thermo-mechanically coupled model that is driven by a battery model, which is able to capture normal charge and discharge use cases, short circuiting, and thermal runaway triggered by an exothermic chemical reaction. Herein, the focus is not so much on the mechanical model as the application in mind investigates a controlled heating of one cell in a battery pack. Following this, a thermal model will be calibrated using anisotropic thermal conductivity to account for the layered structure in battery cells and isotropic thermal conductivity for the housing of the battery module and pack. Moreover, a thermal contact needs to be defined which accounts for heat transfer via direct contact as well as radiation. The battery model is based on a so-called homogenized Randle circuit approach, where the parameters for the model will be calibrated using several charge and discharge experiments. On top of the battery model sits a short circuit model which can be triggered by either temperature or mechanical deformation. The model for thermal runaway will be included in homogenized fashion using an energy release rate function to include the energy that is released during the exothermic chemical reaction. The energy release function is activated when a critical temperature is reached and calibrated using autoclave tests in a controlled burn environment.

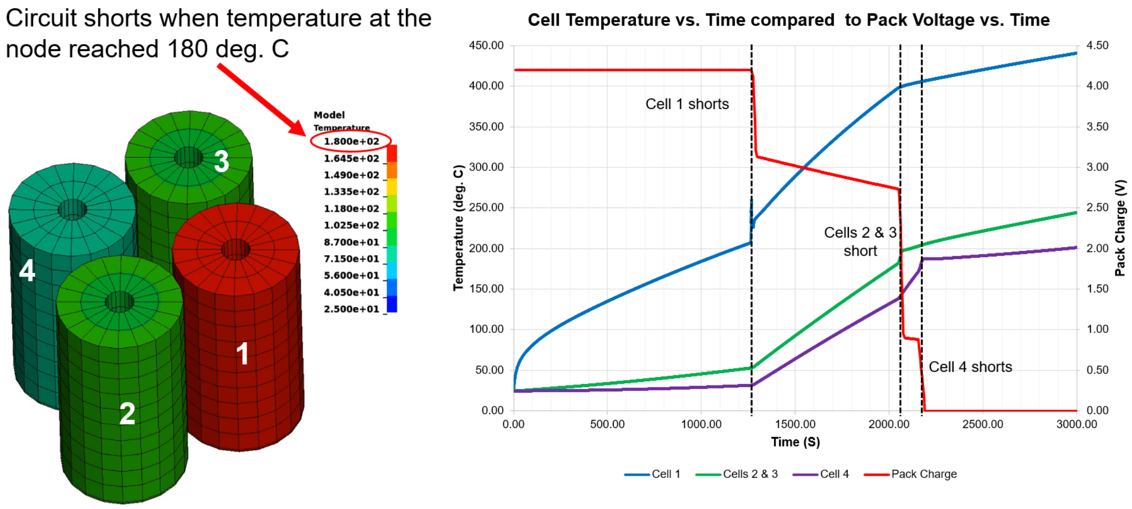


Fig.1: Relationship between voltage drop and temperature jump while the battery undergoes an internal short. Battery 1 is heated by a heat patch while other batteries heat up vie radiated energy from battery 1.

Simulation of Short Fiber Reinforced Plastics in LS-DYNA using Envyo mapped fiber orientations obtained from process simulation in Moldex3D

Marcus Gustavsson[1], Björn Stoltz[2], David Aspenberg[1]

[1]DYNAmore Nordic AB
[2]IKEA Components AB

1 Abstract

Accurate representation of materials is an essential part of the quest for realistic and predictive simulation, not least for anisotropic materials such as short fiber reinforced plastic (SFRP). Still, it is common to neglect the anisotropic properties of SFRP components when evaluating the structural performance of the design in FE simulation, thereby often failing to predict a realistic mechanical behavior and failure. Neglecting the anisotropic features of SFRP, especially at an early stage in the design development, may lead to a design that is not viable for the component in question.

A common manufacturing process for thermoplastics components is injection molding. How this process is set up will have a significant influence on the anisotropic properties for SFRP components; two geometrically identical components using the same material can still have vastly different structural performance and characteristics, given different manufacturing conditions.

Simulation of the injection molding process gives the ability to evaluate and iterate the design of the manufacturing process with relative ease, for instance, gate placement, which greatly influences fiber orientation and weld line locations. Fiber orientations predicted by the process simulation can efficiently be transferred to the structural simulation using a mapping tool such as Envyo. Working with process simulations can also simplify the simulation-based design process by replacing other techniques for obtaining information about fiber orientation, such as computer tomography (CT).

Ideally, one would like to have the actual fiber orientations in the test specimen when performing material characterization of SFRP materials, e.g., obtained via CT scanning. However, under the assumption that the process simulation provides accurate fiber orientations, a pure simulation-only approach could be used instead.

In this paper, a practical methodology for working with injection-molded SFRP components in LS-DYNA is presented, where the complete simulation chain from process simulation to structural simulation is performed using commercial software. Injection molding simulations using Moldex3D provide the fiber orientations that are transferred to LS-DYNA for structural simulation using Envyo. The study focuses on fiber orientation results and capturing its effect on structural properties by applying the LS-DYNA material model *MAT_4A_MICROMEC/MAT_215.

The aim is to show and evaluate the suggested simulation-only approach under real circumstances, including calibration using test specimens. The focus is on how to work with SFRP, primarily fiber orientation, from a practical point of view, although other aspects of the characteristics inherent to SFRP that should be considered are also discussed. Thus, possibly be the first step in getting started with SFRP simulation in LS-DYNA.

A VCCT- Cohesive Approach for the Efficient Modelling of Delamination in Composite Materials

Pierre M. Daniel[1,2], Johannes Främby[3], Martin Fagerström[4], Pere Maimí[2]

[1]Btechc - Barcelona Technical Center S.L., Martorell, Spain
[2]AMADE, University of Girona, Girona, Spain
[3]DYNAmore Nordic AB, Gothenburg, Sweden
[4]Dept. of Industrial and Materials Science, Chalmers University of Technology, Gothenburg, Sweden

1 Summary

The accurate modelling of delaminations is necessary to capture the correct behavior of composite structures subjected to demanding loads. While the use of cohesive elements is valid when the discretization is smaller than the failure process zone [1], for many composite materials it implies using a fine mesh, typically smaller than 1.0 mm, leading to excessive computational cost for large structures. On the contrary, the Virtual Crack Closure Technique (VCCT) allows the prediction of delamination growth in larger elements [2] but lacks of an energy dissipation mechanism. Therefore, it leads to excessive vibrations when the delamination propagates in dynamic analyses. The present work aims to combine the best of both methods in order to develop a viable solution for large structures, allowing for coarser meshes than what is possible to use today. To do so, the VCCT is used as a failure criterion to predict damage initiation while a cohesive-like model is added to dissipate the released energy. The model has been implemented in LS-DYNA in the frame of an adaptive user element recently published [3,4]. The model has been validated with Double Cantilever Beam, End-Notched Flexure and Mixed-Mode Bending tests. It demonstrates the ability of the method to accurately model delamination with larger elements and higher stable time step.

2 Literature

[1] Turon A., Dávila C.G., Camanho P.P., Costa J.: "An engineering solution for mesh size effects in the simulation of delamination using cohesive zone models", Engineering Fracture Mechanics, 2007, 74(10):1665-82.

[2] McElroy M.W.: "An Enriched Shell Finite Element for Progressive Damage Simulation in Composite Laminates", NASA/TP–2016-219211, 2016.

[3] Främby J., Fagerström M. and Karlsson J.: “An adaptive shell element for explicit dynamic analysis of failure in laminated composites Part 1: Adaptive kinematics and numerical implementation”, Engineering Fracture Mechanics, 2020, 240, 107288.

[4] Främby J. and Fagerström M.: “An adaptive shell element for explicit dynamic analysis of failure in laminated composites Part 2: Progressive failure and model validation”, Engineering Fracture Mechanics, 2021, 244, 107364.

Failure prediction with *MAT_215 in LS-DYNA for short and long fiber reinforced polymers

Peter Reithofer[1], Harish Pothukuchi[1], Stefan Kolling[2] and Jens Schneider[3]

[1]4a engineering GmbH, Traboch Austria, www.4a-engineering.at;
[2]Institute of Mechanics and Materials, Technische Hochschule Mittelhessen, Giessen, Germany
[3]Institute of Structural Mechanics and Design, Technische Universität Darmstadt, Germany

1 Introduction

The demand for weight reduction in the automotive industry has led to a strong interest in various composite material applications. The focus of the present material model development has been set to failure prediction and post failure energy consumption for classical polymer matrix composites reinforced by fibres or endless reinforced materials (e.g. carbon, glass, kevlar, etc.). The properties of composite materials are often highly influenced by the manufacturing process, which typically consists of injection molding in case of short (SFRT) and long fiber reinforced thermoplastics (LFRT). The fiber orientation develops during the injection process through the extensional flow and shear flow in the mold, see Fig. 1.

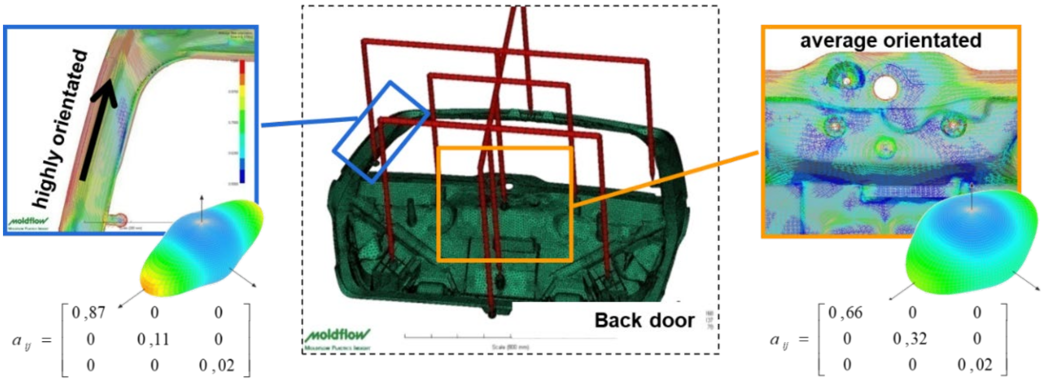


Fig.1: Typical fiber orientation regions in an injection molded part [1].

2 Material Behavior of SFRT and LFRT

Fiber size, geometry, content, and orientation have a very significant influence on the performance of a structural component. Fig. 2 gives a short overview on the mechanical material behavior of SFRT generated by dynamic three-point-bending tests. The tests were carried out on the testing device IMPETUS®, which was developed for dynamic material characterization [1].

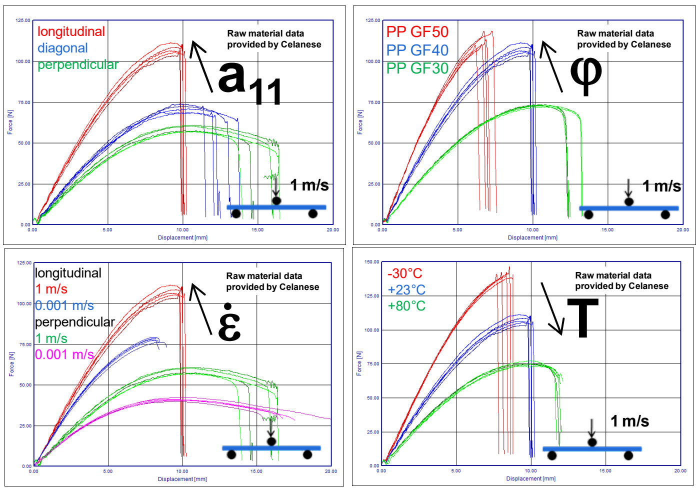


Fig.2: PPGF40 - Influence of fiber orientation, fiber content, strain rate dependency and temperature shown by the force-displacement curves of three-point-bending tests [2].

3 Material Characterization and Validation

In order to obtain a predictive simulation model, a high-quality material description of a so-called material card for the numerical simulation is required. In this contribution we will focus on a micro mechanical based material model in LS-DYNA® - **`*MAT_215`**. To obtain such a material card, a concept is needed, starting with molding adequate plates for test specimens to characterize the basic deformation and failure behavior in a standardized workflow up to final validation on component level.

Static and dynamic three-point-bending and puncture tests can be used to calibrate damage and failure criteria. Exemplary validation results (force-displacement) for a PPGF30 material are shown in Fig. 3. In case of using solid elements, the failure mechanism in the puncture case can be simulated quite well.

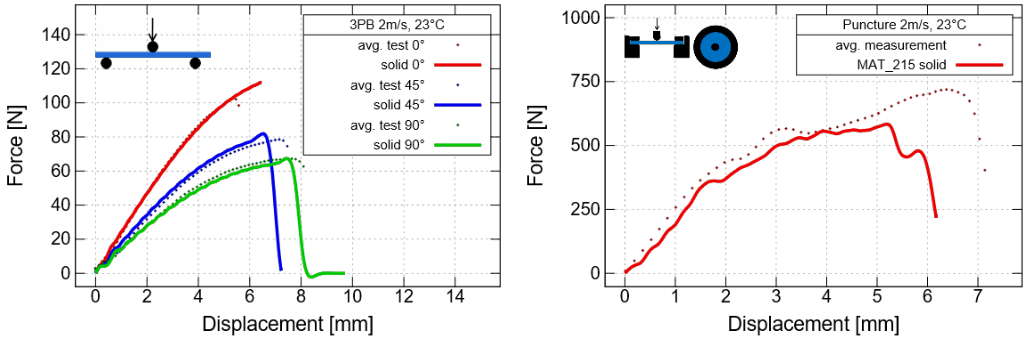


Fig.3: Validation results – Solid: left – dynamic bending for specimen cut out of a plate under different orientations; right – dynamic puncture test also conducted on IMPETUS®.

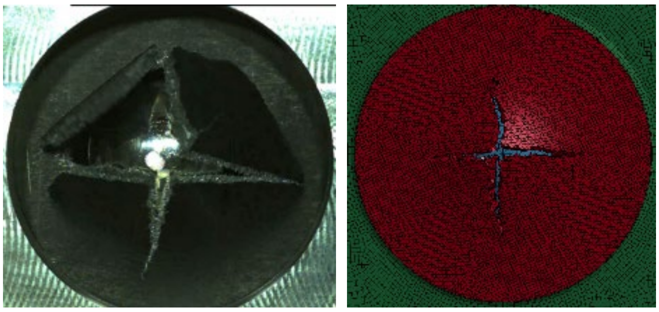


Fig.4: Failure under biaxial loading test versus numerical simulation.

Several SFRT and LFRT materials (e.g. PA6GF30, PPGF30, PBTGF30 among others) were already successfully characterized using the concept described. An overview of these investigations will be given with focus on the prediction of failure in crash and impact simulation.

Keywords: **MAT_215, material modeling, simulation of SFRT and LFRT, failure criteria for polymer composites, dynamic three-point bending tests, dynamic tensile tests, dynamic puncture tests*

4 Literature

[1] P. Reithofer, A. Fertschej, Dynamic Material Characterization Using 4a impetus; Polymer Processing Society Conference 2015 in Graz, AIP Publishing LLC.

[2] P. Reithofer, B. Jilka, Short and long fiber reinforced thermoplastics material models in LS-DYNA; 13th LS-DYNA Anwenderforum, Bamberg 2014.

Prepreg forming simulation

Siddharth Kumaraswamy[1], Abhik Dutta[2], Anders Bernhardsson[3], Mats Landervik[3], Malin Åkermo[2]

[1]Volvo Car Corporation
[2]KTH Royal Institute of Technology
[3]DYNAmore Nordic

The usage of composite materials in automotive body structures has the potential of reducing weight and thereby improving energy efficiency of the vehicles. Two key factors that limit their usage are long cure time for the material and the lack of simulation support. The recent development of snap-cure or rapid-cure prepregs can address the former problem. For the second problem, LS-DYNA simulations can support the design of composite parts and production process which can improve both their structural properties and manufacturability, avoiding the economic and environmental costs of trial and error used today to obtain defect free parts. This paper concerns the simulation of the forming of parts using unidirectional (UD) carbon fiber prepreg.

Compression molding of carbon fiber reinforced snap-cure prepregs have the potential of substantially reducing cycle times and manufacturing costs while providing the enhanced mechanical performance of the traditional prepreg hand lay-up process. However, the high level of automation in the process poses the risk of design and lay-up driven defects that may have a negative impact the quality and performance of the part. The objective of the simulations is to be able to improve the process, predict defects such as wrinkles and the resulting fiber angles that influence the part performance. The parts consist of number of layers of unidirectional (UD) prepreg material stacked on top of each other in a sequence where each layer can have different fiber direction to optimize process and product. The interaction between the layers will play a big role for the process and it will influence the wrinkle formation as well as the fiber rotations. Efforts have been made previously to model the friction in detail, see the manual [1] for the keyword *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE where the friction is defined in terms of the Hershey number and a Stribeck curve model is used to predict the change of the coefficient of friction with respect to different parameters. But it has not been possible to account for the difference in friction depending on the relative fiber angle in the layers in contact and the sliding direction. This difference in friction depending on direction is believed to be very important for the draping and sliding behavior in a compression molding process as well as for instance vacuum forming of prepreg composites. Previous research has indicated that the frictional load can vary by as high as one third for varying fiber angles [2]. It is also known that the consolidation pressure has a large effect on the interply friction [3][4]. To improve the capability of LS-DYNA to predict defects and properties in the forming process, direction dependence was implemented in the orthotropic friction for the mortar contact, see the manual [1]. The contact uses the fiber direction of the materials in the two layers in contact and lets the user input 4 friction values taken from tests of layers with different fiber directions sliding against each other in different directions. The 4 friction values can furthermore be replaced by tables to account for the dependencies of velocity and pressure.

The input format for the direction dependency is chosen to be a compromise between the amount of testing needed to determine the parameters and accuracy of the resulting predictions. An extensive testing of the friction between prepreg layers has been done. The testing shows the dependency on fiber directions, sliding angle as well as velocity and pressure. The analysis of the test results and the predictions made by the contact code for intermediate directions, i.e., directions for which the friction is interpolated by the code, indicate that satisfactory accuracy is obtained with the reduced input used.

Moreover, the feasibility of the orthotropic friction in a full-scale forming process is investigated. The simulations show that the contact performs well and has a certain impact on the result compared to a standard isotropic friction formulation.

The work in this paper was carried out with support from the Vehicle Strategic Research and Innovation program, FFI.

1 Literature

[1] LS-DYNA® KEYWORD USER'S MANUAL, VOLUME I, 04/05/21 (r:13718), LS-DYNA Dev, LIVERMORE SOFTWARE TECHNOLOGY (LST), AN ANSYS COMPANY.
[2] Sjölander J: PhD dissertation, “Improving Forming of Aerospace Composite Components through Process Modelling”, KTH 2018, ISBN 978-91-7729-881-6.
[3] Sun J, Li M, Gu Y, Zhang D, Li Y, Zhang Z. Interply friction of carbon fiber/epoxy prepreg stacks under different processing conditions. Journal of Composite Materials. 2014;48(5):515-526. doi:10.1177/0021998313476320
[4] Dutta A.: MSc thesis, “An experimental investigation of interply shear in fast curing composite prepregs.,” KTH 2020.

New Development of the Gap Closure Feature in LS-DYNA ICFD

Chien-Jung (Peggy) Huang[1], Facundo Del Pin[1], Iñaki Çaldichoury[1], Rodrigo R. Paz[1]

1Livermore Software Technology LLC, an ANSYS company

Abstract

There is a great interest in the fluid-structure interactions (FSI) of flow mechanics around valves in the heart or in mechanical parts. The capability to allow flow blockage due to the valve closure is very important. A new feature for managing the contact of immersed structures in the LS-DYNA ICFD solver is presented and tested. In the framework of conformal mesh used in ICFD solver, when two structures come into contact, the possible overlap or intersection of surface meshes at the contact location would cause the simulation to fail. As a solution, a clearance between two contacting surfaces is necessary. In the gap detection process, if the proximity between two surfaces is below a threshold value, the two surfaces are considered in contact in ICFD solver. To eliminate the resulting artificial flow leakage, the flow control treatment is performed on the elements in the gap at the contact location so that the flow is obstructed when contact occurs. The FSI simulations of 2D pump and the 3D pipe are presented to demonstrate the ability of this new feature.

New developments and future road map for the ICFD solver in LS-DYNA

Facundo Del Pin, Rodrigo R. Paz, Peggy Huang and Inaki Caldichoury

Ansys, Inc.

1 Abstract

This paper will discuss some of the new additions that will be part in the R13.0 release for the Incompressible CFD (ICFD) solver in LS-DYNA. The paper will also cover some of the highlights of R12. In the past year there has been an increased interest in the model of problems that involve the simulation of free surfaces, Fluid Structure Interaction (FSI) and porous media flow. These topics will be discussed, and the new features/improvements will be presented. The road map is a collection of feature requests from LS-DYNA distributors, Ansys ACE organization, academic collaborators and customers. Based on this a brief discussion the top topics will be presented including immersed interface techniques, gap closure models, multi-species transport and tighter integration with Ansys tools.

Truck Frame Optimization Considering Crashworthiness, NVH and Static Responses

Hong Dong[1], Juan Pablo Leiva[1], Brian Watson[1], Weizhao Gao[2], Feng Pan[2]

[1]OmniQuest
[2]ShareFEA Engineering Technology, Ltd.

Abstract

This paper demonstrates how to efficiently perform optimization for vehicle structures, taking into account nonlinear responses from LS-DYNA crash simulation, as well as responses from linear loading conditions such as NVH and Static. The optimization process is based on the Equivalent Static Load (ESL) method, and uses an iterative process which utilizes the non-linear structural analysis results from LS-DYNA and the linear structural analysis and optimization capabilities of GENESIS. With this integration, the combined multidiscipline problem can be solved with only a few LS-DYNA simulations (5 to 10). In addition, large-scale optimization techniques, such as topology, topometry, topography and freeform, can easily be employed. The optimization process and results will be demonstrated using two examples: topology optimization of a beam cross-section under impact and static loading and topometry design of a truck frame under crash, normal modes, and static loading conditions simultaneously.

An enhanced Design exploration using Modal decomposition of Key events in Frontal crash simulation

Masahiro Okamura[1]

[1]JSOL Corporation

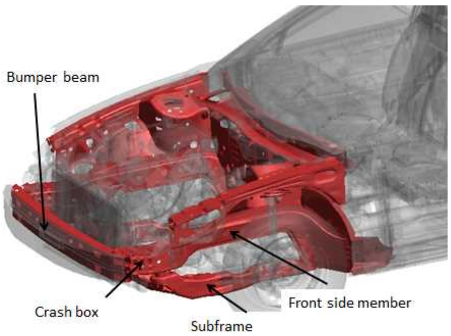
1 Issue of conventional response surface approach

In recent years, CAE has been used extensively in vehicle development, and parameter study of sheet metal thickness for design exploration and optimization is one of the major applications.
Response surface method is commonly used for this application among various analysis tools. The concept is to connect input variables such as sheet metal thickness and output variables such as firewall intrusion with non-linear functions such as radial base, kriging, and neural network.
This approach works fine with problems in case the degree of non-linearity is low. However, problems such as frontal crashes are highly non-linear so that this approach tends not to work well. The fundamental problem is a lack of understanding of the mechanisms and physics behind the phenomenon. Directly connecting scalar values from input and output does not fully explain “why” and “how” the changes in input affect the output, since complex causal chains of events behind the relationship between input and output variables are in a black box.

2 Proposed approach using DIFFCRASH

2.1 An example frontal crash problem

In order to showcase the proposed approach, a parameter study of the US NCAP full-frontal crash model has been conducted. NCAC Ford Taurus model[1] has been selected for this application as shown in Fig.1. The thicknesses of important parts have been parameterized as shown in Table 1.



(a) Engine room of NCAC Ford Taurus model

(b) Selected parts for the study

Fig.1: Design parameters of the frontal crash model

name	Description	Range [mm]
THK25	Thickness – Bumper	1.0 – 2.0
THK12	Thickness - Inner L	1.4 – 2.4
THK08	Thickness - Outer L	1.4 – 2.4
THK14	Thickness - Reinforcement L	1.9 – 2.9
THK21	Thickness - Inner R	1.4 – 2.4
THK18	Thickness - Outer R	1.4 – 2.4
THK23	Thickness - Reinforcement R	1.9 – 2.9
THK37	Thickness – Firewall	0.4 – 1.4

Table 1: Thickness variation of the crash relevant parts

A quasi-Monde Carlo simulation with 160 runs by Latin-Hyper Cube sampling has been conducted with LS-OPT. The maximum acceleration is calculated from driver side B-pillar acceleration signal filtered with CFC60. The maximum firewall intrusion is measured on the passenger side. The maximum wall force is the contact force between the rigid wall part and the vehicle filtered with CFC60.

2.2 Analysis process

By using DIFFCRASH[2], the influence of parameters is visualized as a contour plot in animations. Using this information together with the time history curves will enable engineers to easily find out major events in the crash and relevant areas. Once parts which play a key role in each important event and its timing, modal decomposition of deformation with PCA(Principal Component Analysis). This derives 2-3 dimensional vectors which represent the important event.
Part thicknesses, which are the input parameters in this case, give influence to the behavior of relevant parts. The influence appears in the modal plot so that regression analyses of the input parameters and modal contribution factors will reveal which input parameters are dominant. As the first step, thickness variation and modal contribution factors of each part in each event are studied by regression analysis. The behavior of a part will give influence to connected parts during the crash as a causal chain. In this case, the behavior of the bumper beam gives influence to the crash box and side members, and they affect the behavior of the firewall. Regression analyses of the modal contribution factors visualizes the causal chain. Lastly, the correlation between the modal contribution factors at 80ms and output values haven checked.

3 Summary

The analysis results through the process are summarized in Fig. 2. It shows the main path for the maximum wall force is from THK25 through mode1 at 20ms (also influenced by THK12), mode2 at 30ms, and mode2 at 80ms. On the other hand, The maximum acceleration and the maximum firewall intrusion are influenced mainly by THK37.
A similar result can be obtained by the response surface method. However, this approach gives engineers a clear view of the causal chain in the time & spatial domain, and helps engineers understand the mechanism for better and efficient structural design.

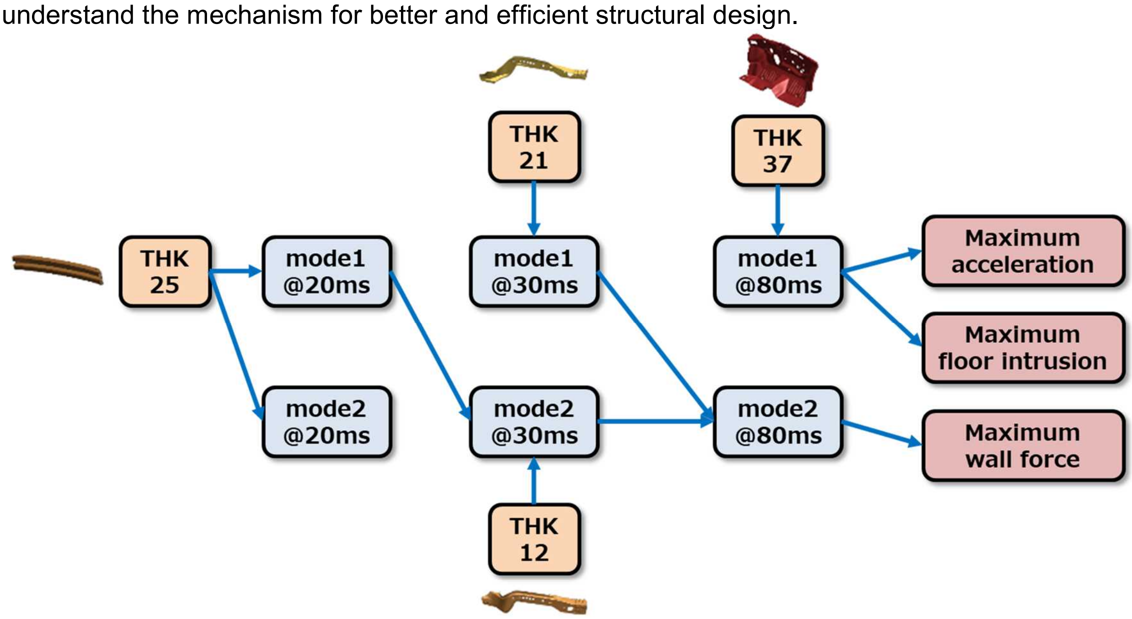


Fig.2: The influence path of the crash event in this study

4 Literature

[1] National Crash Analysis Center: Finite Element Model Archive,
http://www.ncac.gwu.edu/vml/models.html
[2] Borsotto D. et. al.: “Improving robustness of Chevrolet Silverado with exemplary design adaptations based on identified scatter sources”, 10th European LS-DYNA Conference, 2015

Numerical and Experimental Correlation of a Survival Cell Designed for a Bus Body Structure

Felipe Biondo[1], Alexsandro Sordi[1], Guilherme Magnabosco[1]

[1]Marcopolo s.a

1 Abstract

The behavior of mechanical structures, when subjected to impact load, is a matter of great relevance and its applications in terms of vehicle collision. When we analyze the superstructure of a bus, those vehicles must be tested according to prerequisites established in standards such as UNECE ECE 29 (European standard) or CONTRAN 629/2016 (Brazilian standard). The standards prescribe to use a pendular system to evaluate the frontal structure of the vehicle. In this regulation is defined the height and the mass that will collide with the structural modulus. However, the procedures described in these standards do not represent the real collisions involving these types of vehicles. This can be seen when comparing the energy imposed on the test module, detailed in CONTRAN 629/2016, where the energy imposed on the vehicle is approximately 20 kJ on each side of the test module, this corresponds to a collision of a 5 tons vehicle at 10 km/h or a 20 tons bus at 5 km/h.

In this context, this study aimed to develop a survival cell for the bus driver called FIA - (Frontal Impact Absorber), aiming to increase vehicle safety in situations where a frontal collision occurs. To reach the final concept of impact absorber, several nonlinear explicit structural analyzes were performed using the LS-Dyna software. In order to prove the efficiency of the FIA, a physical test was performed. The test consisted of a semi-frontal collision between two buses, where one vehicle remained stationary and the other collided with it at a speed of 40 km/h using an offset of 50% between the vehicles. A device was developed in order to keep the vehicle in its trajectory during the motion.

When evaluating the results, a correlation was noted between the experimental assay and the FEA analysis. The biggest differences in displacements found are in the range of 14%. When comparing the vehicles tested, the difference between the models with and without the FIA package becomes evident. The bus without this component had a deformation value measured at the point of greatest displacement of 648 mm, whereas the vehicle with FIA device, 273 mm, resulting in 58% less deformation in this vehicle. Therefore, it is evident that this device can help to minimize the damage caused to the occupants of the vehicle that is equipped with this device during the event of a collision.

Emphasis on Heat Affected Zone (HAZ) Modeling Around MIG Welded Joints in Crash CAE Virtual Predictive Full Vehicle Models

Santosh Pethe[1], Mohana Channegowda[2], Sanjay Patil[1], Anantharam Sheshadri[1], Kalid Jaboo[1]

[1]FCA US LLC, Auburn Hills, Michigan USA
[2]Altair Product Design Inc. Troy Michigan USA

Abstract

Current challenges in the auto industry are compelling virtual simulations to predict strength and rupture of MIG welded joints. Prediction of such joints enhances the design and development process. In body-on-frame vehicles most metal parts are joined with MIG welds and strength evaluation of such joints are crucial to vehicle crash and safety performance. Virtual simulation capabilities with these predictions help in enormous ways to reduce cost and time involved in proto-type testing of vehicles in the product development cycles. In the process of product design, it is very important to have accurate material model representations to have better crash CAE predictions. Current advancements in material models using GISSMO enable CAE analysts to capture ruptures for sheet metal parts. This is an effort to extend the rupture prediction of the Heat Affected Zone (HAZ) residual effect of the MIG welding process. Computational challenges are on the anvil to anticipate the IIHS small overlap complex fracture crash mode. Body-on-frame vehicle strategy is to contain a significant amount of the impact energy by frame and its critical joints. These joints are developed by MIG welding high strength low alloy or ultra-high strength steel sheet metals with gauges ranging from 2mm to 4mm. MIG welding introduces Heat Affected Zones (HAZ) adjacent to the weld lump. Mechanical strength of HAZ is degraded by the MIG welding process. In order to predict the separations and ruptures, CAE models need to capture strengths of the joints along with their HAZ. It is possible to model the parts and joints in detail with very small mesh sizes (approximately 0.5mm or even lower), but the down side of this would be higher computational time to run the models. We have to adhere to the calculated balance between computational time and accuracy of the CAE models. Determination of elasto-plasticity with rupture dependency on different stress states can be formulated by testing specimens taken from sheet steel blanks and such complex GISSMO material models are developed in-house. However, the size effect of HAZ makes it very difficult to generate coupons and their corresponding GISSMOs. In order to investigate mechanical strength of HAZ, hardness testing methods are employed. It is well known that the hardness is directly related to strength of metal, hence micro hardness tests are conducted on test specimens to get insights of the HAZ strength degradations. Furthermore, HAZ hardness degradations are compared with parent sheet metal hardness and accordingly parent material GISSMOs are scaled to mimic the properties of HAZ. Different CAE modeling methods for representing the MIG weld and HAZ areas were looked at (shells, solid and beam). Shell modeling method was adopted due to limitations of mesh size and computational time. Heat Affected Zone (HAZ) properties and modeling methodology was proven with well correlated multiple sub-component level physical tests. The same methodology was carried on to the full vehicle for predicting small overlap test mode responses. The crash test and CAE predictions showed very good agreement in terms of pulse, kinematics and intrusions.

Influence of Solidification-Dependent Microstructure on Subsequent Metal Forming Operations

Johannes Kronsteiner[1], Sindre Hovden[1], Stephan Jäger[1], Evgeniya Kabliman[2]

[1]LKR Light Metals Technologies, Austrian Institute of Technology, Giefinggasse 2, 1210 Vienna, Austria
[2]Technical University of Munich, TUM School of Engineering and Design, Department of Mechanical Engineering, Chair of Materials Engineering of Additive Manufacturing, Boltzmannstr. 15, 85748 Garching, Germany

Abstract

In the course of metal manufacturing, the liquid phase during casting and its subsequent solidification play major roles in characterizing the material properties (both micro- and macroscopic). Conventional metal deformation simulations would not take into account any initial spatial variations but assume a uniform microstructure. Physically based material models, however, allow to simulate microstructural effects based on measurable microstructural properties.
The melt flow in the component determines the casting thermal field and thus the grain evolution during solidification. The melt flow depends on the components' dimensions and various other casting process parameters. Thus, it is beneficial to use a hydrodynamic solver (CFD). The resulting microstructures (grain size distribution, etc.) can be afterwards mapped onto the following deformation/heat-treatment process simulations.
In the proposed workflow, a simulation of grain growth during solidification generates the initial spatial information ensuring the accuracy of following metal deformation simulations. Results are presented for a lab-scale hot compression test. By comparison with a homogeneous grain diameter distribution, the influence of a heterogeneity on the recrystallization behavior after the deformation was investigated.

Recent Developments of the EM-Module in LS-DYNA – A Discussion

Lars Kielhorn[1], Thomas Rüberg[1], Jürgen Zechner[1]

[1]TAILSIT GmbH, Graz, Austria

Since 2017 TAILSIT has maintained a close collaboration with Ansys/LST, formerly LSTC. Our partnership focuses mainly on the enhancement of LS-DYNA's electromagnetic (EM) solver module which is based on a coupling between Finite Elements (FEM) and Boundary Element Methods (BEM). This approach makes the EM solver highly suited for multiphysics problems. Prominent examples are, e.g., the simulation of parts moved by electromagnetic forces as well as processes like metal forming, welding, and induction heating.

Amongst other things, TAILSIT has designed and implemented:
- A monolithic FEM/BEM solver,
- Support for ferromagnetic materials, permanent magnets and
- The calculation of EM forces

during these last four years. These new features considerably expand the range of applications in which LS-DYNA might be used nowadays.

One of TAILSIT's latest additions to the EM-solver module is the development of a robust and optimal scaling preconditioner for the system of equations that allows for the treatment of non-conducting regions in the FEM domain. In conjunction with LS-DYNA's mechanical capabilities, the preconditioner permits, e.g., the study of snapping and latching magnets (Fig. 1). Thanks to LS-DYNA's FEM/BEM coupling approach, the modeling of such phenomena can be done in an intriguingly simple fashion. This most recent development marks a vast improvement and truly represents a unique feature which is currently not available in any other commercial product.

In this talk we give a brief introduction to the theory and present some benchmark examples. The practical application of the new preconditioner is presented elsewhere.
Instead, we will address the capabilities and current restrictions of the EM module from our point of view and discuss some strategies on how to overcome them.

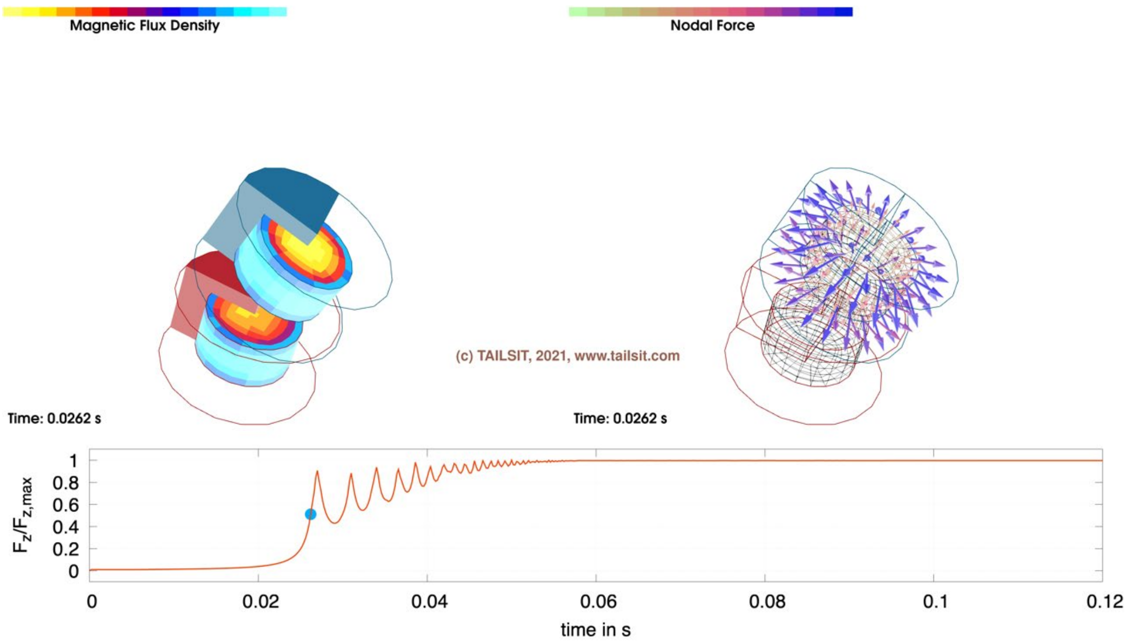


Fig.1: Two snapping magnets coated by non-magnetic material: magnetic flux density (left), nodal forces (right) and the total attracting force component (bottom)

Benefits of coupling FLACS-CFD® and LS-DYNA® for hydrogen safety applications

Pierre Glay[1], Laurent Paris[2]

[1]Dynamore France SAS, Versailles, France
[2]Gexcon France SARL, Paris, France

Introduction

There is a need for transitioning to an energy system with less greenhouse gas emissions and more sustainable energy production and consumption. A long-term structural change in energy systems is needed. Germany and France, among other countries, have decided to scale up the green hydrogen sector, with fundings of 9 billion and 7 billion euros respectively in the next 10 years.

Hydrogen as a new energy vector has many advantages over traditional hydrocarbon-based fuels. It is energy-efficient and can be environmentally friendly if it is be obtained from renewable sources. Potentially, in the future, it can solve many ecological and energy security issues. For more than a century, hydrogen has been produced and used for commercial and industrial purposes with a high safety record. However, the wider use of Fuel Cells and Hydrogen (FCH) technologies by the public (and not only by trained professionals) will require a new safety culture, innovative safety strategies, and breakthrough engineering solutions.

A methodological study on FSI with thermal coupling in LS-DYNA: ALE and SPH for low pressure die casting (LPDC) processes.

Salvatore Scalera[1], Simone Cavariani[1]

[1]DYNAmore Italia srl.

1 Introduction

LS-DYNA is a general-purpose finite element program capable of simulating complex real world problems. It is widely used in many applications and industries, historically mostly in the automotive world but it is increasingly used in other fields. Looking at manufacturing processes, forming and machining have a deep literature [1] [2] [3] while casting processes see less discourse.
Casting simulations require the simulation of fluid structure interaction together with the evaluation of the thermal exchanges between piece and die. For this reason, a methodology to approach the simulation of casting processes was developed. After a short description of the LPDC process, the writing will present an overview of the needed theoretical background, followed by more details of the model setup for the LS-DYNA simulations. In conclusion, the results and possible outlooks of this work will be discussed.

1.1 Low pressure die casting

In this manufacturing technique a die is filled with molten metal coming from a pressurized furnace. The metal rises into the die, the furnace is usually below it, and the pressure is held until the piece is fully solidified.
There are two main phases that can describe the process: the initial mold filling and then the cooling/solidification of the piece. The first one is especially challenging for a simulation because many different effects must be considered:

1. The material's viscosity is changing with temperature
2. The material density is increasing with decreasing temperature
3. The description of the fluid's flow and its interaction with the mould walls are fundamental to simulate a correct thermal exchange

Once the mould is filled, great care must be dedicated to the change of phase of the material, to help identify areas of the piece that could remain isolated from the reservoirs with the risk of obtaining cavities. Cooling rates are also crucial to determine the microstructure, alloy composition and mechanical properties of the final piece.

2 Simulation approaches

The aim of this paper is to describe a methodology that was developed to simulate metal casting processes using the Multiphysics capabilities of LS-DYNA. The initial focus of the study was centered into determining the best tools to accurately simulate the fluid structure interaction while also considering the thermal exchanges that are characteristic of processes like LPDC. After a literary review, two approaches were selected. The first one utilizes the arbitrary lagrangian eulerian (ALE) approach and the second one models the fluid with smooth particle hydrodynamics (SPH) particles, as both simulation techniques have been extensively used for the description of fluids [4] and can also be used in a thermal simulation. One other technique that could have been used for these simulations is the incompressible fluid dynamics (ICFD) solver, as described in the work by M. Timgren [5].
The following section will give a brief description of the concepts at the base of the two modelling approaches, before going into the details of the LS-DYNA models prepared for this study.

2.1 ALE

As the name suggests, this method includes both a lagrangian and a eulerian formulation. The former describes the deformation of the material using a mesh that follows it and therefore this formulation encounters problems when high distortions are present. In contrast, the eulerian formulation fixes the mesh in place and has the material flow through it. This approach does not suffer the same problems as the previous but presents great difficulty in describing surfaces and boundary conditions. The ALE formulation introduces an additional arbitrary reference frame that can move with respect to the other two. For a generic field variable f the relationship between its derivative in the different reference systems can be written as [6]:

$$\frac{\partial f(X_i,t)}{\partial t} = \frac{\partial f(x_i,t)}{\partial t} - w_i \frac{\partial f(x_i,t)}{\partial x_i} \tag{1}$$

Where X_i and x_i are respectively the lagrangian and eulerian coordinates, w_i is the relative velocity between the material velocity v and the mesh velocity u. Therefore, the relationship between the velocities is trivially described by: $w = v - u$. Applying the latter equation to the conservation equations, mass, momentum and energy, we can obtain [6]:

$$\frac{\partial \rho}{\partial t} = -\rho \frac{\partial v}{\partial x_i} - w_i \frac{\partial \rho}{\partial x_i} \tag{2}$$

$$\frac{\partial v}{\partial t} = -(\sigma_{ij,j} + \rho b_i) - \rho w_i \frac{\partial v_i}{\partial x_j} \tag{3}$$

$$\frac{\partial E}{\partial t} = -(\sigma_{ij} v_{i,j} + \rho b_i v_j) - \rho w_j \frac{\partial E}{\partial x_j} \tag{4}$$

To solve equations (2,3,4), LS-DYNA approaches them doing an operator split. The calculation is split for every time-step, initially the solver performs a lagrangian only calculation and the equations are formulated with $w = 0$. In the second phase, named *advection*, the mesh is moved back and there is transport of material through the mesh boundaries. There are many different ways to perform the advection phase: M. Souli [6], N. Aquelet [7] and Benson [8] provide a detailed analysis of the most common ones.

With regards to the thermal coupling the treatment is straight forward as the simulation has a lagrangian step. It is possible to go from the classical heat conduction equation (5)

$$\frac{dE_T}{dt} = \frac{1}{\rho}\nabla(k\nabla\theta) + \dot{q} \tag{5}$$

to the weak form:

$$\int_\Omega w_j \rho c \frac{d\theta}{dt} d\Omega = -\int_\Omega \nabla^T w_j k \nabla\theta d\Omega + \int_\Omega w_j \dot{q} d\Omega + \int_\Gamma w_j q d\Gamma \tag{6}$$

Which represents respectively: the variation in internal energy, the conduction entering and exiting the body, the heat generation inside the body and the heat flux across the boundary of the body. This can be solved using the FEM method and time integration schemes, more details are explored by Shapiro [9]. When simulating a thermomechanical process, the mechanical and thermal solver are weakly coupled. Each one of them has its own time integration scheme: the mechanical solver based on the calculated temperatures updates all the relevant quantities that then are used by the thermal solver to calculate new temperatures. The cycle is staggered and the solution proceeds one solver at the time.

Another approach that was explored initially is the S-ALE solver, an ALE formulation that brings several improvements both in robustness, ease of use and speed. However, at the time of writing, thermo-mechanical coupled simulations are not yet supported [10].

2.2 SPH

The SPH method discretizes continuous media approximating them with an equi-spaced grid of particles. It is a meshless lagrangian method. In contrast to the more common FEM, in which the governing equation relies on minimizing the potential energy, here what is evaluated in each particle are the principles of conservation of mass, momentum and internal energy [11]:

$$\frac{d\rho}{dt} = -\rho \frac{\partial v_i}{\partial x_i};\ \frac{dv_i}{dt} = \frac{1}{\rho}\frac{\partial \sigma_{ij}}{\partial x_j};\ \frac{dE}{dt} = \frac{\sigma_{ij}}{\rho}\frac{\partial v_i}{\partial x_j} \tag{7}$$

Where ρ is density, v is velocity of the particle, x is the position, E is the internal energy, σ is the stress and i and j are the indexes of the two particles at which the equations are evaluated.

It is possible to approximate continuous field variables with a discrete system of particles thanks to the interaction between them. The behavior of one is influenced by (and influences) all particles that lie inside the so-called smoothing length $\bar{h}$. The approximation of a continuum of the SPH method begins by switching from a continuous integral to a discrete sum of contributions, also called quadrature formulas for moving particles. This formulation can be used to discretize the previous conservation equations [12] and here is written for a generic function f:

$$\int_{\Omega} f(x)dx \approx \sum_{j\epsilon P} w_j(t) f\left(x_j(t)\right) \tag{8}$$

Where P is the set of particles, $x_j(t)$ is the position of the particle and $w_j(t)$ is its weight.
Considering $\bar{h}$, the interparticle distance and the scaling function W_{ij}, it is possible to approximate f for a given particle x_i with $\bar{f}(x_i)$ [12]:

$$\bar{f}(x_i) = \sum_{j\epsilon\Omega} w_j(t) f\left(x_j(t)\right) W\left(x_i - x_j, \bar{h}\right) \tag{9}$$

The scaling function, known as smoothing kernel, weights the contribution of all the other particles based on the distance from the ith particle and on the smoothing length $\bar{h}$. It is defined as [12]:

$$W\left(x_i - x_j, \bar{h}\right) = \frac{1}{\bar{h}} \Theta\left(\frac{x_i - x_j}{\bar{h}}\right) \tag{10}$$

Where Θ is generally a cubic B-spline that approximates a gaussian distribution, smoothing the contribution of particles that are further away.
The next step is to describe how heat exchange is threated in the SPH method. The considerations on the weak coupling between thermal and mechanical solvers described in the previous chapter apply also to SPH particles. The difference is related to the usage of the particle approximation; it is possible to go from equation (5) to this energy balance equation [13] :

$$\frac{dE_t}{dt} = \sum_j \frac{m_j}{\rho_i \rho_j} \frac{k_i - k_j}{\left|x_{ij}\right|^2} x_{ij} \nabla_i W_{ij} \tag{11}$$

Equation (11) applies to the heat conduction between two SPH particles.
In general, heat exchanges between different bodies (solids and shells) are managed through the **`*CONTACT`** keywords but only two-way contacts are applicable with the thermal solver, as it uses segments to compute the heat fluxes. SPH particles can only be used with one-way contacts (i.e. **`NODE_TO_SURFACE`**) hence they are not compatible with the standard treatment.
For this reason, Xu and Wang [14] developed a novel coupling approach through the keyword **`*DEFINE_ADAPTIVE_SOLID_TO_SPH`**. The method still possesses some limitations and needs further developments, but it is functional. It relies on the generation of hybrid elements (brick elements with SPH particles embedded inside) that work as link between the actual SPH part and the rest of the structure. At the time of writing the only parameter that is controllable by the user is a coupling conductivity parameter that by default is the average of the thermal conductivity of the two parts.
Alternative approaches to the thermal exchange between SPH elements and other types of elements are present in literature, like the work done by Frazer et al. [15] but these methods are not implemented in LS-DYNA as of version 12.0.

3 Simulation setup and outputs

Both approaches, even though quite substantially different, share most of the keywords. To save computation times only a slice of a mould with a simple geometry was modelled, resulting in a quasi bi-dimensional model. Figure 1 shows a snapshot of the mould during the filling process.

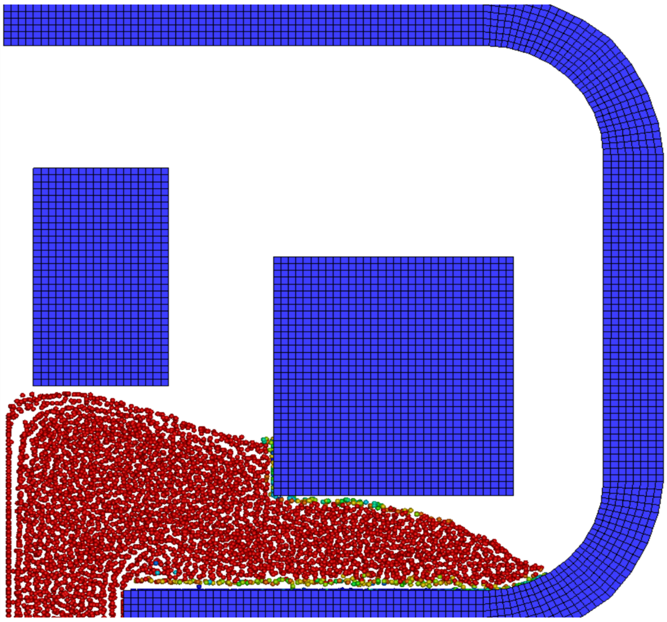


Fig.1: Snapshot of the mould during the filling process. In this case, the molten aluminum is modelled with sph particles.

The mould was modelled with hexahedral solid elements with a simple bilinear steel material model (***MAT_024** together with ***MAT_THERMAL_ISOTROPIC**). The fluid was molten aluminum with a pour temperature of $700\,°C$. In both cases a material model based on ***MAT_NULL** together with an equation of state (***EOS_GRUNEISEN** with $C = 0.54\frac{cm}{\mu s}$ and $S_1 = 1.35$) was chosen to describe the fluid's behaviour. For the ALE model the keyword is ***MAT_ALE_VISCOUS**, while for the SPH model ***MAT_SPH_VISCOUS**. These two analogous material models allow to activate two user defined subroutines (f3dm9ale_userdef1 and f3dm9sph_userdefin). The subroutines add a dynamic viscosity variable with temperature to the material model. The values of viscosity were evaluated using a viscosity versus temperature curve [16]. A side study was carried out to ensure that the fluid-dynamic behaviour of the two approaches was compatible, as the treatment of the viscous stress is not the same [4].

Both models included a keyword (***MAT_ADD_THERMAL_EXPANSION**) that adds a thermal expansion property via a thermal expansion coefficient α, which can change with temperature. The keyword allows for an orthotropic treatment of the phenomenon as well.

There are other possible material models [17] that include a more sophisticated range of relevant outputs. For example, ***MAT_UHS_STEEL** gives the possibility to calculate the phase constitution after cooling, based on the chemical composition and temperature variation history. It is possible to obtain a contour of austenite, ferrite or martensite phase fractions, hardness or yield stress distribution. Unfortunately, it as of now only supported for SPH particles.

In both simulations, in order to reduce the computational cost was made use of the **TSF** parameter in ***CONTROL_THERMAL_SOLVER**. It is the thermal speedup factor, which was increased from 1, during the filling part of the simulation, to 10 after the motion of the fluid had resided, to accelerate the solidification of the molten metal. It is possible to set it variable following a load curve by imputing a negative value in the **TSF** field, this option is not described in the manual at the time of writing but is already supported for LS-DYNA version 12.0. This minor approximation allows great reduction in the time to complete each simulation. The effect of the parameter is shown in Figure 2.

The outputs that are possible to obtain from these simulations are mostly similar for both approaches. Of course, temperature contours and residual stresses are available, they can be plot graphically or output as time series of specified locations. One of the possible advantages of the SPH formulation, due to its lagrangian nature, is the possibility to track the oxide formation in the piece during the filling and solidification phase.

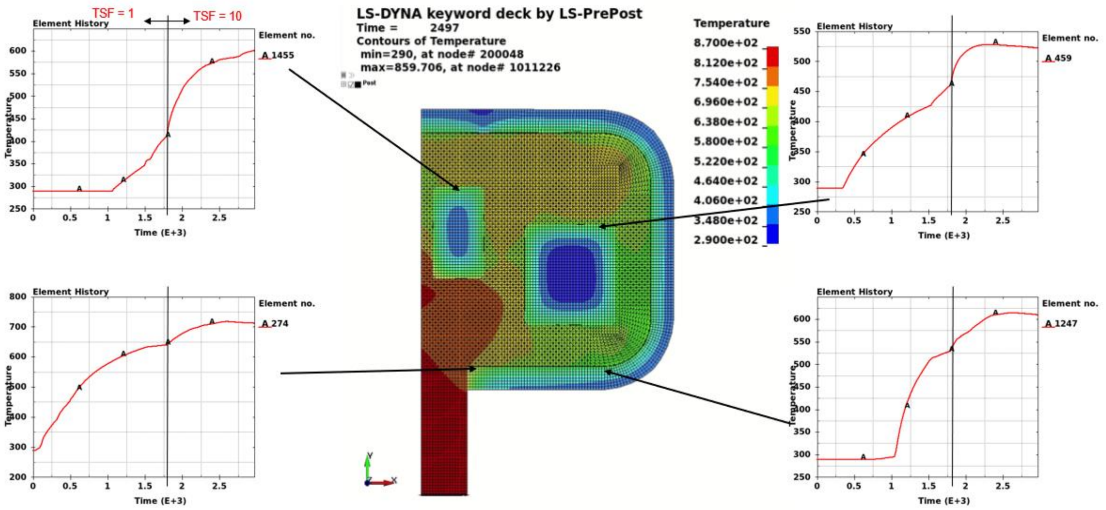


Fig.2: Effect of TSF parameter with the fluid modeled with ALE

As described in the study by Cleary [16], by estimating where, when and for how long the molten particles are exposed to the atmosphere (hence oxidation is happening) it was possible to have a detailed map of the possible critical location. Cleary introduced a linear accumulation model for the oxides in aluminum that follows the relationship [16]:

$$\frac{dO_x}{dt} = k_l \tag{12}$$

Where O_x is the oxide and k_l is the rate constant. This model introduced in the SPH particles would allow to predict the location of concentration of these defects and to evaluate different geometries in order to reduce the presence of oxides. This model would be less straight forward to implement for the ALE formulation.

Table 1. summarizes the possible outputs that are obtainable from the two models.

		SPH	ALE
INPUTS	Fluid structure interaction	☑	☑
	Thermal coupling	☑	☑
	Viscosity variable with temperature	☑	☑
	Thermal expansion coefficient	☑	☑
	TSF	☑	☑
	***MAT_UHS_STEEL**	☑	×
OUTPUTS	Residual stresses analysis	☑	☑
	Temperature profiles during the simulation	☑	☑
	Oxide Formation	☑	×
	Porosities	☑	☑

Table 1: Summary of the strengths and weaknesses of the two approaches

In the next two chapters, the differences between the two modelling approaches will be described.

3.1 ALE

To model the fluid using the ALE method, **`*SECTION_SOLID`** with **`ELFORM=12`** was chosen, as for this formulation the thermal analysis is more mature. The thermal solver struggles when more than one material is present inside one element, situation that would happen if in the simulation the ALE multi material formulation is used. In particular, the heat exchange was managed by the contact keyword **(`*CONSTRAINED_LAGRANGE_INTO_SOLID`**) by including a thermal exchange coefficient. The phase change from liquid to solid metal was enforced with a rapidly increasing viscosity of the material when it approaches the solidification temperature.

3.2 SPH

While in the ALE model the boundaries of the simulation were dictated by the actual ALE mesh, for the SPH model all the interfaces non described by the mould faces were modelled by using the **`*RIGIDWALL`** keyword. Great care was given to the selection of the parameter **`CPCD`** (thermal conductivity) in the **`*DEFINE_ADAPTIVE_SOLID_TO_SPH`** as it was the only free parameter available to match the thermal behavior of the ALE model. Even though the steady state response of the two model was compatible, this one parameter was not sufficient to capture the right transient thermal response of the fluid close to the mould and of the mould. In order to introduce more flexibility in the thermal exchange we considered variable also the thermal density of the hybrid SPH elements. LS-OPT was used to find the best combination of the two parameters based on the match between the time-history of two nodes close to the fluid-mould interface in the SPH and ALE model.
In addition to the possibility of modelling the solidification of the molten aluminum by greatly increasing the viscosity of the model, another way was experimented. By defining a sensor on every SPH particle it is possible to gradually freeze the fluid. Each sensor checks the temperature of the element and when the temperature falls under the solidification threshold it activates 3 **`*BOUNDARY_PRESCRIBED_MOTION`** cards that lock the particle in place. The feature works very well to describe a growing solidification front, but it is very computationally intensive, as many sensors are needed, and the feature was not exactly intended for this massive use.

4 Conclusions

This study followed the development of a custom LS-DYNA usermat package in which the viscosity of a material changes according to its temperature. Two methods (ALE & SPH) were used to simulate a low pressure die casting process to study their capabilities and their shortcomings. Targeted tests were conducted to minimize the numerical effects on the results, both on the fluid dynamics and on the thermal simulation, which highlighted that the SPH method requires a much smaller thermal timestep to reach convergence. The preliminary results show convergency of the two methods and the outlook of this work would be to compare numerical and experimental results for a final evaluation of the aforementioned methodology.

5 Literature

[1] Y. X. a. J. W. X. Zhu, "New Metal Forming Keywords in LS-DYNA," in *16th International LS-DYNA® Users Conference* , 2020.

[2] L. Z. Y. X. a. H. F. X. Zhu, "Advances in LS-DYNA® for Metal Forming," in *15th International LS-DYNA® Users Conference* , 2018.

[3] X. Z. &. L. Zhang, "Advance in Sheet Metal Forming - One-step Solution, Multi-Beads, Gravity Prebending, Auto Nets, and Local Compensation," in *12th International LS-DYNA® Users Conference* , 2012.

[4] S. Tokura, "Validation of Fluid Analysis Capabilities in LS-DYNA Based on Experimental Result," in *10th European LS-DYNA Conference*, 2015.

[5] M. Timgren, Coupled CFD and thermomechanical analysis of cooling in a die quenching tool, 2014.

[6] A. O. L. L. M. Souli, "ALE formulation for fluid-structure interaction problems," *Computational methods for applied mechanical engineering,* 2000.

[7] M. S. N. Aquelet, "A new ALE formulation for sloshing analysis," *Structural Engineering and*

Mechanics, 2003.

[8] D. J. BENSON, " AN EFFICIENT, ACCURATE, SIMPLE ALE METHOD FOR NONLINEAR FINITE ELEMENT PROGRAMS," *COMPUTER METHODS IN APPLIED MECHANICS AND ENGINEERING,* 1989.

[9] B. Shapiro, *Using LS-DYNA for Heat Transfer & Coupled Thermal-Stress Problems,* 2012.

[10] H. Chen, "LS-DYNA®Structured ALE (S-ALE) Solver," in *14th International LS-DYNA Users Conference*, 2016.

[11] J. M. K. Damaziak, "Comparison of SPH and FEM in thermomechanically coupled problems," in *Computational Technologies in Engineering*, 2018.

[12] J. L. Lacome, "Smooth Particle Hydrodinamics (SPH): A new feature in LS-DYNA," in *6th International LS-DYNA conference*, Detroit, 2000.

[13] J. Xu, "Heat Transfer with Explicit SPH Method in LS-DYNA," in *12th International LS-DYNA Users Conference*, 2012.

[14] J. W. Jingxiao Xu, "Thermal Coupling Method Between SPH Particles and Solid Elements in LS-DYNA," in *11th European LS-DYNA Conference*, 2017.

[15] L. S.-G. L. I. K. a. Y. C. Kirk A. Fraser, "Hybrid Thermo-Mechanical Contact Algorithm for 3D SPH-FEM Multi-Physics Simulations," in *IV International Conference on Particle-Based Methods*, 2015.

[16] P. W. Clary, "Exstension of SPH to predict feeeding, freezing and defect creation in low pressure die casting," *Applied Mathematical modelling,* 2010.

[17] Hallquist, *LS-DYNA manual v 12.*

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, Switzerland, France, Italy and USA.