Ansys

DYNA MORE
An Ansys Company

Book of Abstracts

14TH EUROPEAN LS-DYNA CONFERENCE

October 18-19, 2023 | Baden-Baden, Germany

Oasys
LS-DYNA ENVIRONMENT

Hewlett Packard Enterprise | AMD

The 14th European LS-DYNA Conference and Users Meeting is organized by

Ansys

DYNAmore – An Ansys Company

DYNAmore Nordic – An Ansys Company

In association with

Oasys LS-DYNA ENVIRONMENT

Dynas+ Numerical Simulation

DYNAmore GmbH, an Ansys Company, Industriestr. 2, D-70565 Stuttgart, Germany
Tel. +49 (0)711 - 459600 - 0, info@dynamore.de, www.dynamore.de

ISBN 978-3-9816215-7-0

Sponsors

Platinum Sponsors

Oasys LS-DYNA ENVIRONMENT

Hewlett Packard Enterprise | AMD

Gold Sponsors

CORNELIS NETWORKS

d3VIEW A Data-to-Decision Platform

rescale

Silver Sponsors

4a ENGINEERING

Applus+ DatapointLabs

BETA SIMULATION SOLUTIONS

DEP MeshWorks

eta Innovation Starts Here

gns

GNS Systems IT Services for Engineering

JSOL

SCALE IT-Solutions for CAE

scapos

Table of contents

Table of contents

Table of contents

Table of contents

Table of contents

Table of contents

Table of contents

Table of contents

Table of contents

Oasys Ltd is the software house of Arup, a global firm of designers, consultants and experts dedicated to sustainable development.

We have been working with Ansys (and formerly Livermore Software Technology Corporation), the developers of LS-DYNA, for over thirty years, using the software for our own consulting work as well as distributing to clients in the UK, Europe, Asia, and North America.

Our in-depth knowledge has enabled us to create the Oasys LS-DYNA Environment – tailored tools to prepare models, interpret results, and share solutions. They are at the leading edge of pre- and post-processing software and are used by LS-DYNA customers worldwide.

Our suite of products play a pivotal role in enabling our clients to deliver projects that might not otherwise be achievable without complex analysis.

Our portfolio includes:

LS-DYNA: a non-linear finite element solver, world-leading in automotive crashworthiness and other specialist domains.

The Oasys LS-DYNA Environment: PRIMER (pre-processing), D3PLOT, D3PLOT Viewer, T/HIS, REPORTER, and REPORT Viewer (post-processing), and SHELL, all designed specifically for LS-DYNA.

Finite Element (FE) Models: a range of specialist FE models developed with companies including Cellbond and Arup and licensed to clients; examples include crash barrier models, pedestrian impactors, and occupant dummy models.

Other Tools: a range of additional tools to support LS-DYNA including HYCRASH, FEMZIP, LS-OPT, LS-TaSC, JFOLD and HPM / HMRD.

With operations globally, our team of experts support users to bring products and ideas to life.

Conference Platinum Sponsors

Hewlett Packard Enterprise | AMD

Manufacturing leaders rely on computer-aided engineering (CAE) as a critical tool in developing new and more innovative products faster. CAE eliminates the need to build multiple prototypes by augmenting physical models with computer-generated models, reducing costs and accelerating time to market.

However, optimizing a design to ensure optimal fidelity now requires more iterations and even more complex simulations. Despite having finite data center capacity, CAE centers are under pressure to take on new workloads and employ new techniques to meet federal and industry standards — these include Multiphysics simulation, AI/ML, and the use of digital twins.

To address these challenges, customers who run Ansys LS-DYNA software on highly efficient HPE-powered AMD cluster platforms can reduce design timeframes and get better-performing products to market faster.

Ansys, HPE and AMD collaborate to deliver solutions that optimize LS-DYNA performance and accelerate product development cycles. We provide solution reference architectures with recommended hardware configurations to run LS-DYNA (and all Ansys applications) to optimize both cost and performance. As a result, customers can streamline and simplify the decision process.

We also provide joint customers with:

- Standard Ansys application benchmarks results
- Customer-based benchmarks using your own models
- Technical CAE workflow specific whitepapers

With these documents and benchmark data, HPE and AMD customers can better understand the hardware requirements and capabilities needed to sustain and fully benefit from Ansys innovations.

Contacts:

Tony DeVarco
WW – Director, HPC and Manufacturing Vertical
HPE
anthony.devarco@hpe.com
Ph: 1-510-364-0408

Mary Bryan Bass
HPC Product Marketing
AMD
Mary.Bass@amd.com
Ph: 1-615-428-8522

Visit: www.ansys.com/hpe

Cornelis Networks is the leading independent provider of purpose-built, scale-out interconnects for high-performance computing, artificial intelligence, and high-performance data analytics. Leveraging 20 years of experience dating back to the genesis of InfiniBand, Cornelis Networks designs leadership-class fabrics delivering the lowest latency and highest message rate at extreme scale. The superior price-performance of Cornelis Omni-Path enables OEMs and system integrators to deliver highly cost-effective network solutions, freeing cluster budgets to deploy additional compute and accelerator resources, resulting in greater overall system performance and faster time to insight for customers' mission-critical workloads.

d3VIEW, Inc. is head-quartered in Rochester Hills, Michigan, USA. Founded in 2003, the d3VIEW team is led by its Founder and Chief Executive Officer, Suri Bala, who with his decades of experience in the simulation industry and through continuous innovation, pioneered the data-todecision platform. The key mission of d3VIEW is to empower engineers and scientists with knowledge and insights from their data to accelerate virtual product design, development and optimization.

Multiple interactive applications which are integrated seamlessly in one comprehensive platform not only make d3VIEW more appealing with its powerful features in the areas of data mining, storage, analysis, collaboration, visualization, prediction, material calibration, to name a few, but also extend the possibility of tremendous automation and tackling of complex engineering problems by anyone with minimal exposure or expertise. Originally designed and developed for addressing the pain points in Crashworthiness, d3VIEW is now widely used in a variety of other science and non-science areas including NVH, HVAC, Battery Management, Precision Medicine, Long Range Planning, etc.

For more information on how d3VIEW can help address your needs, please email info@d3view.com.

Conference Gold Sponsors

Rescale provides high performance computing built for the cloud to empower engineers while giving IT security and control. The Rescale platform makes it simple for engineers and scientists to harness the most advanced software and computing architectures for cutting-edge simulation and AI-driven innovation. For IT, the Rescale platform provides full-stack security and support, and delivers policy-based financial and architectural controls to maximize performance and efficiency. Rescale powers the world's leading companies to accelerate innovation across industries including life sciences, automotive, energy, semiconductor, aerospace, and manufacturing.

Learn how you can modernize high performance computing at Rescale.com

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

4a engineering GmbH is a technically oriented research and development company specialized in testing and simulation solutions for material engineering and battery applications. The core competence of 4a engineering GmbH lies in the development of specialized testing hard- and software as well as material characterization and material card generation for finite element simulations. Furthermore, 4a engineering GmbH is specialized in the abuse characterization of battery cells and cell stacks and the development of validated simulation models of battery cells. 4a engineering GmbH works with a wide range of specially developed simulation software and testing systems with the aim to further increase automation within the entire process chain of material card generation and battery characterization. In addition, 4a engineering GmbH has considerable experience in the application of complex material models, as well as the development of new material models and simulation methods.

Applus DatapointLabs Technical Center for Materials is a US-based center of excellence for the measurement of physical properties of diverse materials needed for product development, CAE, and R&D, providing data for over 1,800 companies around the world, including automotive, aerospace, appliance, biomedical, consumer products, electronics, and material supplier verticals. With our testing services, TestPaks for CAE, and Matereality software for materials, we help companies build enduring data collections that accurately represent the materials used in their products. To date, more than 30,000 materials have flowed through the laboratory, including plastics, metals, rubbers, foams, adhesives, and composites. Recognized as an approved test lab by leading OEMs, we are ISO 17025 and Nadcap accredited. DatapointLabs supports LS-DYNA and Ansys users with TestPaks and CAETestBench validations for both programs. TestPaks provide ready-to-use material models for specific simulation tasks.
Learn more at www.datapointlabs.com/about

BETA is an engineering simulation solutions provider, dedicated to the development of state of the art software systems. For more than 30 years, we have been developing software and delivering services for the front runners of numerous sectors, by listening to their needs and taking up even the most demanding challenges.

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.

Conference Silver Sponsors

DEP MeshWorks 2023: A software designed by CAE Experts for CAE Experts.

TRANSFORM: Harness the power of ConceptWorks and Full Vehicle Morphing.
- Electrify your designs with eMOD.
- Dive into Multi-Disciplinary Optimization (MDO) using Reduced Order Modeling.
- Rapidly conceive your BIW directly at the FE level, ensuring your designs are always cutting-edge and efficient.

AUTOMATE: Complexity, simplified.
- With our advanced AI/ML algorithms, effortlessly navigate the demands of LS-DYNA.
- Hexamesh the most intricate parts with unmatched precision, transitioning from weeks of effort to mere hours.

ACCELERATE: Propel your workflow with unmatched efficiency.
- Seamlessly transition from early concept to final validation, slashing 30-70% off your process time and consistently staying ahead of the competition.

INNOVATE: Remain at the forefront of CAE innovation.
- Our pioneering algorithms promise a powerful and immersive user experience, perfectly tailored for LS-DYNA, ABAQUS, NASTRAN, and ANSYS.
- Reach optimal designs quicker, eliminating the tedious back-and-forth.

Curious? Witness the revolution at our booth. Tailored demos await.

ETA's engineering CAE-driven solutions (experience, innovation, and partnership) reduce the cost of product design and development by streamlining testing and prototyping processes. We strive towards complete virtual prototyping using our experience, products, and advanced technology.

As a true leader in CAE solutions, our expertise guarantees client success

Our wealth of simulation project experience paired with our strategic partnerships with global manufacturers and research institutes uniquely positions us to serve all our client's needs.

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.

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 sys-tems 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.

JSOL Corporation, as an ANSYS channel partner, provides cutting edge CAE technology and technical consulting around LS-DYNA and our own software products. We also offer expertise in modeling and itsknow-how as well as a software based on the expertise. JFOLD is an analytical software which creates intricate folded airbag models for automotive crash simulation. J-SEATdesigner is an integrated simulation tool for seat design. We also develop simulation tools for press forming - JSTAMP for analyzing metal forming and J-Composites for composite material. J-Concept for crash simulation at concept design phase is coming soon. Please come to visit our exhibition booth for more details.

SCALE specializes in providing pioneering IT solutions for simulation data management and CAE processes. Unleash the power of innovation with our flagship software solution, SCALE.sdm., designed to orchestrate the holistic management of simulation and test data within the realm of virtual product development.
Our portfolio of IT services encompasses a wide spectrum, with a special focus on bespoke software development and robust support for CAE processes and data management. We are also committed to advancing new simulation methodologies.
Based in Germany, SCALE operates from branches positioned in pivotal automotive industry hubs like Ingolstadt, Stuttgart, and Braunschweig, as well as in the innovative heart of Saxony's Silicon Valley, Dresden. Additionally, we have a state-of-the-art development center in the vibrant city of Jaipur, India, as well as a dynamic subsidiary in Columbus, Ohio, USA. Moreover, we maintain close collaborations with esteemed partners across Asia who assist in the distribution and support of our cutting-edge products in that region.
Team up with SCALE, where innovation, expertise and a global network come together to propel your ambitions.

scapos provides sales, marketing and support for advanced software solutions arising in particular from a number of Fraunhofer institutes. The scapos product portfolio includes: optimisation software, computer-aided engineering tools, natural language processing and fast solvers. Its industrial customers range from large corporations to SMEs.
Complementing its own activities, scapos works with more than 20 distribution partners worldwide.
At the 14th European LS-DYNA Conference 2023, scapos will be highlighting: the MpCCI suite (co-simulation supporting LS-DYNA and data mapping for complex CAE workflows); ModelCompare, SimCompare and SimExplore (support tools for the comparison of FE models and crashworthiness simulation results) and MESHFREE (fluid and continuum mechanics simulations).

The presented software products of scapos are:

Contact:
Web: www.scapos.com
Email: info@scapos.com
Phone: +49 2241 14 4400

Recent Developments in LS-DYNA I

R. Grimes (Ansys)

Key success factors for digital vehicle development at Mercedes-Benz

M. Hermle, M. Feucht (Mercedes-Benz)

MegaCastings in Body Structures - A MegaChallenge for Crashworthiness CAE

J. Jergeus, P.-A. Eggersen (Volvo Cars)

Evolution and Future of crash safety CAE in Toyota

K. Kawahara (Toyota)

Data Driven Modelling of Materials and Structures

S. Hiermaier (Fraunhofer EMI)

Recent Developments in LS-DYNA II

T. Erhart (DYNAmore)

Development of Far-Side Sled Simulation Model with Airbag for Virtual Testing

Hisaki Sugaya[1], Yu Kanayama, Kanae Matama[1], Yusuke Chida[1],
Takashi Kikuchi[1], Nanami Kobayashi[2]

[1]Honda Motor Co., Ltd
[2]Auto Technic Japan co., LTD.

1 Background

Currently, Euro NCAP is planning to introduce a virtual test to improve the robustness of safety performance. Starting with the Far-side impact test, where robustness will be evaluated at different angles and seat positions. Since robustness is evaluated only by simulation, it is crucial to improve the accuracy of the model.
Therefore, the objective of this study is to verify the model accuracy level by comparing the simulation with an actual SLED test with the airbag as a benchmark for the virtual test.

2 Method

In Step 1, the accuracy of the single-component model was verified. The load time history results of the impactor were compared with the simulation.
In Step 2, the accuracy of the far-side SLED model with the far-side airbag was validated by comparing the response characteristics of WorldSID dummy model version 7.6.1 produced by Dynamore with the actual test dummy using ISO18571.

3 Results

In the single component verification, the load time history characteristics of both the center console and airbags were simulated and reproduced the actual device test well.
In the SLED validation of Step 2, by modeling the belt around the buckle using 2D element, the dummy response of the whole body was scored by ISO18571 with 0.53, 0.49, and 0.72 for the head, spine T4, and Pelvis acceleration, respectively. In terms of the load cell, the upper neck load cell had 0.56 for the force and 0.55 for the moment. The average ISO score for the major measurement was 0.64.
The acceleration for the lower body of the World SID dummy showed over 0.72 of the ISO score, while the upper body was 0.55, indicating a lower accuracy level for the upper body. Compared to the overall results, the ISO score for Neck was particularly low, indicating that there are issues to be addressed.

4 Discussion

By reproducing the SLED test in simulation, the average ISO score of 0.64 indicates a reasonable result for reproducing the actual test as the starting point of the virtual test.
However, there are some areas, such as T4 and Neck where the ISO score is low.
The T4 acceleration magnitude was a large gap between the two during airbag deployment. It may cause vibration frequency response of the interaction between the WorldSID dummy and the airbag model differs significantly from that of the actual model. For the neck of the WorldSID dummy, the ISO score was particularly low. According to the WorldSID user report, in case of the high-speed impact, neck force and moment has a gap even if the simplified configuration. It may suggest the velocity dependence may be modified to improve. Further improvement of both models is considered necessary for future work.

5 Conclusion

In this study, on the Far side, we compared the simulation and the test with WorldSID dummy and calculated ISO score as a benchmark. It obtained 0.64 as the mean of major items showing reasonable results to reproduce the test.

Far side crash correlation and sensitivity study for virtual testing

Roland Schirmer[1], David Ide[1], Carlos Gonzáles[2]

[1]Stellantis N.V.
[2]Applus Idiada S.A.

1 Abstract

In 2024 the monitoring phase of the virtual far side occupant assessment is going to start. The vehicle manufacturer will carry out the physical sleds and virtual tests. Variations of the impact angle and the seat position are going to be assessed with purely virtual tests. The car manufacturer has to show that the correlation level of his simulation model is sufficient. For that the ISO score rating according to ISO/TS18571 and the selected ATD injury criteria are used on the two validation tests.

To be prepared for this challenge, Stellantis put together a cross functional, international CAE team of methods development and safety department members. The task was to test if the existing model content and the level of detail in the subsystems fulfill all performance requirements and to identify the key enablers to reach the correlation targets. A compact class production vehicle was chosen, which does not have specific countermeasures for the far side load case.

In addition to the vehicle itself, the kinematics of the Worldsid50 LS-Dyna simulation model version 7.6.1 and its predictivity was investigated under far side load conditions. This dummy version is not certified for the far side virtual testing.

Some enablers for a good correlation have been identified which affect both the dummy environment and the dummy itself. This presentation is an extract from the examined model characteristics and the drawn conclusions.

2 Literature

Euro NCAP: “VIRTUAL FAR SIDE SIMULATION & ASSESSMENT PROTOCOL”, Version. 1.0, 2023
Euro NCAP: “FAR SIDE OCCUPANT TEST & ASSESSMENT PROCEDURE”, Version 2.3, 2022

Crash

Multi-stage Analysis Approach to Low Speed Vehicle Impacts using the *SENSOR Keyword

Hari Patel[1], Ben Crone[2]

[1]Arup (Specialist Technology, Analytics & Research), London, UK
[2]Arup (Specialist Technology, Analytics & Research), Birmingham, UK

Abstract

The low-speed impact tests outlined in ECE R42 and FMVSS 581 consist of multiple consecutive impacts on a vehicle bumper to assess vulnerability to damage and repairability. Typical CAE approaches to assessing multi-stage analyses involve running each stage of the analysis individually, inputting deformations, stresses, and strains from the end of the previous analysis. This approach typically requires manual model editing before each analysis, which is time consuming and increases the risk of human error.

This paper outlines a methodology using the *SENSOR keywords to run multiple low speed impacts sequentially in a single analysis. This includes the use of *SENSOR keywords to initialize each impact in order, slow and restrain the vehicle and terminate the analysis based on the model responses.

The techniques discussed can be applied to other multi-stage analyses and more widely to terminate analyses at a specific event prior to a specified run time, saving computational time and reducing the number of output files.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Development of a 2020 SUV vehicle FE model – Update

Rudolf Reichert[1], Steve Kan[1], Umashankar Mahadevaiah[1], Lukas Fuchs[2]

[1] George Mason University (GMU)
[2] University of Stuttgart

1 Objective

Finite element (FE) vehicle models allow researchers to conduct diverse simulation studies. Members of the Center for Collision Safety and Analysis (CCSA) at George Mason University (GMU), that also built the core team of the formerly known National Crash Analysis Center (NCAC), have been developing a fleet of publicly available FE vehicle models over the past 25 years. This paper describes the latest model, a 2020 Nissan Rogue.

2 Methods

State-of-the-art modeling techniques were used to develop the updated detailed SUV vehicle model. A reverse engineering process, which included geometry generation, vehicle tear down, meshing, connection modelling, and material characterization. Data from available and conducted full-scale crash tests were used to validate the FE model for multiple impact configurations. FMVSS No. 214 side impact load cases included the Moving Deformable Barrier (MDB), pole, and static door crush. NCAP rigid barrier and NHTSA's oblique impact scenarios were used for validating performance in frontal impact conditions. Suspension characteristics were validated using various low-speed suspension tests. Furthermore, pedestrian safety characteristics were validated using available EuroNCAP test data

3 Results

The developed detailed FE model of a crossover SUV consists of about 1250 parts and 3 million elements representing geometry, connections, and material characteristics of relevant structural and interior components. A uniform mesh with an average element size of 8 mm and a timestep of 1 microsecond was used. The developed FE model showed good correlation when compared to test time-history data for most acceleration, deformation, and force measurements in respective impact scenarios, based on Correlation and Analysis (CORA) and ISO 18571.

4 Discussion and Limitations

The FE model represents relevant exterior and interior parts, including body in white, wheels, axles, suspension, seats, door trims, pedals, steering wheel, and instrument panel. Similarly, additional interior detail, such as modelling the roof trim would be needed for FMVSS No. 201 evaluations, for example. The model represents results after a second validation iteration and is continuously being updated.

5 Conclusions

A developed FE model of a 2020 Nissan Rogue SUV showed good correlation with existing crash test data for frontal, side, suspension, and pedestrian safety impact configurations. It is the latest FE model made publicly available by CCSA at GMU. The model has been used for several research studies, including "Crash simulation of FMVSS No. 214 side impact safety performance", "crash compatibility for unoccupied automated driving systems (UADS)", "Development of a THOR-05F FE model", and "Frontal oblique impact research", sponsored by the National Highway Traffic Safety Administration's (NHTSA).

NHTSA Test Data Analytics – Lessons Learned and Data Insights

Suri Bala[1]

[1]d3VIEW, Rochester Hills, USA

Abstract

National Highway Traffic Safety Administration (NHTSA), a subset of United States Department of Transformation, is responsible for keeping people safe on America's roadways. NHTSA enforces vehicle performance standards for automobiles and performs several tests as part of the new-car assessments for crash safety. NHTSA primary focus is to reduce deaths, injuries from motor vehicle crashes. The crash test-data for several crash scenarios are available online for the public and can be downloaded at no cost. This availability of such data is a data-mine for any crashworthiness engineers but can be a daunting task to process such large amounts of data.

This paper will review the work that was undertaken over the last 12 months to import frontal crash test data to provide a benchmarking and prediction ability for LS-DYNA simulations in d3VIEW. The work started with the import of over 500 frontal crash-test data that includes time-history sensors for vehicle and occupants along with media files such as images and videos. This paper will provide an on overview of the data-engineering involved and the insights gained from the data using ML methods and demonstrate how the data can be used in d3VIEW both as a predictive and as a benchmarking tool.

Uncertainty Sources in WorldSID-50M Dummy

Stefan Kronwitter

Dr. Ing. h.c. F. Porsche AG

Abstract

Full vehicle crash tests are never fully identical in terms of their results, even when conducted with the same configuration. This variability arises from the existence of a diverse set of uncertain parameters, which induces uncertainty in the relevant quantities of interest (QoI). Within the domain of occupant safety, the primary QoI are the load signals captured by anthropometric test devices (ATDs). These signals are influenced by variations originating from the ATD itself, which can be e.g., uncertainty in the initial seating position or uncertainty in parameters affecting the performance of an ATD. This study deals with the identification and characterization of relevant and scattering physical input parameters inherent to the ATD itself, exemplified by the WorldSID-50M. The performance of certain body regions of the WorldSID is assessed through certification tests. By analyzing the hardware certification tests of various WorldSIDs and uncertainty quantification in the signals, insights to the body regions that exhibit significant scattering characteristics are provided. Based on this, sets of physical parameters such as the sheet thicknesses of the rib assemblies or material properties are identified as influential parameters. Their uncertainty is characterized by stochastic models such as gaussian distributions. The identified set of parameters, presumptuously causing the uncertainty in the ATD signals, is validated using two approaches. First, certification tests are simulated using individually adapted WorldSID models and the resulting signals are compared with those of the corresponding hardware tests. The second approach involves forward propagation of the uncertainty in the input parameters to the ATD signals. Herein, the stochastic parameter space is discretized via sampling and the drawn sample is incorporated into a parameterized version of the WorldSID model. After simulating the sample certification tests, the uncertainty in the signals is compared with the observed from the hardware tests. Findings from this study can be used to improve the simulation-based analysis of crash tests by using a WorldSID model adapted to the dummy used in the test. The characterized set of scattering input parameters can be applied in virtual robustness assessments of occupant safety systems.

Emergency Brace Positioning and Injury Risk Prediction of Aircraft Occupants under Impact Loading

Galal Mohamed[1], Qiu-rui. Zai[1], Xiao-peng. Shi[2], Kai.Guo[3], Pei-yao. Li[4], Dong-ling. Li[1]

[1]Arup
[2]Science and Technology Innovation Research Institute, Civil Aviation University of China
[3]School of Safety Science and Engineering, Civil Aviation University of China
[4]Sino-European Institute of Aviation Engineering, Civil Aviation University of China

Abstract

THUMS (Total Human Model for Safety) is a detailed biofidelic finite element model of the human body, which encompass different genders and physiques including detailed anatomical features of the skeletal structure, internal organs, and other soft tissues like skin, flesh, and ligaments. The application of THUMS in numerical simulation offers exciting opportunities in automotive and civil aerospace development in areas such as safety, comfort, and ergonomics. These models will play an increasingly important role in the study of human body kinematics and assessment of injury risks in collision accidents.

The Civil Aviation University of China (CAUC) aims to establish a posture database according to civil aviation standards and perform detailed numerical studies on the biomechanical response of aircraft occupants using THUMS in a variety of simulated impact scenarios. CAUC collaborated with Arup and its software house Oasys LS-DYNA, to help build this posture database and explore the feasibility of using Oasys PRIMER (a world leading LS-DYNA pre-processor) with its comprehensive human body model positioning tools and THUMS positioning metadata to achieve complex emergency braced postures.

This paper describes the entire positioning workflow used to achieve complex brace postures using the multi-stage positioning method in Oasys PRIMER and the *CONTROL_STAGED_CONSTRUCTION keyword in LS-DYNA. Further investigations were made to optimize simulation run times. Using a generic aircraft seat, a series of standard dynamic load cases are performed to predict occupant kinematics and the extent of injury risk to the aircraft occupant. This research will contribute to the wider application of THUMS in the aviation industry and promote biomechanical research into aircraft occupant safety.

A data-driven methodology for the analysis and explanation of system behavior in crash simulations

Janis Mathieu[1,2], Michael Di Roberto[1], Michael Vielhaber[1]

[1]Porsche Engineering Services GmbH
[2]Institute of Engineering Design, Saarland University

Abstract

Attributable to model size and complexity of numerical crash simulations, it is not feasible for the engineers to analyze each area or component in detail with the aim of explaining system behavior, especially when these are not the core subject of investigation. In the field of occupant safety, the main explanatory objective is given by the signals of anthropometric test devices (ATD), as they are relevant for the fulfillment of law and consumer protection regulations. Hence, this study proposes a data-driven methodology to automatically determine deviations in ATD behavior and provide causes for the prevalence helping the engineer to faster understand and interpret simulations as well as to ensure quality. In the proposed methodology, sensor signals are used as a basis to describe the time-dependent system behavior providing a more superficial but mesh-invariant and faster computable description compared to approaches based on geometry data. A deviation score for each signal derived from cluster distances determined by a k-means clustering algorithm is used for detecting and quantifying anomalous system behavior. Conspicuous signals are further investigated by calculating deviation scores for time windows to specify time windows for the event occurrence. Based on a causal graph representing dependencies of sensor signals pre-defined by domain knowledge the derived information is then used to reconstruct the event chain. The validation of the presented approach is conducted by processing a dataset with nine frontal impact simulations, whereby the methodology was able to automatically identify main causes as well as secondary effects.

A Systematic Approach Towards Integrated Safety Modelling for Aerospace Applications – Preliminary Results on Rigid Seat Simulations

Nina Wegener[1], Christoph Sauer[1], Paul Schatrow[1], Matthias Waimer[1]

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

Keywords: *Aerospace, Occupant Safety, Anthropomorphic Test Devices, LSTC Hybrid III, Humanetics Hybrid II*

Abstract

In the aviation sector, the historically evolved crashworthiness requirements prescribe seat certification separately from the airframe structure. Based on historical test and accident data the airframe crash behaviour is presumed in terms of crash pulses, which are applied to the seat structure for seat certification (e.g. EASA CS-23/25.562). Certification authorities have recently started to change the regulations from a prescriptive to a performance-based certification, considering the crash performance with the seats integrated in the airframe structure (EASA CS-23 Amendment 5). With this, occupant safety and structural crashworthiness is combined to an integrated safety approach. Due to the high cost of full-scale testing in the aviation sector, extensive use of simulation is of interest. Modelling methods are continuously being developed for crash loading conditions relevant to aerospace, which significantly differ from automotive ones. The German Aerospace Center (DLR) Institute of Structures and Design has extensive experience in developing simulation methods for aircraft crash analysis. In an effort to develop an integrated safety modelling approach for aviation, a research initiative was launched to incorporate advanced passenger safety considerations.

In the first phase, methodologies for modelling and simulation of current certification tests are developed. This includes seating procedures for finite element (FE) anthropomorphic test devices (ATDs) and seat belt modelling methods. Future steps include the investigation of advanced ATDs and human body models, the implementation of seat and passenger systems into structural crash models and the consideration of novel cabin layouts.

SAE Aerospace Recommended Practice 5765B defines a means of assessing the credibility of computer models of aircraft seating systems used to simulate dynamic impact conditions. This includes supplementary test data of sled tests with aerospace approved ATDs on a rigid seat, which are the basis for validation of the developed methodologies in this first phase of the project.

This paper describes a systematic approach to integrated safety modelling for aerospace applications. Additionally, preliminary results of the first phase for LSTC Hybrid 3 and Humanetics Hybrid 2 ATD models on a rigid seat are presented. The simulation study was conducted using LS-DYNA®.

Virtual testing developments of the LS-DYNA® World-SID dummy model

Alexander Schif, Yupeng Huang, Sebastian Stahlschmidt

DYNAmore GmbH, an Ansys company

1 Abstract

In 2024 the Euro NCAP Virtual Testing Crashworthiness (VTC) procedure for far-side impact is introduced. The LS-DYNA DYNAmore World-SID dummy model will be part of this procedure. Separate qualification criteria must be satisfied for the World-SID model. They are specified in Technical Bulletin TB043-1.
LS-DYNA DYNAmore World-SID version 8 will be the first model with the official certificate to satisfy all the defined criteria of TB043-1. TB043-1 includes three different stages of certification. Normative standard requirements are checked in the first stage. Component test simulations of head-neck and lumbar spine represent the second stage. The last stage includes a new sled test scenario representing the far-side load case. The dummy model must pass all three stages to be fully certificated.

Simulation of woven composite structures considering manufacturing effects

Mathieu Vinot[1], Tolga Usta[2], Christian Liebold[2], Martin Holzapfel[1], Nathalie Toso[1]

[1] Institute of Structures and Design, German Aerospace Center, Stuttgart, Germany
[2] DYNAmore GmbH, an Ansys Company, Stuttgart, Germany

1 Abstract

Keywords: composite materials, manufacturing defects, draping simulation, crash simulation

In recent years, advanced material models have emerged in finite-element codes for the simulation of composite materials reproducing realistic failure mechanisms. Through the increased reliability in simulation, less conservative designs of composite structures have been made possible. However, most of the current numerical solutions are considering the composite material independently of real manufacturing conditions, which can strongly affect the local fibre architecture and properties. To extend the potential of composite structures, it is therefore necessary to enhance the simulation models by including additional information from the production processes. To answer this problematic, many works have focused on the detailed simulation of these processes and on the transfer of information between the simulation steps [1, 2].

The present contribution reports on recent advances in the field of numerical process chain for the simulation of woven composite parts within the project “Digital Fingerprint” at the research campus ARENA2036. It focuses on the simulation and validation of the draping process using a stacked-shell approach (Fig. 1, left and middle) coupled to the material models **`*MAT_249_REINFORCED_THERMOPLASTIC`** [3]. The resulting fibre angles are automatically transferred to a simplified single-layered structural model using the mapping software Envyo® [4]. An exemplary load case for the structural simulation (Fig. 1, right) is investigated and the lately available extension of the draping material card **`*MAT_249_CRASH`** is compared with the material card **`*MAT_262`** [3]. Finally, a probabilistic simulation framework using the software LS-OPT [5] to account for manufacturing tolerances is presented.

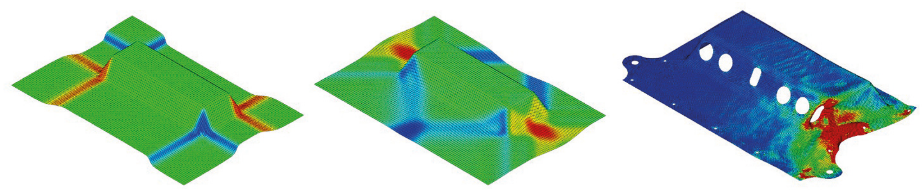


Fig. 1: Resulting shear angle in 0°/90° and ±45° layers after the draping simulation (left and middle) and simulated structural behaviour in the crash simulation (right)

Acknowledgements

This work was funded by the German Federal Ministry of Education and Research within the framework of the ARENA2036 project “Digitaler Fingerabdruck” and managed by the Karlsruhe Institute of Technology.

[1] E.E. Swery, T. Hans et al., Complete simulation process chain for the manufacturing of braided composite parts, Composites: Part A, 378-390, August 2017.
[2] A. Hürkamp, S. Gellrich, T. Ossowski et al., Combining Simulation and Machine Learning as Digital Twin for the Manufacturing of Overmolded Thermoplastic Composites, Journal of Manufacturing and Materials Processing, September 2020.
[3] Livermore Software Technology, ANSYS/LST: “LS-DYNA Keyword User’s Manual – Volume II: Material Models”, June 2021.
[4] DYNAmore GmbH: “Envyo® User’s Manual – Draft”, March 2021.
[5] Livermore Software Technology Corporation, “LS-OPT® User’s Manual: A Design Optimization and Probabilistic Analysis Tool for the Engineering Analyst,” California, 2020.

Modeling composite materials with respect to reinforcement textile construction

P. Böhler[1], M. Gubser[1], J. Ravic[1], O. Döbrich[1,*]

[1] ZHAW Zurich University of Applied Science,
School of Engineering, Institute of Materials and Process Engineering
Technikumstrasse 9, 8401 Winterthur, Switzerland
[*]Corresponding author: oliver.doebrich@zhaw.ch

Keywords: composites, modeling, micro-mechanics, beam-chain, homogenization

1 Abstract

To address the ongoing efforts to virtually design complex composite materials and their high-performance structures and to contribute to the increasing developments towards Industry 4.0, research and development of fiber-reinforced composites is shifting from an experimental domain to a virtually controlled environment. Material models to account for the specific failure mechanisms of layered fiber-reinforced materials have been developed and can be used for largescale numerical structural simulations. However, these models do not account for the individual properties of the reinforcing materials and therefore lack information on microstructural effects and behavior. A near-microscale approach for modeling textile and fiber-based reinforcement structures is presented that allows studies based on the individual textile structure. The digital beam approach is well established in literature for modeling textile processes and textile unit-cells. It allows simulations where the parameters of the manufacturing process are considered, and the resulting reinforcement structure is obtained because of these simulations. This paper addresses such microscale geometric models of reinforcement textiles for use in a virtual composite development chain. The resulting mechanical properties can be predicted for the dry reinforcement textiles by virtual testing and are available as material input parameters for further investigations, such as textile drape simulations. In addition, an approach is presented in which the textile models are used directly for composite modeling. Superimposed models are presented that take advantage of LS-DYNA's ability to Lagrangian-couple solids and provide information about the mechanical behavior of textile-reinforced composites. In addition to the homogenized mechanical properties, additional information such as load distribution within the fibers and other properties based on the specific design of the reinforcement textile can be derived and analyzed.

Composites / Textiles

A user-defined Folgar-Tucker-based fiber orientation material model for compression molding of fiber/polymer-compounds

Dominic Schommer[1,2], Miro Duhovic[1,2], David May[1,2], Joachim Hausmann[1,2],
Heiko Andrä[3], Konrad Steiner[3]

[1]Leibniz-Institut für Verbundwerkstoffe GmbH
[2] Rheinland-Pfälzische Technische Universität Kaiserslautern-Landau, Germany
[3] Fraunhofer ITWM, Kaiserslautern, Germany

Abstract

LS-DYNA® provides an ever-increasing portfolio of material models covering a wide range of material behavior for solving multi-physics problems. The software also provides users the opportunity to implement their own user-defined material models via FORTRAN code, to describe the behavior of very specific materials. In this work, a user-defined material model has been developed to describe the compression molding behavior of sheet molding compounds (SMCs). A SMC is a composite material based on a thermoset resin reinforced by chopped long fibers. During the compression molding of SMCs, very complex material behavior involving elastic compaction and plastic flow (depending on material composition) occurs, which is dependent on the local fiber orientation, temperature and strain rate. One way to describe the processing behavior of SMC materials as simply as possible is using a building block approach. Following the identification of the most relevant material effects, individual building blocks are created containing the respective mathematical solutions (e.g. compaction and plastic flow behavior). Each building block is coupled with its neighboring blocks via its input and output parameters. In this way, complex composite material processing behavior can be simulated using individual building blocks representing individual mathematical models, which can be added or removed as required. Since no material model in the current LS-DYNA solver considers the evolution of fiber orientation (FO) during flow processing of short- or long-fiber reinforced composites, the integration of such an individual building block is presented here. Specifically the influence of FO on a non-linear elastic piecewise plastic material model is considered. In this example, a FO model based on the Folgar-Tucker equation is implemented and applied to Arbitrary Lagrangian Eulerian (ALE) elements. To determine the position of the FO building block within the complete model structure, the required input information to the model and the possible output information from one building block to another is identified. Subsequently, a subroutine containing the corresponding FO mathematical model is written and called within the material function. As a conclusion, it is shown how the implemented FO model behaves during a compression molding simulation and how the resulting FO information can be evaluated at the post-processing stage using LS-PrePost®.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Finite element modelling of textile-soft material interaction using 3D/4D scan data

Ann-Malin Schmidt, Yordan Kyosev

ITM, Chair of Development and Assembly of Textile Products, TU Dresden

1 Introduction

Textile-soft material interaction occurs in various fields. In the field of clothing, the textile interacts with the body. Especially when wearing close-fitting garments, the interaction with the body is important for the function and comfort of the garment. In the example of compression socks, the soft tissue is compressed and deforms the leg (Fig.1: (d)). A model (e.g. FEM) to investigate the textile-soft tissue of the leg helps to design individualized compression socks with defined pressure. The breast - bra - interaction is also an everyday issue for many people with breasts (Fig 1 (c)). The breast moves in relation to the body, and breast and back pain can cause discomfort. The bra interacts with the soft breast and helps to reduce breast movement. A model can help to show which bra pattern and size will have which effect on breast deformation and movement, and therefore comfort.
However, these models would be of great use in the clothing industry, where most clothing development is still based on rigid avatars.
A textile-soft material interaction can also be seen in the field of technical textiles. For example, car seats are made of various soft foams, textiles and seams. The foam is compressed and deformed by the textile and the seams add stiffness. Here the softness of the seat is defined by the cut pattern of the textile, the properties of the foam and their interaction due to the textile tension in the manufacturing process. A model would help to set the parameters prior to manufacturing and create a personalised seating comfort.

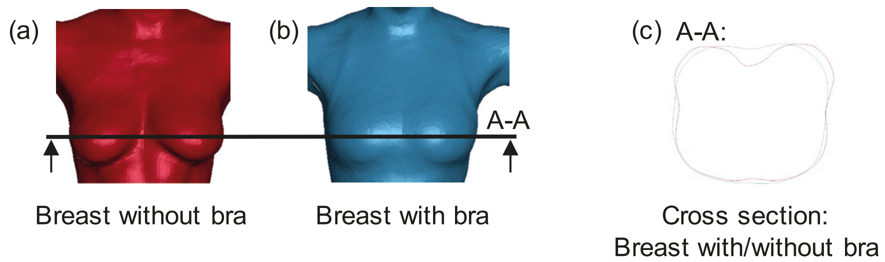


Fig.1: Examples of the deformation due to textile-body interaction: (a) breast without a bra, (b) breast deformed by a bra, (c) cross-section Breast with/without bra

2 Method

In this paper/presentation different FEM models are presented to investigate the textile - soft body interaction. The implementation approach to incorporate 4D scan data of the human body into an FEM model is presented. Different techniques to dress a human body with tight clothing are presented and evaluated in terms of usability, computational cost and accuracy. The deformation of the human body due to the compressive textile is validated using the 4D scan data.

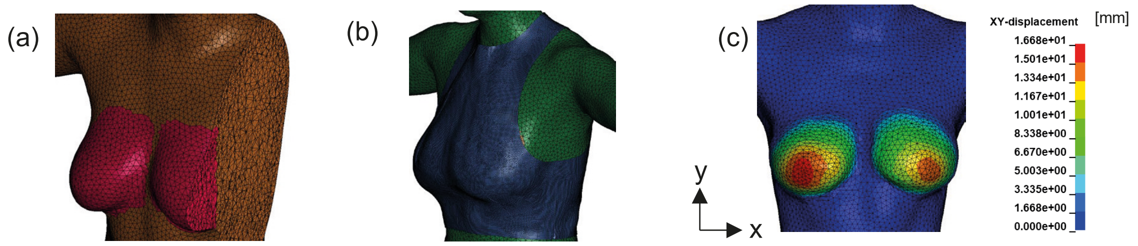


Fig.2: Modelling of textile – soft material – interaction: (a) Model of a breast (b) clothing-soft-body-interaction and (c) pressure distribution at the breast after the dressing simulation

Meshless Methods in Workbench LS-DYNA

Ulrich Stelzmann[1], Yury Novozhilov[1], Alexander Pett[2], Erik Plugge[2]

[1]CADFEM Germany GmbH
[2]Ansys Inc

1 Introduction

CADFEM Germany GmbH are working to create an open library of Ansys LS-DYNA industrial use cases. Two new Industrial Use Cases for Ansys LS-DYNA have been developed by CADFEM in 2023. They focus on using meshless methods and the Eulerian approach for real-world applications: SPG usage with GISSMO damage model to simulate material separation (fig. 1) and SPH/S-ALE solver usage for inertia-dominated fluid-structure interaction (FSI) (fig. 2).

2 Model and Methods

The best practice guides with detailed descriptions of the settings used and the solutions adopted have been developed for both topics under consideration.

- As far as possible, the models were created within the Ansys Workbench and Mechanical environments.
- Most features are already implemented here, which makes these challenging applications much more user friendly to set up.
- Command snippets can be used to insert some missing functions.
- Is it currently necessary to leave the Ansys Mechanical environment only for some postprocessing.
- These examples serve, among other things, to help Ansys integrate such missing features into the LS-DYNA system within Mechanical.

The presentation will first give an overview of the specifics of the two examples. Then, an example of a very flexible infusion bag, filled or to be filled with liquid, is shown in detail. In this context, we compare the two methods, SPH and S-ALE, discuss necessary solver settings for them and derive recommendations for which method is best suited for which application.

3 Figures and Tables

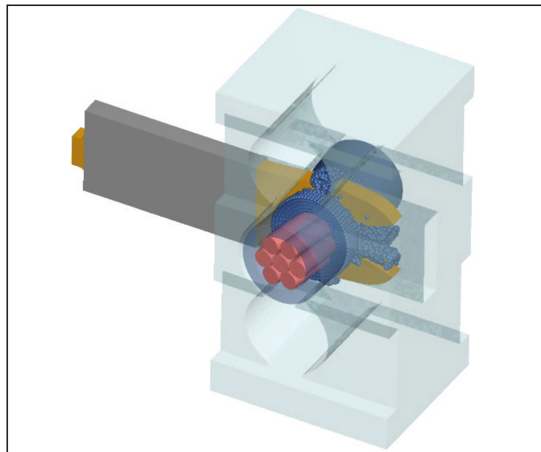


Fig.1: SPG usage with the GISSMO damage model to simulate material separation by the example of cutting wire isolator

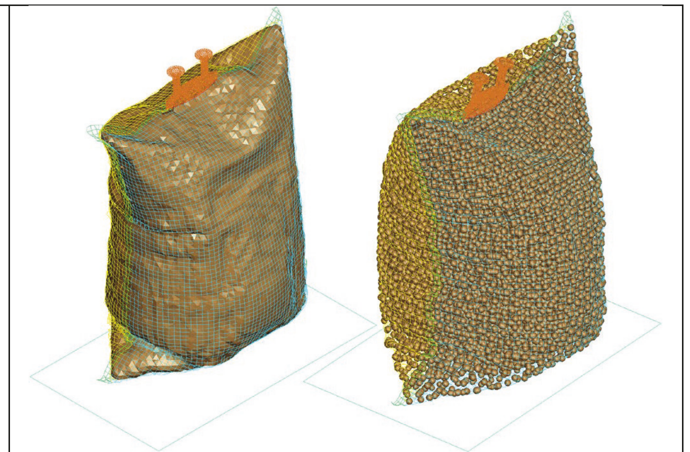


Fig.2: Comparison of S-ALE approach with SPH solver usage for inertia dominated FSI with very flexible container

4 Summary

Developing such test cases for real industrial applications of Ansys LS-DYNA highlights at least two critical aspects: first and foremost, users get ready-made examples and detailed recommendations on how to use the software. This helps to raise the bar for new users to enter the advanced simulation world. Secondly, developing such examples is an excellent stress test of software availability. It allows developers to get detailed, concentrated feedback and to understand the current state of the computational code.

Simulating shot peening: Application on leaf springs

Christos Gakias, Georgios Savaidis

Laboratory of Machine Elements and Machine Design, Aristotle University of Thessaloniki, Greece

Abstract

Shot peening (SP) is a widely used process of surface treatment, based on the impact of small spheres (shots) on the surface of a component. The impact, results on a localized plastic deformation and the development of a compressive residual stress field, that can extend up to a depth of 300-400 μm. This stress field significantly improves the fatigue life of components and prevents the initiation of small cracks. SP treatment can be influenced by various parameters, such as the velocity of shot peening media, many of them, governed by stochasticity. In this study, a structured modelling approach based on the Finite Element Method (FEM) is introduced to account for these stochastic parameters alongside with elastic-plastic behavior and accurately simulate the SP process. Explicit calculation scheme and sophisticated plasticity modelling algorithms are used, that are implemented in LS-Dyna solver. A validation of the model is presented, including a comparison with experimentally derived residual stress profile. These results demonstrate the efficiency of the proposed model and highlights the capabilities of the model to capture accurately the effect of the process. Furthermore, the model was also shown to be capable of examining the effect of the process parameters, such as the shot velocity. Overall, the structured and modular modelling approach that is proposed in this study, can provide valuable information of the results of the SP treatment and can lead to a further understanding and optimization of the process in general.

Simulation Methods

A New Approach to Contacts and Rigid Body Interactions in LS-DYNA

Fredrik Bengzon[1], Anders Jonsson[1]

[1]DYNAmore Nordic AB, An Ansys Company

1 Introduction

Traditionally in LS-DYNA (almost) all contact definitions use a penalty formulation. This means that a penetration in the contact is required to obtain a contact force between interacting entities. It is then up to the user to verify that the penetrations are small enough not to influence the results. The Mortar contacts [1], which have become the preferred choice for implicit analyses, is of penalty type.

Also, the rigid walls (**`*RIGIDWALL`**) use a penalty method in implicit. To find a good penalty stiffness setting may be problematic if solid elements (especially tetrahedra), or soft materials, such as rubber or plastic, are involved. It can be hard to find a good trade-off between reasonably small penetrations and implicit convergence.

In this paper, we present a new approach to rigid walls and rigid analytical shapes (**`*CONTACT_ENTITY`**, see Fig.1) using an augmented Lagrange formulation [4] that strictly enforces zero penetrations. The implementation is intended for implicit analyses. The rigid wall implementation is a re-formulation of the existing functionality, while the implementation of **`*CONTACT_ENTITY`** for implicit analysis is new.

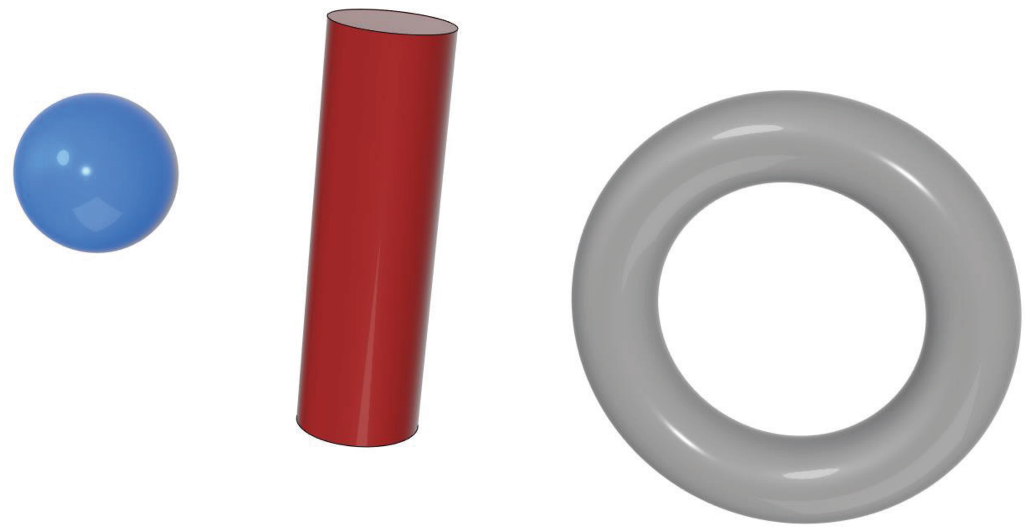


*Fig.1: Examples of pre-define analytical shapes that can be created by **`*CONTACT_ENTITY`**.*

We also present a new small-sliding contact algorithm dedicated to implicit analyses using the same approach. In many analysis applications where implicit is the dominating solution technique, for example static stress analyses and high cycle fatigue analyses, only small relative motions are expected between the parts in the model. For such analyses, small-sliding contact is sufficient, and the small-sliding assumption, meaning that the contact pairs need not to be updated after one initial contact search, simplifies the numerical implementation in many ways. This small-sliding, augmented Lagrange contact uses the keyword **`*CONTACT_CONSTRAINT`** and is activated by the implicit accuracy ***`IACC`*** variable of **`*CONTROL_ACCURACY`**.

The use of Lagrange multipliers will rearrange the structure of the stiffness matrix, generating a saddle point problem. For this type of problem, the traditional quasi-Newton approach of BFGS in LS-DYNA is inefficient. As an alternative, methods of Jacobian-Free Newton-Krylov (JFNK) type [3] can be used to improve the nonlinear convergence, and the implementation is also outlined.

2 Results

As an example of a planar rigid wall, the Taylor bar impact analysis [2], is presented, see Fig. 2 for a comparison of the rigid wall reaction forces to previous implicit implementation and explicit (which should be seen as the reference solution in this case). It can be noted that the previous penalty method (blue curve in Fig. 2) misses the initial force peak, while the augmented Lagrange method follows the trend from the explicit (reference) solution.

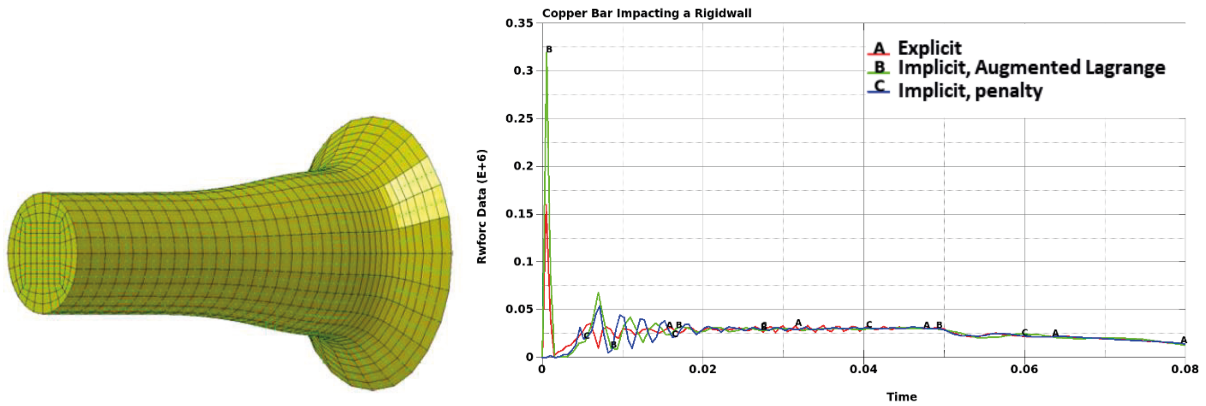


Fig.2: The left image shows a cylinder impacting a rigid wall. The right image shows a comparison of the reaction forces for the explicit solution (red) and the implicit solution using both the penalty method (blue) and the augmented Lagrange formulation (green).

Results for other rigid wall types, constrained contacts, and contact entities will be given in the complete conference contribution.

3 Outlook

For implicit analyses, the Lagrange multiplier approach to rigid walls and analytical rigid shapes (**`*CONTACT_ENTITY`**) will replace the previous implementation of these features. The development of constraint-type contacts for implicit analyses, possibly extending the functionality to large sliding, will continue based on customer input. We believe that in for example process simulations, like extrusion of plastics or forming of carton packages, the use of augmented Lagrange contacts may deliver results even closer to reality, since the details of the tooling may be challenging to fully resolve using a penalty method.
Also, further research on the alternative JFNK methods to improve the nonlinear solver for handling the saddle-point type of problems associated with the Lagrange multiplier approach will be required.

4 Literature

[1] Borrvall, T., "Mortar contact algorithm for implicit stamping analyses in LS-DYNA ", 10th International LS-DYNA conference, 2008
[2] Wilkins, M. L., and Guinan, M. W., “Impact of cylinders on a rigid boundary”, Journal of Applied Physics, 1973, **44**:1200-1206
[3] Knoll, D., and Keyes, D., “Jacobian-free Newton-Krylov method: A survey of approaches and applications”, Journal of Computational Physics, 2004, **193**:357-397
[4] Renard, Y., “Generalized Newton’s methods for the approximation and resolution of frictional contact problems in elasticity”, Computer Methods in Applied Mechanics and Engineering, 2003, **256**:38-55s

Monopile damage assessment from impact with a sub-sea boulder, using an LS-DYNA methodology

David McLennan[1], Francesca Palmieri[1], Andrew Cunningham[1], James Go[1], Richard Sturt[1], Paul Morrison[1], César Tejada[2], Georgios Perikleous[3], Jacob Brandt[3], Mikkel Lubek[4]

[1]Ove Arup & Partners Ltd, London, UK
[2]Ørsted A/S, London, UK
[3]Ørsted A/S, Copenhagen, DK
[4]Ørsted A/S, Skærbæk, DK.

Abstract

During the installation of monopiles (MP) for the offshore wind turbine industry, there is a site-specific risk of impact with submerged sub-sea boulders, depending on the nature of the site geology. Factors such as boulder size, boulder depth, soil properties, and impact angle, will influence the level of damage experienced by the MP due to the boulder impact.
For assessing this damage, an LS-DYNA finite element (FE) methodology is demonstrated in this paper, with the MP modelled using thick shells, and the resistance provided by the soil modelled using discrete beams. For tubular MP geometry, the discrete beam approach has the advantage of reducing the analysis run-time by an order of at least 100x, compared to an approach which requires the modelling of a solid element mesh soil block (such as the established approach of a Coupled Eulerian-Lagrangian mesh). Material properties of the soil are obtained from real-world soil data, and characterised via closed-form solutions and supplementary FE analyses. The methodology makes use of a sequential introduction of soil elements and various contact definitions, to allow for a physically representative interaction between the MP and soil.
The analysis begins with the toe of the MP about to make contact with the boulder, and ends at the final MP embedment depth, having interacted with the boulder for the duration of the remaining portion of the installation sequence. A real-world MP installation involves thousands of individual hammer impacts, but in this study the MP is pushed downwards with a constant velocity throughout the analysis. This modelling simplification has been justified via sensitivity studies, and is another technique which considerably reduces analysis runtime, compared to the alternative of modelling each individual hammer impact of the installation sequence. It results in a 10-metre push having an analysis runtime of approximately 3 hours on 32 CPU.
Results from this LS-DYNA methodology are obtained for a representative range of site-specific input parameters. These results are used to influence on-site MP installation strategies, to avoid locations with known boulders above a certain size, and hence reduce the risk of MP damage and refusal.

Using ISPG technique for Simulating Solder Reflow and TIM Compression in IC Packages

Daniel Vilyatser, Arkaprabha Sengupta, Ayush Kumar, Xiaofei Pan

Ansys Inc.

Abstract

Accurate simulation of solder reflow processes and the behavior of thermal interface materials (TIM) during Integrated Circuit (IC) packaging process is essential for ensuring the structural reliability and thermal behavior of electronic devices. In this presentation, a new approach utilizing the adaptive Incompressible Smooth Particle Galerkin (ISPG) method in LS-DYNA is presented. This method aims to simulate solder reflow processes and TIM spreading behavior during BGA flip-chip die attachment processes and overall IC assembly. The ISPG method combines the advantages of incompressible fluid simulations and SPG (particle) techniques using a Lagrangian approach. Existing methods such as Surface Evolver or Fluid-Structure Interaction (FSI) techniques often struggle to accurately represent the interactions between solid composites and fluid-like molten solder balls during their reflow process. They also face challenges in maintaining efficiency and computational feasibility when simulating complete packages. This work demonstrates the effectiveness of the new approach called Adaptive ISPG which has been recently implemented in LS-DYNA. This approach efficiently captures the relevant physics involved in solder reflow and TIM spreading processes. The ISPG technique utilizes a Lagrangian particle-based approach to solve the Navier-Stokes equation, specifically designed for modeling molten solder and fluid-like TIM used in IC packaging. This technique seamlessly integrates with the structural physics capabilities within the LS-DYNA solver. The evolution of solder and TIM shapes is determined by the material's surface tension and the boundary conditions set by the structure. The adaptive feature of ISPG further enhances its capabilities, enabling improved flow behavior in narrow gaps and around sharp corners, which provides more accurate and realistic simulations of the solder and TIM behavior. This novel workflow offers significant potential in optimizing new parameters of packaging assembly process through simulation-guided adjustments, enhancing the reliability and performance of IC packages.

Comprehensive Digital Twin of a Beverage Can Body Forming Process and Performance Evaluation

Sebastijan Jurendic, Maximilian Weiser

Novelis Deutschland GmbH, Novelis R&D Centre Göttingen

Abstract

Novelis is a world leader in aluminium flat rolled products and a major supplier to the beverage can-making industry. As such, Novelis is deeply involved in supporting the can-making industry to help shape a more sustainable future together. Reducing the amount of metal used for each beverage can is a major driver for improving sustainability of the beverage can packaging, thus Novelis is actively investigating and developing state of the art modelling tools and approaches to support further optimization of the beverage can product. Can-making is an exceedingly high speed and capital intensive process, where traditional prototype based development can be prohibitively time and cost intensive. In order to facilitate beverage can body geometry and process optimization, a comprehensive suite of finite element models has been developed, comprising a virtual can line encompassing all aspects of can body forming and performance analysis. This includes all forming operations, such as cup drawing, re-drawing and bottom forming, as well as accounting for changes in material properties during thermal processing of the can body. The deformation history is passed forward through all the forming operations to the final formed can body, which is used in subsequent performance analysis. The entire suite of can models has been developed using a 2D axisymmetric and a 3D shell element modelling approach in parallel. The 2D axisymmetric analysis focuses on local deformations and stress-strain distribution for in depth formability analysis. For performance test simulations, where the deformation modes can be asymmetric, a 3D shell based modelling approach is required. To evaluate the performance of a can body, several tests common in the can-making industry have been recreated. This includes the dome reversal pressure test, drop test, and the axial load test. The simulations of the tests are performed on models containing the full stress-strain deformation history from the set of forming simulations to provide an accurate representation of the formed can body. The forming and performance models have been validated to varying degrees, using cross-sections of formed can bodies and laboratory performance tests. Good general agreement between the tests and the simulations has been achieved for the available test data. In this work, the forming and performance process chain and selected results will be presented.

High-throughput Simulation and Machine Learning Approaches for Can Body Desig

Maximilian Weiser, Sebastijan Jurendic

Novelis Deutschland GmbH, Novelis R&D Centre Göttingen

Abstract

Novelis is a world leader in aluminium flat rolled products and a major supplier to the beverage can-making industry. As such, Novelis is deeply involved in supporting the can-making industry to help shaping a more sustainable future together. Reducing the amount of metal used for each beverage can is a major driver for improving sustainability of the beverage can packaging, thus Novelis is actively investigating and developing state of the art modelling tools and approaches to support further optimization of the beverage can. High-throughput simulation and machine learning approaches have been developed to enable large-scale parameter studies and accelerate product optimization. Leveraging a previously developed suite of finite element models encompassing an exhaustive set of can body forming and performance simulations, all relevant aspects of the can body forming process have been parametrized, including tooling geometry, material properties and process parameters. Can body performance metrics, such as dome reversal pressure, maximum thinning of the material and maximum axial load, are then evaluated on models containing the full forming deformation history. The entire process of model generation, sequential execution of various forming simulations and can body performance analysis has been fully automated, enabling the generation of a large dataset, containing over 15,000 unique simulated can variants. A gradient boost machine learning model has been fitted to the dataset to generate a surrogate model to be used in optimization and parameter studies. The main performance parameters of the can were successfully captured by the surrogate model with sufficient accuracy to provide guidance on can body design. Finally, a web-based application with a graphical user interface has been developed and deployed, providing simple access to the information to a wide range of users in technical and non-technical fields with real time predictions of can body performance metrics.

Update of a Linear Regression Model to Predict Forming Limit Curves from Tensile Test Data

G. Trattnig, M. Schmid, H. Pauli, L. Wagner, A. Grünsteidl

voestalpine Stahl GmbH, Linz, Austria

Forming limit curves (FLCs) are widely used for the feasibility analysis of deep drawn steel components and the final tool design. The experimental determination of the FLC is usually based on Nakajima tests, which are evaluated according to the ISO 12004-2 standard with the intersection line method. In recent years the additional determination with the time dependent method [1] is used since it more accurately describes the increased forming potential of modern high ductility steel grades found in practical experiments.

The experimental determination of the FLC is time consuming and expensive. Therefore, models are used to predict the FLC, based on simple and frequently available tensile tests results. They enable the easy and quick identification of the forming potential of newly developed steel grades. These models can also be used to analyze the influence of sheet thickness and material properties on the FLC. The implementation of the Abspoel model [2] in different finite element (FE) solvers shows the practical demand.

voestalpine developed a linear regression model which predicts the minor and major strain for each FLC sample geometry, published in 2012 [3]. This model was based on 332 FLCs, determined by the intersection line method for the steel grades available at that time. Now, ten years later, the useable database increased to 771 FLCs determined by the intersections line method and 254 FLCs determined by the time dependent method. The database includes novel steel grades, such as high ductility and martensitic steels, and steels with tensile strengths up to approx. 1400 MPa. It also contains FLCs of innovative hot rolled steel grades with sheet thicknesses up to 4 mm.

In this paper the process of data selection and model estimation is shown. Different subsets of parameters from available material test data, like sheet thickness, yield strength, tensile strength, uniform and ultimate elongation are used as predictor variables.

With this updated model, FLC predictions like shown in Fig. 1 are possible. The graph presents an accurate description of the experimental FLC of a complex phase steel with improved formability, independent of the FLC determination method. To get a better assessment of the models potential and limitations, the model FLCs are compared with a comprehensive set of experimental FLCs. Those experimental FLCs are freely distributed by voestalpine in form of LS-Dyna material cards.

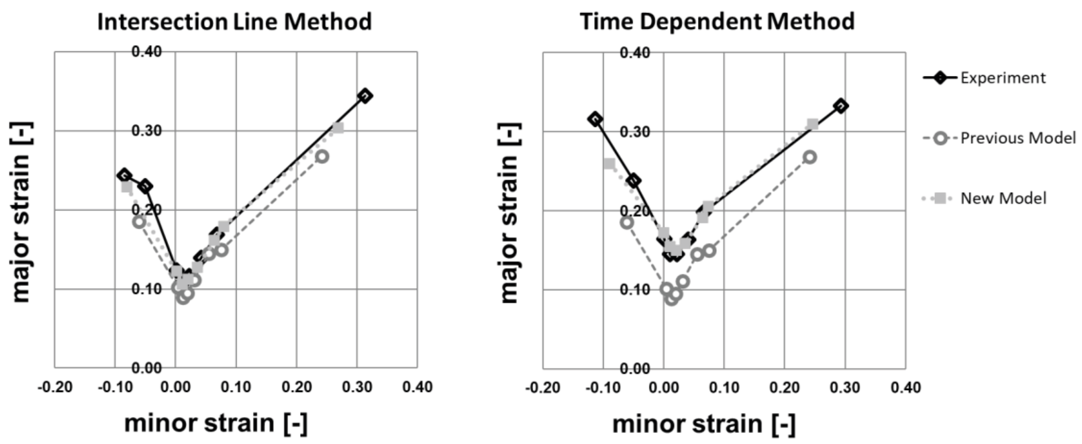


Fig.1: FLC of a CR780Y980T-CH GI with 1.22 mm thickness, determined by the intersection line method (left) and the time dependent method (right). The model prediction of the previous model (Schmid et al. 2012) and the new model is shown as an example.

[1] Hotz, W., Merklein, M., Kuppert, A., Friebe, H., Möller, T, Klein, M.: „Time dependent FLC determination – Comparison of different algorithms to detect the onset of unstable necking before fracture", Proc. of IDDRG 2012, Mumbai, India, 258-264
[2] Abspoel, M., Scholting, M.E., Droog, J.M.M.: "A new method for predicting Forming Limit Curves from mechanical properties", J. Mater. Process. Tech. 213, 2013, 759-769
[2] Schmid, M., Hackl, B., Pauli, H., Grünsteidl, A.: "A linear regression model to predict FLCs from tensile test data and sheet thickness", Forming Technology Forum 2012. June 5-6, IVP, ETH

Simulation of Hollow Embossing Rolling for Bipolar Plate Forming using LS-DYNA©

Franz Reuther[1], Verena Psyk[1], Verena Kräusel[1]

[1]Fraunhofer Institute for Machine Tools and Forming Technology IWU, Chemnitz

Abstract

Hollow embossing rolling constitutes a promising forming technology for metallic bipolar plates due to the high achievable production rates. The simulation-based process optimization is impeded by the incremental forming character and modeling of fine channel structures, which leads to large model sizes and computation times. This paper presents a shell-based finite element modeling approach using LS-DYNA© for bipolar plate forming simulation. Essential boundary conditions of the modeling are discussed, and recommended setting parameters are derived.

Keywords: hollow embossing rolling, roll forming, simulation, fuel cell, bipolar plate

Recent combined developments in LS-DYNA and Ansysform

Jeanne He[1], Xinhai Zhu[2]

[1]Forming Simulation Technologies LLC, Northville, USA
[2]ANSYS, Livermore, USA

Abstract

Ansysform is a relatively new preprocessor that was developed in order to provide the users with an intuitive interface for sheet metal forming simulations with LS-DYNA's robust solver. It is a big advantage to have the same person in charge of the development of Ansysform and the manufacturing functions of LS-DYNA as this results in a close tuning of the preprocessor and the solver to each other. The current project is an example of that close cooperation which has bore fruit many times previously already.

In an industrial sheet metalforming process, the first operation consists in clamping the blank by closing the binder to the die. This is a force controlled process and the ultimately applied force is an important parameter of the process. In the simplest case the binder is brought down and clamps the blank between the binder and the fixed die. In this case the force applied by the binder is known as a function of time or at the very least the ramp-up time of the force is known and is of the order of tens of milliseconds, thus well within the range of an explicit simulation.

Historically, numerical simulations of this binder closing process using a force driven load have not been successful and resulted in bulging or rippling of the blank due to dynamic effects in the model which could be due to any of the numerical techniques typically used in stamping simulations such as mass scaling, mesh adaptivity or others yet (in most cases the ramp time was also scaled down in order to reduce the computation time) . The classical solution was to use a displacement driven load to close the binder. However this would often not result in the correct level of binder force and thus reduced the overall reliability of the numerical results.

A recent development is LS-DYNA has resolved the issue. It is now possible to use a force driven load to close the binder thus resulting in the correct binder force automatically without generating any parasitic dynamic effects in the blank. This paper will explain the new methodology and highlight it's advantages by showing a number of examples .

Developments in *MAT_WINFRITH_CONCRETE and Application to Modelling of Segmented Concrete Tunnel Linings

Richard Sturt, Arup Gianmarco Montalbini

Arup

Abstract

Tunnels constructed by Tunnel Boring Machines are lined with precast reinforced concrete segments. The joints between the segments must resist a combination of compressive load and bending moments induced by non-uniform pressure from the soil. Failure modes to be considered during design include splitting of the joint face under concentrated compressive load, and tear-out of bolts or spalling of the exterior or interior faces of the segments under bending actions. The capacity of the segments to resist such failures may be explored in detail using LS-DYNA's nonlinear concrete models. The paper includes examples from an investigation into a tunnel that partially collapsed shortly after construction. The failure modes revealed by the site investigation of the collapsed tunnel matched well with those shown by the model. The models described in the paper make use of some recent enhancements to the "Winfrith" concrete material model (*MAT_084) that will be released in R15, such as confinement-dependent post-yield softening in compression. The material model is now capable of simulating all the likely failure modes of tunnel segment joints including spalling, bearing and shear failures as well as splitting of the joint face. Applications to steel fibre reinforced concrete (SFRC) as well as plain concrete are described.

Modelling the dynamics of well perforation

Marco Serra, Engenya GmbH

Abstract

One of the crucial steps in the completion of many oil wells is their perforation to establish communication with the target reservoir. This is a very short duration, high energy event in which a series of shaped charges is fired to produce corresponding perforations into the hydrocarbon-bearing formation. This event gives rise to violent pressure dynamics that die out within a second or two, depending on the specific completion design. During this time, the nature of the pressure dynamics and resulting fluid response determine the initial quality of the communication between the well and the formation, which has significant consequences for the overall productivity over the well's lifetime, as well as the integrity of the tool string components. This paper describes the development and validation of an approach to simulating well perforation that was initially driven by the need to understand the causes of failures experienced in the field during some of these highly energetic events. The nature and scale of the problem presents some modelling challenges, but the value of the increased depth of insight provided by the LS-Dyna solution over legacy software is clear and makes a strong case for running enhanced predictive simulation as a matter of course. Applying a systems perspective to deliver a simulation capability focused on identification of risk to structural integrity as well as completion design optimization, has delivered a tool that remains flexible and scalable in all of its components.

Simulation Misc. I

Implicit and explicit analysis of a forklift truck operating in an ISO container

Sean Duvall

Nuclear Transport Solutions

Abstract

Radioactive material and other dangerous goods are frequently transported in specialised ISO containers. One such ISO container is the NTS designated 3573 ISO container. In order to place items, typically drums strapped to pallets, inside the ISO container, it is necessary to operate a forklift truck inside the ISO. One of the operating companies required a heavier forklift truck for a particular operation, which exceeded the normal operating mass. LSDYNA was used to assess if there would be any permanent damage that could affect operation or challenge the welds within the ISO container. The analysis considered both static and dynamic loads as well as predicting the upper limit on the mass of the fully laden forklift truck before damage exceeded a given criteria. This presentation gives details of the approach used for dynamic, static and limit analysis for a forklift truck moving in and out of the ISO container.

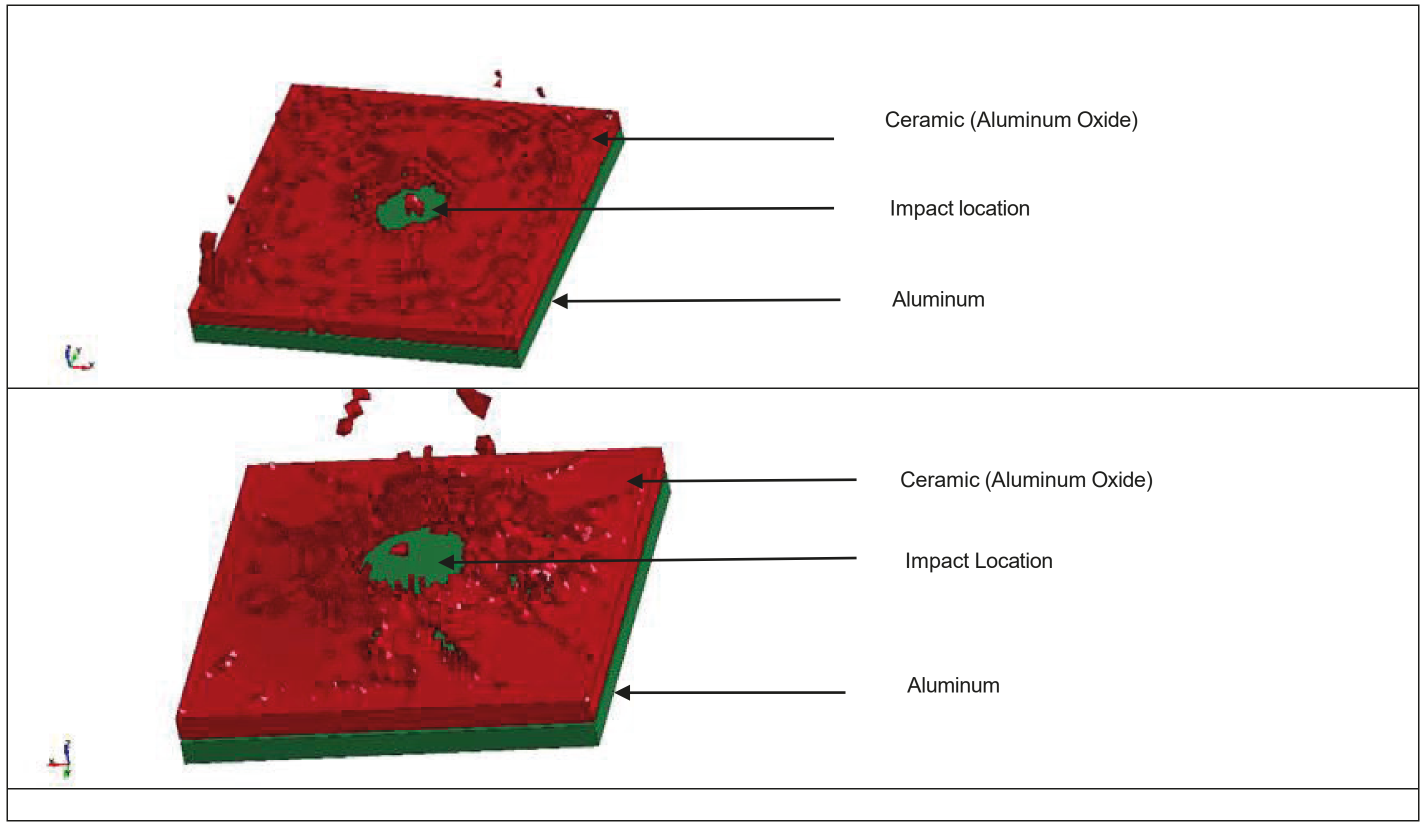


Fig.1: Comparison of Ceramic Damage between a Perfect Transmission Boundary and an Imperfect Transmission Boundary

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Introduction to Ansys LS-DYNA Quality Assurance Activities

Rocco Bombardieri[1], Juan Pu[1], Fabian De Leon[1], Luis Vidal[1], Mohammed Qureshi[1]

[1]Ansys LST

The objective of the presentation is to provide an overview of the activities carried out by the LST Quality Assurance (QA) team in supporting LS-DYNA and related Products. Over the past year, significant changes have been made in the activities of the LST QA team in order to expand the support for all LS-DYNA-related products.
The primary focus of the team is to improve the quality of LS-DYNA solver and ensure customers satisfaction in all aspects during its continuous development. This requires work in various areas. On one hand, the LS-DYNA Test Library is under continuous development, with the addition of new tests to cover both new and existing features and options. These tests are categorized based on LS-DYNA features and run time. On the other hand, in order to maintain the code stability on a daily basis and help catch the bugs introduced due to code changes, a fully automated Post-Commit Regression Process has been developed. This process gathers all the daily code commits, builds LS-DYNA executables and runs solver regressions, ensuring thorough verification of all supported parallel applications (SMP, MPP and HYB) and precisions (SP & DP) on different Linux platforms. Automated regressions are run daily through ADO build pipelines, results are investigated, sorting non-pass cases by type (i.e., errored subroutines, error messages, benchmark discrepancies etc.), Git-Bisect is performed to identify the daily commit(s) that introduced new errors and by comparing the results with the previous runs. An automated email is sent daily with the summary of the results to the development teams to promtly resolve the bugs. This Post-Commit process has been extended to all active LS-DYNA branches, including R11, R12, R13, R14, and DEV.

During LS-DYNA pre-release stage, release candidates built by the release team undergo more extensive and comprehensive testing and evaluations, including license testing, regressions across different MPIs (Intel MPI, Platform MPI, OpenMPI for Linux, Intel MPI, Platform MPI and MS MPI for Windows), compilers (Intel and aocc) and platforms.

Ongoing activities include memory profiling to monitor the LS-DYNA solver's efficiency. Regular memory profiling and leaks detection are conducted using the regression strategy mentioned earlier and with the use of Valgrind. For this purpose, executables are built with special flags and regressions are run with a selected subset of test cases that cover the main features of the code. Efforts are underway to extend features coverage.

In their continuous commitment to support all products within the LS-DYNA ecosystem, the team is developing testing strategies for Ansys Forming (solver and GUI), LS-OPT (solver and GUI), LS-TASC (solver and GUI), LS-PrePost and LS-Rreader. Products that have dedicated solvers undergo similar testing procedures as described earlier. Additional activities can be carried out as needed, in collaboration with the development teams. For Graphical User Interfaces (GUI) testing, automated tests are established when an API is available, covering all exposed functionalities. Interactive tests are utilized during hardening and pre-release stages to ensure the quality of the user interface.

Furthermore, the LST QA team is actively working to expland the library of the Verification Manuals for the LS-DYNA solver capabilities. The Verification Manuals collection in the last release covers LS-DYNA structural, thermal, EM, ICFD, and ISPG features. These manuals replicate applications with closed-form solutions or those available in literature. Models are created for this purpose, and the results calculated with LS-DYNA are compared against benchmarks. The manuals, including application descriptions, result comparisons and input decks, are released to users. Tests are added to the Test Library and maintained accordingly.

Lastly, the LST QA team contributes to the activities of the LS-DYNA Aerospace Working Group (AWG) by maintaining the open-source test cases, generating relevant reports, and providing updated results for each official release.

Current Strategies of Glass Modelling with MAT_280

C. Wilking[1], T. Graf[1], M. Feucht[2]

[1]DYNAmore GmbH, an Ansys Company
[2] Mercedes-Benz AG

The influence of pretension on reinforced concrete beams subjected to fuel tanker explosions

Joel Smith[1], James Bache[2], Jessica Klimenko[2], and Ben Smith[1]

[1] Arup Australia Pty Ltd, Sky Park, One Melbourne Quarter, 699 Collins St, Docklands, VIC, 3008, Australia;
[2] Arup Australia Pty Ltd, Level 5, Barrack Place, 151 Clarence St, Sydney, NSW, 2000, Australia.

1 Abstract

Pretension in reinforced concrete beams is commonly used within the construction industry to provide sufficient precompression to the tension face of an element, such that static dead and live loads develop low or no tension stress within the concrete components of the element. While this is advantageous for conventional structural design, allowing the element to carry more load over longer spans, beam elements subjected to significant uplift loading, such as those experienced during an accidental explosion, can develop additional tensile stresses on the element's top surface.

This paper presents an investigation into the use of pretensioned beam elements to construct a pedestrian structure over an existing dangerous goods transportation route. Utilising LS-DYNA, the impact of a fuel tanker explosion occurring underneath on the integrity of the structure and the potential risk to life safety are interrogated.

Pretensioned beam elements have a distinct construction sequence that affects the internal stress state prior to the application of accidental blast loads. To capture this, the pretension, placement, and design load applications occur in distinctly separate phases.

Computational Fluid Dynamics (CFD) analysis with Viper::Blast was completed on the beam mesh to calculate the blast loads over the span of the element, while also considering reflected surfaces from nearby structures and the roadway underneath the fuel tanker.

Post blast loading and rebound phases in the model have been interrogated against the typical failure modes identified for a reinforced concrete beam element, such as flexure cracking, compressive crushing of concrete and yielding of reinforcement components. Finally, a sensitivity study was conducted on under pretensioning the element relative to the structural engineering design. This was intended to better understand the potential failure modes under uplift blast loading and potentially find a better balance between the structural design requirements and resilience under accidental blast loads.

Verification of Concrete Spalling Simulation with Wave Propagation Theory

Sunao Tokura[1],

[1]Tokura Simulation Research Corp.

1 Introduction

Research on failure modes of concrete structures under impact load is very important for evaluating the safety of structures. Depending on the geometry and impact energy of the projectile, several kinds of the failure mode including surface failure, penetration, and spalling on the back surface. In this paper, we focused on the spalling failure, and tried to compare the fracture area and the scattering speed of the fragments after the fracture predicted by the wave propagation theory and the results of impact simulation using Ansys LS-DYNA. As a result, it was confirmed that LS-DYNA has the ability to reproduce theoretical predictions.

2 Analysis model

A simple cubic concrete block with a side length of 200 mm was modeled. The number of one-point integration solid elements is 64,000. The model geometry is shown in Fig.1.

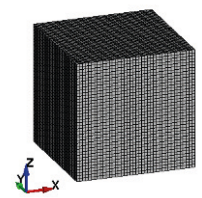


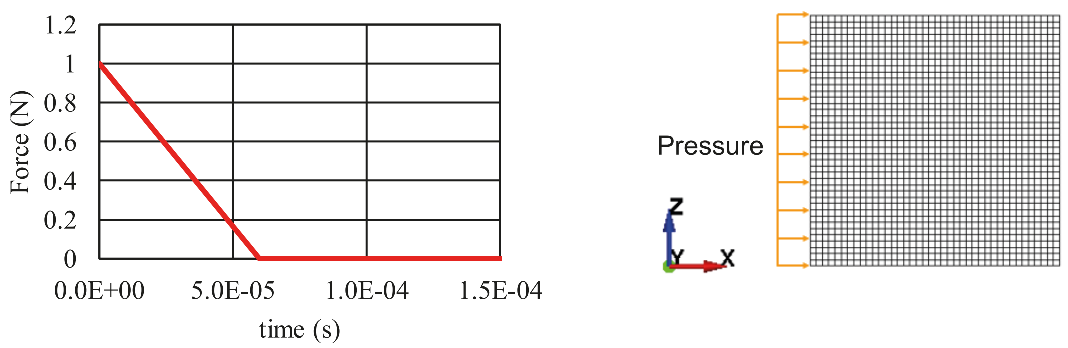
Fig.1: Concrete block model

3 Material properties

`*MAT_WINFRITH_CONCRETE` (**`*MAT_084`**) was used to define the concrete material. Several failure criteria were investigated and the location of cracks was displayed on the analysis model using D3CRACK file generated by this material model.

4 Analysis conditions

The concrete model is placed in space freely. No constraint condition is defined in the model. Triangular impact load was applied on a surface of the model using **`*LOAD_SEGMENT`** keyword as shown in Fig.2.



(a) Impact load history (b)Impact load (pressure) distribution

Fig.2: Impact load

5 Results and discussion

The result of the simulation is shown in Fig.3. A stress wave was generated from the side to which the impact load was applied, propagated through the concrete, and reflected on the opposite side. As a result, it was observed that cracks were generated at a position slightly away from the surface due to the negative pressure of the reflected wave. It was confirmed that the position of these cracks almost coincided with the position predicted by the wave propagation theory.

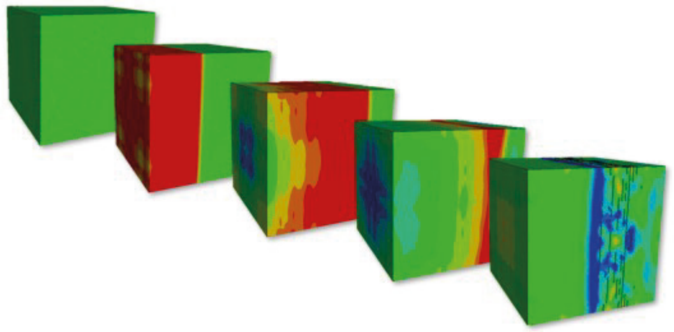


Fig.3: Stress wave propagation and generated cracks

6 Conclusions

Through this investigation, it was confirmed that the location of spall failure due to impact load and the behavior of fragments could be reproduced by LS-DYNA and the concrete model implemented in it, according to the wave propagation theory. This technique is expected to provide a highly reliable simulation of the impact failure process of actual concrete structures.

A Study of RC Beam-Column Against Close-in Blast Loading Using 3D ALE Mapping to S-ALE Technique

Zoey Lim Siew Fern[1], Sun Jian Yun[1], Heng Zi Jing[1], and Dr. Ang Choon Keat[1]

[1] Prostruct Consulting Pte Ltd., 38 Cleantech Loop, #01-31, Singapore 636741.

Abstract

A two-stage loading of a RC beam-column involving blast load and post-blast compression load to determine the residual axial capacity. This paper presents a mesh sensitivity study on the RC beam-column against close-in detonation using 3D ALE mapping to 3D S-ALE model technique. The mesh sensitivity study is performed to investigate the effect of different mesh sizes on 3D ALE mapping to the S-ALE model. The simulation results were compared to a full-scale blast test of a RC beam-column test specimen.

Keyword: ALE Mapping, RC beam-column, Close-in detonation, S-ALE

Modelling of laser impact on typical composites aeronautical structures for bonding quality assessment

Julie Rougier (1), Teddy Maillot (1), Charlotte Michel (1), Vincent Lapoujade (1)

(1) Dynas+, 5 avenue Didier Daurat 31400 TOULOUSE France, t.maillot@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 LAST (Layer adhesion test) technology appears to be a promising alternative as a non destructive control mean to asses bonding quality. The laser impact creates a plasma that induces shock waves propagation in the structure. 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, along with PIMM laboratory, CNRS - UMR 8006, and RESCOLL, 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)

Two years after a first paper on this project, many improvements and new methodologies have been done. This new paper focuses on the numerical approach in LS-DYNA to improve representativeness of the finite element models. Integration of new LS-DYNA developments in composite modelling, combining with LS-OPT studies will be presented in different load configurations. Additionally, confrontation to experimental results will comfort the methodology developed during this project.

Similar part identification integrating machine learning approaches with a SDM workflow

Marcelo Pintado[1], Marko Thiele[1], Akhil Pillai[1], Uwe Reuter[2], Florian Moldering[3],

[1]SCALE GmbH
[2]TU Dresden
[3]Audi AG

1 Summary

Machine learning approaches for geometric part recognition have been evaluated with 3D automotive data [1], obtaining a near-perfect accuracy performance to find the exact part (one-to-one match) and with a small data proportion (one vehicle). However, engineers are also looking for plausibility for similar geometric matches to be implemented in a productive assembly line, obtaining not necessarily the same part but all similar shapes.

By doing so, we enhance the CAE process for advanced vehicle development in the early stages, using machine learning in the design analysis for the process of Carry Over Part (COP), comparing the information from unlabeled vehicle developments (automobiles from other OEM companies) and filtering similar design (or production facilities, specifications, etc.) to assist the automotive manufacturing process (see Fig. 1).

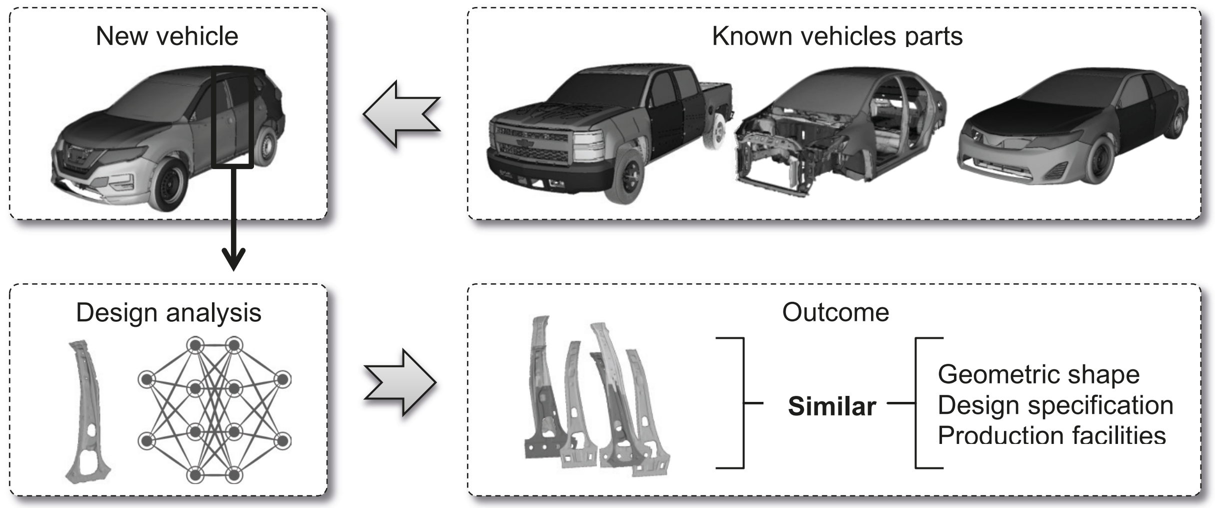


Fig.1: Research's motivation and objective (Vehicles are courtesy of the CCSA team [2]).

This work integrates a machine learning workflow with HPC resources and automotive CAD data using SCALE.model [3], a powerful data management system solution for large CAE databases. Herewith, we investigate the feasibility of different techniques in 3 main sections. The first section is the input preparation, which starts with a 3D point cloud transformation and includes data augmentation. Some approaches were investigated, which led to the iterative closest points (ICP) algorithm [4], which aligns all point clouds similarly and generalizes the model.

The second section is architecture customization, which involves adjusting the machine learning algorithm [5] for the specific use-case. The third section is the training customization, modifying the process with a proposed loss function exploiting the geometric similarities [6] between automotive parts, with a non-supervised class clustering and smooth labeling (see Fig. 2).

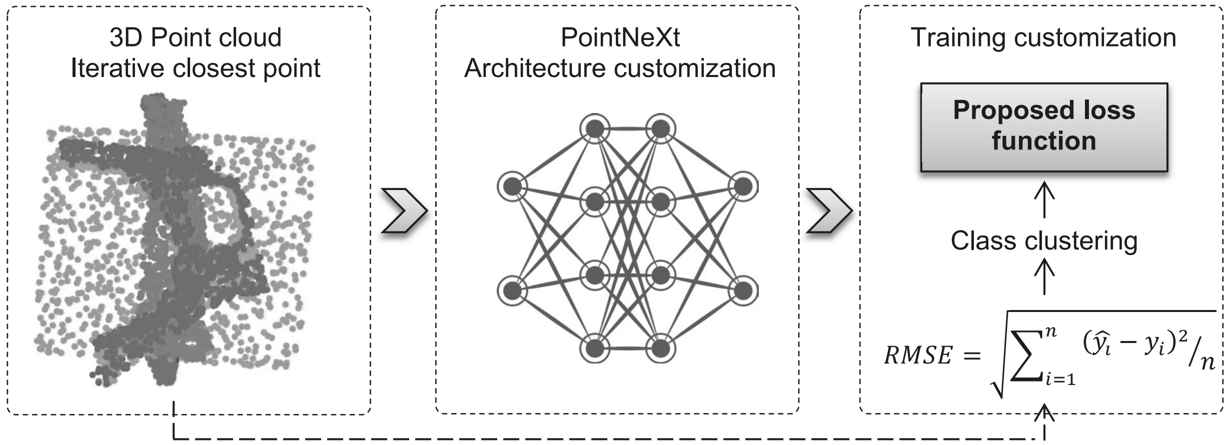


Fig.2: Machine learning workflow to find similar parts.

Findings indicate a more plausible model with a clear increase of similar parts in the top 20 predictions even for new and previously unreported automotive data (see Fig. 3), using the accuracy value ($Accuracy[\%] = \frac{\#\ Similar\ parts\ in\ top\ 20}{\#\ Similar\ parts} \times 100\%$) for comparison purposes. Such improvements within the scope of the research will strengthen the relationship between artificial intelligence and the automotive construction industry.

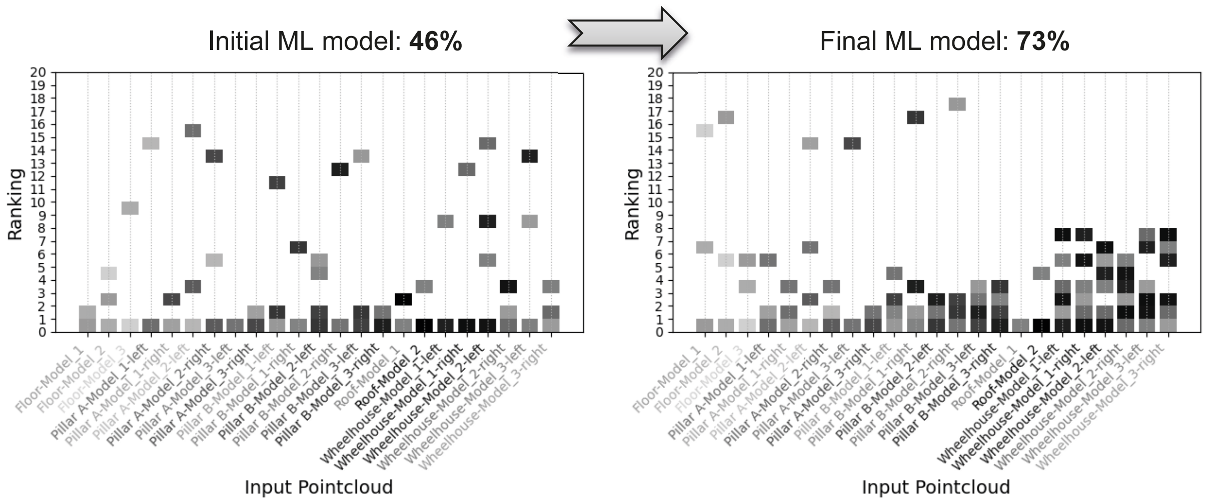


Fig.3: Initial and final state from a machine learning model within the scope of the research.

KEYWORDS: machine learning, SDM, similar parts, point cloud, iterative closest point, pointnext.

2 Literature

[1] Pillai A., Reuter U., Thiele M.: "Estimation of spot weld design parameters using deep learning". *12th European LS-DYNA Conference 2019*. No. 1, 2019, pp. 1-16.

[2] George Mason University. Center for Collision Safety and Analysis (CCSA).

[3] SCALE GmbH. 2023. SCALE.model (1.182.0). [Software]. [Accessed 01 May 2023].

[4] Zhang J., Yao Y., Deng B.: "Fast and robust iterative closest point". *IEEE Transactions on Pattern Analysis and Machine Intelligence*. No. 2, 2022, pp. 1-19.

[5] Qian G., Li Y., Peng H., Mai J., Hammoud H., Elhoseiny M., Ghanem B.: "Pointnext: Revisting pointnet++ with improved training and scaling strategies". *36th Conference on Neural Information Processing Systems (NeurIPS 2022)*. No. 2, 2022, pp. 1-18.

[6] Gare G., Galeotti J.: "Exploiting class similarity for machine learning with confidence labels and projective loss functions". *Computing and Research Repository (CoRR)*. No. 1, 2021, pp. 1-9.

Event Detection Methods for Multi-Sensor CAE-Data

Dr. Rodrigo Iza Teran[1], Dr. Daniela Steffes-lai[1], Mandar Pathare[1], Prof. Dr. Jochen Garcke[1,2]

[1]Fraunhofer Institute SCAI
[2]University of Bonn.

Abstract

Virtual product development especially in car development requires the evaluation of multiple sensors signals in the simulations as well as for comparison with the real product. Comparing many virtual sensors from many simulations turn out to be a time consuming and challenging task. We propose a methodology and workflow setting that address this challenge, allowing a similarity comparison of hundreds of sensors of hundreds of simulations detecting similar events or very different behavior as outliers. The approach uses a method of dimensionality reduction combined with hierarchical clustering. The dimensionality reduction reduces the virtual sensor data information such that a visual comparison of thousand sensor signals can be easily performed in 3D, the hierarchical clustering on the other hand allows a localized comparison of sensor signals. The approach is demonstrated using binout Ls-Dyna data from a frontal crash example with many model variants containing many sensor data per simulation.

Integration of Advanced third-party Data Analysis Approaches into SDM Systems

Leichsenring, F.[1], Borsotto, D.[2], Kracker, D.[3], Abdelhady, N.[2], Thiele, M.[1], Liebscher, M.[1]

[1]SCALE GmbH
[2]SIDACT GmbH
[3]Porsche AG

Using a simulation data management system leads to a complete record of all simulations carried out, over a defined period of time. This record set usually includes all simulation metadata (such as load case, solver version, and components) and the simulation output (raw output and derived secondary results, such as key-results). Having all this data centrally organized and stored in a uniform structure allows further data analysis across multiple simulations. These data analysis solutions can help to extract insights from the data stored in the SDM system, identify patterns and trends, and make data-driven decisions.

In SDM systems, fundamental data analysis and visualization solutions are often integrated or could be integrated relatively quickly, but moderate to complex approaches are more challenging to integrate, which is typically different from the core business of an SDM system vendor.

Some popular third-party options include business intelligence tools and services (e. g. AWS Quicksight), data visualization software (e.g., Dash), and machine learning algorithms (Femalyst, SimExplorer, AWS Sagemaker). Ultimately, the choice of third-party solutions will depend on the specific needs and goals of the business organization using the SDM system.

This presentation will demonstrate the technical approach of SCALE.result - the Add-on concept - for integrating third-party solutions into a SDM system by utilizing Femalyst from SIDACT as exemplary show case.

The tool Femalyst aims to identify anomalies in structural behaviour (outlier or so-called events) and indicate in which area of the structure the simulation result deviates substantially from the previous results. The analysis results of events are provided with a score value, allowing the user to assess the degree of deviation. Each detected event is further visualized by preview animations and 3D sub-models, which allows the user to instantly evaluate the outlying simulation.

The presented Add-on approach enables SIDACT to integrate their third-party tool by extending the SDM System with proprietary frontend and backend parts using SCALE.result and access the simulations within the platform.

In the contribution the integration of the outlier evaluation in a design workflow is highlighted at a structural model provided by the Porsche AG. Based on a rigid wall loadcase from USNCAP car safety program and a DoE of metal sheet thickness, the information gain due to the evaluation of part based outlier detection will be presented in contrast to global performance metrics, such as injury criteria or structural objective quantities.

Raising the treasure of SPDMs – How data compression and automatized event detection support engineers

D. Borsotto, V. Krishnappa, S. Müller, F. Natter, T. Roth, K. Schreiner, H. Talaat, C.-A. Thole, T. Weinert

SIDACT GmbH

Abstract

To cope up with the ever growing amount of simulation runs being performed, tools and techniques are needed to store the huge amount of simulation data and to make use of data being stored. While current Simulation Data Management systems allows managing and accessing datasets and would facilitate putting this into action for analysis, the demand on bandwidth and storage increases. Even with SPDMs, the users usually only had tools and 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. With the availability of such tools, the value of past simulation data increases. This gives rise to two challenges. The first challenge is to implement an efficient storage mechanism keeping as many simulation results as possible. This challenge can be met by data compression. It has proven in many application fields that specialized compression tools like FEMZIP for simulation results outperforms general compression tools like Winzip. The second challenge is to introduce a tool capable of analysing Terabytes of simulation result files and supporting the engineer in his task of creating crashworthiness design. Model order reduction allows to built a database meeting the second challenge. Continuously being fed with new simulation runs, a database makes it in reasonable time possible to automatically detect unknown behaviour in the most recent simulation runs compared to all predecessors at a time. To achieve this, the database does not only need to store and detect every new deformation pattern, but in addition needs to handle geometric changes and being able to detect local effects. Such databases allow 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.

Universal Data Space for Vehicle Development: Managing and Orchestrating Workflows via Policy-driven Data Transfer in Global Enterprises

Nadine Riske, Christopher Woll, Nikhil Mitaplli

GNS Systems

Abstract

While the traditional "design – build – test" development cycle often took several years, today it is essential to bring innovations to market in the near future. Data-driven product development has moved into the focus of the automotive industry to develop innovative automobiles in a more focused and shorter timeframe. However, this requires a coherent strategy that addresses the common challenges in handling engineering data – including data storage and management, optimal data access, traceability – in a future-oriented way. In particular, the broad Ansys product portfolio, with its multitude of tools in structural analysis, crash behavior analysis, and pedestrian impact analysis, demands a solid data repository. One approach is a universal data space that allows cross-system access to the growing number of data types and systems by a Digital Thread. It supports developers in using even complex data contexts for product development with Ansys tools such as ANSYS Mechanical, LS-DYNA and Ansys Fluent. Data transparency, data redundancy and data democratization as well as data management become easier, more efficient, and even more sustainable with using AI methods. This is accomplished by combining disparate systems (e.g. test benches) with the unified, consistent data platform. The platform allows automotive engineers worldwide to access data with domain-specific, intelligent semantic data models. Data from different locations, such as from a structural analysis at headquarters in Germany, from a crash analysis in Japan, and from a pedestrian impact analysis at a safety center in Sweden, can be easily managed and orchestrated via the Universal Data Space. Referring to a use case, GNS Systems outlines how an intelligent, policy-driven information flow and associated data management addresses all aspects of cost reduction, efficiency and time reduction regarding the search for relevant information. The storage, transfer and management of access rights for globally uploaded data is legally compliant at all times and meets the highest requirements in the automotive industry. Case-optimized access to global data also makes it easy to perform data integrity checks and achieve reliable results for autonomous driving simulation. Companies benefit overall from a unified ecosystem that leverages data from different sources in a sustainable way.

Investigation of Mechanical Behaviour of Lithium-ion Battery under Loading and Suggestion of Simplified Modelling Approach

Atsushi Takahashi[1], Shinichi Amano[1], Kei Saito[1], Yasuhito Aoki[2].

[1]JSOL Corporation
[2]Toray Research Center, Inc.

1 Abstract

Ensuring battery safety is one of the key issues in the design of electric vehicles. In many cases, batteries are designed to be placed in strong cases or with sufficient clearance to prevent serious damage. On the other hand, to develop a vehicle which is lighter and can run longer, it is necessary to reduce the weight of battery cases and the clearance between cells. To meet the above requirements, it is important to fully understand the mechanisms leading up to the occurrence of short circuits that cause thermal runaway, and to feed such information back into the design. Numerical analysis is one of the effective ways to evaluate battery cell deformation and damage. It requires an accurate understanding of the phenomena and the creation of an appropriate analytical model to reproduce it. However, a detailed and complex analytical model is computationally expensive, so it is desirable to build a simplified model to efficiently evaluate battery health in automobile collisions. In this study, first, to clarify the mechanism of internal short circuit generation in a real battery cell, a test was devised that held it in a deformed state while the internal cross section was observed with a digital microscope. As a result, shear layers were observed in the cross section. Considering the appearance of shear layers to be the trigger for an internal short circuit, a detailed analytical model was constructed to reproduce this phenomenon, and the amount of strain, which is considered as the criterion for internal short circuit, was predicted from the analytical results. A simplified model of the battery cell was then constructed. Here, we focused on cell deformation, including buckling, that occurs under loads parallel to the direction of layer stacking, and created an analytical model to reproduce this deformation. The electrical characteristics were modeled using a randles circuit, and internal short circuits and heat generation were evaluated.

Enhancing Battery Safety through Multi-Physics Simulation Approaches

Matthias Morak, Christian Gösweiner, Jure Vinkovic,
Harish Pothukuchi, Simon Hudales, Martin Schwab

4a engineering GmbH

Abstract

The rapid development of battery technology has led to an increasing demand for accurate and reliable simulation techniques to assess the performance and safety of battery cells with respect to their use cases. To address this need, this presentation focuses on the importance of multi-physics simulation in capturing the complex interactions between electrochemical, thermal, and mechanical phenomena within battery cells, ultimately enhancing their safety behavior. In recent years, battery safety has emerged as a critical concern due to the potential risks of electrical failure or mechanical deformation leading to electrochemical reactions inducing a thermal runaway. By integrating various physics phenomena into a cohesive modeling framework, it becomes possible to simulate and predict the behavior of battery cells under different operating conditions and external influences. This comprehensive approach enables a better understanding of the underlying mechanisms governing safety-related phenomena and facilitates the development of effective mitigation strategies. The proposed modeling approach combines electrochemical models, thermal models, and mechanical models to capture the intricate interplay between these three domains. To tackle the multi-physics simulation challenge, advanced numerical methods and simulation tools will be employed. By leveraging the capabilities of LS-Dyna, the presentation will showcase how the proposed multi-physics modeling approach can accurately capture the dynamic multi-physical behavior of battery cells. The presentation will outline the ability of multi-physics simulation to evaluate battery safety and its potential to guide the design and optimization of battery cells with enhanced safety features. Furthermore, the outcomes of the simulations can inform the development of effective thermal management strategies, material selection, and structural design to mitigate safety risks and improve the overall performance of battery systems. Overall, this presentation aims to demonstrate the significance of multi-physics simulation in understanding and predicting the safety behavior of battery cells. By combining electrochemical, thermal, and mechanical effects into a unified modeling framework, informed battery design decisions are enabled, leading to safer and more reliable energy storage solutions.

Battery Performance Evaluation using Workflows for Tests and Simulations with AI

Suri Bala[1]

[1]d3VIEW, Rochester Hills, USA

Abstract

Battery Management Systems (BMS) provides wealth of information on battery performance that is challenging for any data-management systems to process and provide meaningful insights about the battery. Among several challenges the primary ones include the wide variety of data-formats coupled with the large number of points in each of the time-history samples measures that in some cases can be in millions of points making it difficult to process in real-time.

This paper will focus on the unsupervised methods developed in d3VIEW to publish data from BMS systems and real-time computations of time-history transformations and event detections followed by long-term storage and visualization of data. Over the last few years, workflows with decision-support have replaced several manual performance evaluations and this paper will show examples of real-world evaluations with ability to learn and predict using well-known ML methods involving time-series data.

Crash Simulation of Public Transport Vehicle Traction Battery

J. Dohnal[1], M. Šebík[1], M. Popovič[1]

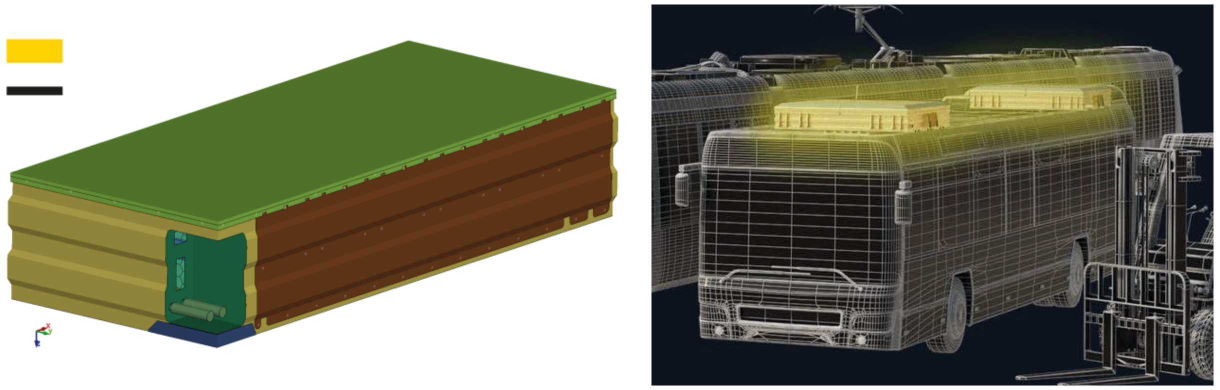
[1]SVS FEM s.r.o., Czech Republic

1 Abstract

Nowadays, lithium-ion batteries are considered as most efficient source of power for electric vehicles (EVs). With the increasing utilization of EVs, the requirements for higher performance, lower weight and improved safety are also growing. These demands can be fulfilled by an improved traction battery design, which consists of decreased battery frame mass or higher number of battery cells. However, with these improvements come several negative aspects, such as higher risk of battery frame intrusion or reduction of space between the cells. Due to these factors, the risk of battery damage is rising and it is crucial to predict and better understand the behaviour of the battery cells during critical situations, such as vehicle crash.

Ansys LS-DYNA is a useful tool for the evaluation of battery cells during abusive scenarios. It offers the creation of multi-physics model that could predict coupled mechanical, electrical and thermal responses. Electro-chemical behaviour of a cell is described by equivalent (Randels) circuit, which is incorporated in homogenized BATMAC model used in the paper. This simplified model is sufficient for the assessment of battery cell response to short circuit, which can be triggered by mechanical damage (penetration, deformation), overheating or high current flow. Depending on the state of charge (SOC), a short circuit can lead to uncontrollable, self-heating state called thermal runaway. This state poses the greatest safety risk for Li-ion batteries. To calibrate and optimize the numerical model, various mechanical, thermal and electrical tests need to be conducted.

The paper focuses on crash simulation of public transport vehicle traction battery with prismatic cells. This work is part of a comprehensive project dedicated to the development of a resilient early warning battery failure system. This system shall enable the fire rescue service to promptly differentiate between non-hazardous technical faults and critical accidents, ensuring timely response. Another crucial part of the project is the design of a self-extinguishing and cooling system for the battery pack. These designs aim to improve the safety and extend the lifetime of the new traction battery. Project is in progress in cooperation with CTU in Prague, NANO POWER a.s. and TÜV SÜD Czech s.r.o.



a) b)

Fig.1: a) FE Model of battery pack, b) public transport bus traction battery placement.

Thermal Runaway in Electric Vehicle Crash Simulation using LS-DYNA

Pierre L'Eplattenier, Inaki Caldichoury, Kevin Kong, Vidyu Challa, Dilip Bhalsod, Srikanth Adya, Mike Howard

Ansys Inc.

Abstract

Safety is an important functional requirement in the development of large-format, energy-dense, lithium-ion (Li-ion) batteries used in electrified vehicles. Many automakers have dealt with this issue by enclosing the batteries into robust protective cases to prevent any penetration and deformation during car crashes. While this worked well for first generation vehicles, consumers are increasingly interested in higher range, which makes overengineered heavy protective cases detrimental for range. A more detailed understanding of battery behavior under abuse is therefore necessary to properly make design trade-offs. Computer aided engineering (CAE) tools that predict the response of a Li-ion battery pack to various abusive conditions can support analysis during the design phase and reduce the need for physical testing. In particular, simulations of the multi-physics response of external or internal short circuits can lead to optimized system designs for automotive crash scenarios. The physics under such simulations is quite complex, through coupling structural, thermal, electrical and electrochemical. Moreover, it spans length scales with orders of magnitude differences between critical events such as internal shorts happening at the millimeter level, triggering catastrophic events like the thermal runaway of the full battery. The time scales also are quite different between the car crash happening in milliseconds and the discharge of the battery and temperature surge taking minutes to hours. A distributed Randles circuit model in Ansys LS-DYNA® can mimic the complex electrochemistry happening in the electrodes and separator of lithium-ion batteries. The Randles circuit model is coupled with the mechanical solver of LS-DYNA where the deformations due to a battery crush allow the definition of criteria to initiate internal shorts. The internal shorts produce Joule heating which are transferred to the thermal solver of LS-DYNA where the temperature rise can trigger exothermal reactions leading to thermal runaway. In order to correlate mechanical deformations to the onset of internal shorting, subsequent appearance of thermal runaway, resulting gas emission, fire, and/or explosion, a series of crush experiments on 26 Ah pouch cells and modules have been performed by Ansys. Benchmarks between experimental and numerical results from LS-DYNA allowed the tune up of cell dependent parameters. The cells were then used in different simulations of module level and full electric vehicle level crashes. This paper will present the workflow from single cell experiments to full crash modeling including the sequence of steps, which can be repeated for other electric or hybrid vehicles with different battery cells.

Realistic articulation, positioning and simulation of Human Body Models using Oasys LS-DYNA tools

Manu Agarwal, Galal Mohamed

Arup

Abstract

The development of biofidelic human body models (HBMs) is essential in vehicle and transport safety development to predict human body kinematics and injury risks and responses to a variety of impact scenarios. However, realistic articulation and positioning of complex HBMs for simulation-based analysis remains a challenging and time-consuming task. This paper describes recent efforts and updates to support and improve the positioning of HBMs ready for simulation-based analysis using LS-DYNA and Oasys PRIMER, a powerful LS-DYNA pre-processor.

Oasys PRIMER positioning tree files have been created for the most popular industry leading HBMs, adding realistic biofidelic joint detailing to the original models, including bending and straightening of the spine, and realistic shoulder/clavicle movement. These positioning files have been created based on detailed anatomical research and include beam entities and provide stop angles to various joints to ensures realistic relative movement between various parts of the model. When these files are loaded into Oasys PRIMER they enable easy and interactive positioning of these complex HBMs.

The paper also describes various interactive and automation tools that are available inside the Oasys LS-DYNA suite of pre and post software. Using these tools, users can simply select the desired joint and move it to the desired position. An intuitive graphical user interface and visualisation tool has also been developed to aid the user to customise the visual attributes of the HBM entities and facilitate a smoother positioning process. Finally, Oasys PRIMER supports a powerful scripting interface that can be used to automate the HBM positioning process and help create generic workflows for different positioning and crash load cases.

Assessment of Abdominal and Skeletal loadings and Kinematics during Frontal Impacts through a Novel Tool for HBM Variants Generation Based on the Occupant’s BMI

Zouzias D., Fokylidis A., Lioras A., Rorris L

BETA CAE Systems SA

1 Abstract

HBM variant generation tools are solely based on morphing techniques to adjust the shape of the external surface to a target depending on the BMI of interest. Even though this approach can quickly produce HBM variants, downgrades the model’s mesh by stretching the elements and compromising their quality. Furthermore, the dependence of the abdominal organs morphology on the occupant’s BMI is rarely, taken into account. This paper presents a novel tool for automatic HBM variant generation, that respects elements’ quality taking also into account the volume of the abdominal organs.

2 Methodology

A parametric study of a frontal sled test was conducted in LS-DYNA v12, using GHBMC M50 v6 Occupant as reference. The reference loadcase was compared with the loads applied on M95 and a newly generated model of an obese occupant (BMI 44) through BETA CAE’s HBM Variant Generation Tool developed within the environment of ANSA. M50 was modified to the morphology of an obese occupant by adapting the HBM’s external surface and the abdominal organs’ volume per occupant’s BMI. For the external dimensions of the HBM, a human model scan of high BMI was utilized as a target surface to which the reference skin was fit. The newly created volume of subcutaneous fat was automatically filled with hexa and tetra elements. The volume of the stomach, liver, pancreas, kidneys and gallbladder was morphed to fit the volume prescribed per BMI by clinical research. Frontal crash simulations were subsequently conducted to compare the kinematics and loads between variants.

3 Results

A new variant of an obese occupant was successfully generated within 25 mins. 97.8% and 95.5% of solid and shell elements respectively created for the external fat tissue, had a jacobian value above 0.9. Results were aligned with previous studies and accident statistics on the kinematics and load localisation providing additional level of detail. Compared to M50, obese model suffered more severe impacts of the lower extremities onto the dashboard as a result of the limited space for the occupant in the cabin. The loading profile of the skeleton differs with stresses also localised on the spine. The loading of the upper abdomen was reduced exhibiting a delay in building-up. M95’s profile combined the aforementioned with increased time of load building-up but similar severity of impact of the lower extremities onto the dashboard.

4 Conclusions

A new tool for the rapid generation of HBM variants is presented and utilized to compare the loading on the abdomen area and skeleton of three different HBM variants. The study was able to verify the different loading conditions observed on occupants of different BMIs on the respective areas. The loading of the upper abdomen was reduced and exhibited more time to build-up on the obese and M95 occupant model compared to the average. The lower extremities of the obese and M95 occupant seem to have higher probability of injury with shorter time of impact onto the dashboard. Different loading profiles were observed also on the spinal cord which aligns with accident statistics.

Comparative Study of Positioning HBM to Cycling Postures Based on Experimental Data

Maria Oikonomou[1], Athanasios Lioras[2], Lambros Rorris[3], Thomas Nikodelis[1], Athanasios Mihailidis[1]

[1]Aristotle University of Thessaloniki
[2]BETA CAE Systems SA
[3]BETA CAE Systems International AG

1 Abstract

The objective of Crash Analysis is to ensure the safety of a diverse spectrum of road users through the utilization of several finite element (FE) crash simulations. While a considerable number of studies have focused on evaluating the motion and the potential injuries of occupants and pedestrians, the investigation of two-wheel vehicle users remains underrepresented, even though they are considered vulnerable road users and two-wheelers are common personal transportation. Therefore, studying their kinematic behavior in multiple collisions is crucial to ensure safe and convenient travel.
A limited number of studies dedicated to two-wheelers are conducted using Human Body Models (HBMs), even though they are deemed the most proper way to explore human behavior in various crash scenarios. HBMs are generated in the seating and standing postures, representing the occupant and pedestrian accordingly, and consequently, the articulation of the HBMs to adopt the distinct cyclist posture is challenging. Thus, the comprehensive evaluation of cyclists' safety using HBMs entails the resolution of several difficulties.
This study undertakes the exploration of distinct ways of identifying cyclists' postures by employing different data collection processes. A comparison between the data collection methods is executed to specify the most rational approach. Furthermore, this study develops an articulation method that positions the HBM in a target posture. Thus, the HBM is positioned in the cyclist's target posture using data computed by the chosen data collection method. The acquired knowledge will facilitate the investigation of unique non-standard initial positions and contribute to the extensive examination of cyclists' behavior in multiple crash situations using HBMs.
The study was conducted in LS-DYNA, using the THUMS M50 HBM. The positioning process IS developed in ANSA Python scripts and using the ANSA HBM Articulation tool.

Dummies for Future Testing Scenarios

Nenad Dimitric

Humanetcs Humanetics Europe GmbH

1 Abstract

Virtual testing (VT) using finite element (FE) models is a rapidly evolving that is revolutionizing the way vehicles are designed and assessed. It has been introduced in many regions world-wide such as the China Automotive Technology & Research Center (CATARC) and Euro NCAP to develop safer vehicles. It allows for the simulation of a wide range of crash scenarios, including frontal, side, far side and rear impacts, as well as active safety technologies to reproduce real life vehicle crashes. This can help to identify and mitigate safety risks early in the design process, which can save human lives and reduce the societal and vehicle development costs.
This paper reviews advancements in anthropometric test devices (ATD) FE model technology for VT in terms of new products and increased accuracy in injury predictions. The THOR Precrash dummy is an advanced ATD used for simulating frontal impacts during the pre-crash maneuvers such as automatic emergency breaking (AEB). The THOR-AV is another advanced ATD for frontal impact used for simulating the reclined occupant posture targeting the Autonomous Vehicles design. The THOR-5th and EVA-RID are advanced dummies to represent women population in various crash events. In addition to new products, advancement in sensing and measuring technologies, for example the pressure vest deployed in the HIII-5th (Hybrid III 5th) chest to measure localized chest compression for injury analysis. Such technologies help product design teams to understand the injury risk and account for needed countermeasures early in design cycle to make vehicles safer. There has been advancement in the side and far side impact scenarios using the side impact ATDs such as World Side Impact Dummy (World SID).

This is Hans - The new Human Body model of DYNAmore

A. Gromer, D. Freßmann

DYNAmore GmbH, an ANSYS company

Simulation of back-injection molded parts using MAT_058 and MAT_215

Kai-Chien Chuang, Sanjay Kumar Sardiwal, Boris Cordero Porras, Patrick Scholz, Olaf Hartmann

ARRK Engineering GmbH

1 Abstract

Within the modern automotive industry, long-fiber-reinforced polymers (LFRPs) have gained increasing popularity because of efficient production of complex geometries in combination with relatively high stiffness and strength. Increased mechanical performance can be achieved by combining LFRP with continuous fiber composites, such as UD-Tape while using back-injection molding. The combination of these two material types poses a challenge in CAE, because of their individual anisotropic behavior.

The aim of this paper is to present a simulation approach for modeling the combination of LFRP with continuous fiber-reinforced thermoplastics (CFRTP) using the micro-mechanical-based material model MAT_215 for PP-LGF30 and MAT_058 for CF-PP-UD-Tapes in LS-DYNA®. For material characterization purposes, coupon tests (e.g. tensile) are performed under different angles in order to determine the material properties of PP-LGF30. CT-Scans are performed on samples extracted from specific positions of the plate to identify the orientation tensors. For the CF-PP-UD-Tapes, additional coupon tests were performed in order to determine the longitudinal, transverse and shear properties.

Based on the coupon test results, the material properties were characterized for PP-LGF30 using MAT_215 taking the local anisotropy into consideration. For the CF-PP-UD-Tapes, the MAT_058 material model was calibrated with the experimental data. The numerical results in LS-DYNA® obtained for the coupon samples showed a good agreement with the experimental test results for both the PP-LGF30 and the CF-PP-UD-Tapes material. The anisotropy was also well captured for the LFRP at coupon level for Tension 0°, 45°, and 90°.

Furthermore, the calibrated material cards were implemented on a component level in order to validate their capability to capture the overall structural response of the simulation in comparison to the experimental results. Fiber mapping was implemented with fiber orientation data taken from a molding simulation. The numerical results from the component test showed a good agreement with respect to experimental data.

Modelling Delamination in Fibre-Reinforced Composites subjected to Through-Thickness Compression by an adapted Cohesive Law

Moritz Kuhtz[1], Johannes Gerritzen[1], Jens Wiegand[2], Andreas Hornig[1], Maik Gude[1]

[1] TUD Dresden University of Technology, Institute of Lightweight Engineering and Polymer Technology, Holbeinstraße 3, 01307 Dresden
[2]COMPACT Composite Impact Engineering, 12 Gateway Mews, London N11 2UT

1 Abstract

The LS-DYNA material model *`MAT_COHESIVE_MIXED_MODE` (*MAT138) is used to predict the delamination behaviour under a superposition of out-of-plane tensile σ_3 and shear stresses $\tau_{II} = \sqrt{(\tau_{13})^2 + (\tau_{23})^2}$. Kellner [1] provides code for an LS-DYNA user material model which, like *MAT138, is based on the work of Dávila and Camanho [2].
The influence of a compressive stress in the thickness direction σ_3 is not taken into account in both material models. However, some works show an increase of the interlaminar properties like shear strength R_{II} and the mode II critical energy release rate $\mathcal{G}_{IIc}$ with increasing compressive stress σ_3 [3]. While Catalanotti et al. use the single parameter η to model the impact of a compressive stress on both strength and critical interlaminar energy release rate, here, two independent parameters η_R and $\eta_{\mathcal{G}}$ are chosen in order to take the different phenomena causing delamination initiation and growth into account:

$$R_{II} = R_{II0} \sqrt{(1-\eta_R\ \sigma_3)},\ \text{for}\ \sigma_3 < 0,$$
$$\mathcal{G}_{IIc} = \mathcal{G}_{IIc0}\left(1-\eta_{\mathcal{G}}\ \sigma_3\right),\ \text{for}\ \sigma_3 < 0,$$

where R_{II0} and $\mathcal{G}_{IIc0}$ are the interlaminar strength and critical energy release rates without superimposed compressive stress. This approach is used to develop a new user material model that factors the increase in interlaminar properties with increasing interlaminar compressive stress in.
To verify the user material model, various σ_3-τ_{13}-stress states are generated in a single-element test and the failure behaviour is analysed. The results are compared to both the *MAT138 and the model provided in [1]. The user material model presented here is additionally used to simulate a three-point bending test, where the delamination failure is overestimated when using *MAT138 [4]. By comparing the simulation results with the experimental investigations, the benefit of the new user material model is shown.

2 Literature

[1] Kellner, L: " How To - user defined material models with LS-Dyna on Windows", https://www.researchgate.net/publication/327623424_How_To_-_user_defined_material_models_with_LS-Dyna_on_Windows, available on 30th May 2023.
[2] C. Dávila and P. Camanho, “Decohesion Elements using Two and Three-Parameter Mixed Mode Criteria,” in American Helicopter Society Conference, Williamsburg, VA, 2001. https://ntrs.nasa.gov/api/citations/20020010916/downloads/20020010916.pdf, available on 30th May 2023
[3] Catalanotti, G.; Furtado, C.; Scalici, T.; Pitarresi, G.; van der Meer, F.P.; Camanho, P.P.: The effect of through-thickness compressive stress on mode II interlaminar fracture toughness, Composite StructuresVolume 182 (2017), 153-163. DOI: 10.1016/j.compstruct.2017.09.014
[4] Kuhtz, M.; Richter, J.; Wiegand, J.; Langkamp, A.; Hornig, A.; Gude, M.: Concepts for Increased Energy Dissipation in CFRP Composites Subjected to Impact Loading Conditions by Optimising Interlaminar Properties. Aerospace 2023, 10, 248. DOI: doi.org/10.3390/aerospace10030248

Automatic Generation of High Performance Material Models for Long Fiber Reinforced Plastics in Crash Simulations

Timo Schweiger, Jörg Lienhard, Hannes Grimm-Strele

Fraunhofer Institute for Mechanics of Materials IWM

Abstract

Long fiber reinforced plastics (LFRPs) offer excellent mechanical properties and are widely used in automotive and aerospace industries. Accurately modeling the behavior of LFRPs under crash conditions is crucial for designing lightweight and safe structures. However, creating reliable material models for LFRPs is challenging due to their complex microstructure and anisotropic nature. This study presents an automatic method to generate highly accurate material models for LFRPs, specifically tailored for crash simulations. The proposed approach employs virtual testing on representative volume elements (RVEs) that model the LFRP material with fibers and matrix separately. To ensure accuracy, a micro material model is applied, which is calibrated using real micro material experiments combined with computed tomography (CT) scans to obtain the actual fiber orientation distribution. An anisotropic material card is then calibrated using the generated data. The influence of various factors, including strain rate, fiber orientation, fiber concentration, and stress state, is accurately described within the material model. The model's predictive capabilities are validated against a range of experimental tests, including tension, shear, and biaxial loading conditions. The main advantage of this method is its efficiency and reduced experimental effort compared to established techniques. By leveraging virtual testing on RVEs and incorporating real micro material data, the proposed approach significantly reduces the time and resources required for material characterization. This enables quicker development and optimization of LFRP structures for crash applications, leading to improved safety and reduced time-to-market. The presented automatic method for generating highly accurate material models for LFRPs offers a valuable tool for engineers and researchers involved in crash simulation and design optimization. Its ability to capture the intricate behavior of LFRPs under various loading conditions paves the way for enhanced structural analysis and lightweight design in numerous industries.

Investigation on failure modeling for short glass fiber reinforced thermoplastics under crash loading

Martin Meyer[1], Hugo Hennion[1], Christian Liebold[2], Stefan Hartmann[2]

[1]Faurecia Autositze GmbH
[2]DYNAmore, an Ansys Company

1 Abstract

In this presentation, we will introduce the current methodology to calibrate a **`*MAT_ANISOTROPIC_ELASTIC_PLASTIC (*MAT_157)`** material card for crash modeling of short fiber reinforced plastic (SFRP) components. The current implementation of stress-based failure criteria (Tsai-Hill/Tsai-Wu) will be discussed together with a possible approach utilizing **`*MAT_ADD_GENERALIZED_DAMAGE.`** Based on generic examples and real components, advantages and disadvantages of the currently available methodology will be discussed.
Afterwards, new strain-based failure criteria recently implemented in LS-DYNA for **`*MAT_157`** based on the Tsai-Hill/Tsai-Wu approach are investigated, showing improved predictability compared to the stress-based criteria. Besides the discussion of the various failure modeling approaches, an insight into proper material calibration methods in both, tension and compression will be given.
The authors will conclude with a discussion of the limitations of the shell theory when modeling seat structures for automotive applications made from plastic components. Special consideration will be given to the available mapping procedures from molding to structural simulation models and how they affect the result finite element analysis.

Multiphysics simulation of electromagnetically-controlled shape morphing composite

Piotr Bartkowski, Łukasz Pawliszak

Warsaw Univeristy of Technology.

Abstract

The multiphysics simulation of electromagnetically-controlled shape morphing composite (e-morph) is presented in this work. The structure is composed of a granular core and an outer shell made of silicon, with channels filled with liquid metal (EGaln). Composite can change its shape in a controlled way by Lorentz forces, while the jamming mechanism allows the granules used in the core to maintain the desired shape. Due to its properties, the composite can find many applications in the field of soft robotics. This work provides a detailed description of the composite and about magneto-thermo-mechanical modeling methodology. The simulation work was compared with experimental research. At the end the potential applications of the composite are presented.

Keywords: EM actuator, Granular jamming, Liquid metal, Multiphysics simulation, Soft robotics

Importance of Plasticity for GISSMO Calibration in Automotive Safety Applications

Richard Burrows

NIO Performance Engineering

Abstract

Industrial application of the GISSMO model available in LS-Dyna has evolved over more than a decade to create a flexible tool used to describe fracture for many industrial problems. Extensive research and publication thereof has been the result on this effort, and resulting from the flexibility which GISSMO offers are many approaches for calibration and application. Data generation and the methodology used in CAE are important for a useful model in this plethora of industrial problems. A treatment is given here looking into the influence of post localisation work hardening and the effect this has on firstly the data generation side of the engineering problem, through the calibration process and final application. Experimental data is discussed and how the processing can be performed to result in an Automotive Safety card working for larger shell elements in the 3-6mm range. Direct Image Correlation results discussed herein are Shear, Uniaxial Tension, Plane Strain Bending, Membrane Plain Strain in three directions, as well as Balanced Biaxial test results. Discussion is given to incorporation of high strain rate tension tests performed up to 500/s and how this can be incorporated to be of industrial use. Hydraulic Bulge Test results are also discussed and their relevance to flow curve extrapolations and the limitations of relying solely upon one stress state for Flow Curve extrapolation. Industrially relevant Dual Phase and Press Hardenable Steels are discussed including Top Hat experiments easily applied for a validation step before the card can be deemed verified and validated. A discussion is given surrounding the additional steps required for accurate CAE of complex formed parts used in an automotive body structure.

Advanced Plasticity & Fracture for Structural Car Body Metals in Crashworthiness CAE analysis: SAMP-1 plus GISSMO

Alejandro Domínguez[a], Pablo Cruz[a], Lluis Martorell[a], Adrián Ros[a], Eduardo Martin-Santos[a]

Courtesy of: Andrew Hall[b], Patrick Kelly[b], Ilyasuddin Syed[b]

[a]Applus IDIADA

[b]Faraday Future

Abstract

This paper presents an engineering process for generating material cards that accurately capture the plastic and fracture behavior of car body structural metals in forefront crashworthiness CAE analysis. The aim is to demonstrate that advanced plasticity approaches can be employed without significantly increasing the complexity of material characterization. The paper challenges common assumptions in the automotive CAE community regarding metals plasticity and highlights the limitations of using the well-known MAT 24. It emphasizes the need for *Multi-Loading-mode-Yielding* plasticity models and proposes the use of SAMP-1 (MAT 187) material card for improved accuracy. SAMP-1 is calibrated with the same coupon test matrix as the one used for the calibration of the GISSMO fracture model with no need of enlarging the coupon test matrix for the Multi-Loading-mode-Yielding plasticity calibration.

To establish SAMP-1 as a standard for car body metal structural parts in crashworthiness CAE analysis, a user-oriented approach is proposed. The approach is validated for shell-based CAE models and it is illustrated using an aluminum extrusion alloy. The paper describes a comprehensive coupon and component experimental campaigns for plastic and fracture characterization courtesy of Faraday Future. The results highlight the importance of assessing the material yielding dependence on different loading modes and evaluates the suitability of MAT 24 and SAMP-1.

The paper proposes the use of the Strength Differential concept as a simplified approach for defining *Multi-Loading-mode-Yielding* plastic behavior using SAMP-1.

In summary, this paper offers a user-oriented approach to enhance the accuracy of material characterization in crashworthiness CAE analysis for car body structural metals. It underlines the benefits of considering *Multi-Loading-mode-Yielding* behavior and recommends the use of SAMP-1 plus GISSMO. The proposed methodology is experimentally validated at coupon and component level and it is applicable to shell-based crash CAE models.

Simulation of ductile fracture at high strain rate and temperature in shear sensitive materials using the plasticity-damage self-consistent model (PDSC)

Gabriel Testa, Gianluca Iannitti, Andrew Ruggiero, Sara Ricci, Nicola Bonora

University of Cassino and Southern Lazio, Cassino I-03043, Italy

Abstract

In materials that exhibit a shear effect on fracture strain, stress triaxiality alone is insufficient to describe stress state effect on ductility. For these materials, it has been proposed that ductility also depends on the third invariant of the stress deviator. Recently, Bonora and Testa [N. Bonora, G. Testa, Plasticity damage self consistent model incorporating stress triaxiality and shear-controlled fracture mechanisms – model formulation, Eng. Frac. Mech. 271, 108634 (2022).] derived a plasticity damage self-consistent (PDSC) model considering damage contributions due to different micromechanisms such as intervoid necking, shearing, and sheeting for an arbitrary stress state. This model formulation has proved to predict accurately ductile fracture under an extensive range of stress triaxiality and Lode angle parameter and especially in compression. The PDSC has been extended to account for strain rate and temperature effects. In this work, the PDSC has been implemented in LS-DYNA and used to predict ductile fracture in Taylor cylinder and rod-on-rod (ROR) impact tests on Al 2024-T351 performed at 290 m/s and elevated temperature (80°C, 155°C and 240 °C). The ability of the model to anticipate the fracture mode and the transition from shear-controlled fracture to void nucleation and growth type of fracture is shown.

Failure model calibration of a DP1000 dual phase steel using solid and shell elements for crash simulation

Florence Andrieux[1], Silke Klitschke[1], Andreas Trondl[1]

[1]Fraunhofer-Institut für Werkstoffmechanik IWM

Abstract

In the trend of lightweight construction more and more advanced high strength steel sheets (AHSS) come to application in automotive structural components. Since the ductility of high strength steels is relatively low, damage behavior of these materials must be modeled. In automotive structures AHSS components are usually discretized with shell elements. With the increase of computation capacity attempts with solid elements are made to capture the loading state more accurately, especially after necking. For this reason, it is convenient to develop a method which enable a systematic model calibration from the 3D to the 2D loading situation. The failure strain of metallic materials depends on stress state. In the past years several studies have shown that the stress triaxiality is not sufficient to describe failure and empirical models were extended to consider the effect of Lode parameter. Recently it was also show that the amount of bending seems to influence failure, namely the failure strain increases with the amount of bending. In this work a DP1000 dual phase steel was investigated both experimentally and numerically. To study the dependence of damage behavior on loading state, smooth, notched and shear tensile tests and tension-bending, punch and Nakajima tests were performed. A general damage model based on a critical failure strain depending on triaxiality and Lode parameter was calibrated for solid elements. The failure stain curve for shell elements was derived from the failure surface for solid elements considering the relationship between triaxiality and Lode parameter under plane stress condition. Moreover, the influence of the bending factor was incorporated into the failure model for shell elements. FE simulations with LS-DYNA were performed to determine the local values of triaxiality, Lode parameter and bending ratio for each specimen type. The applied damage models were calibrated and verified by simulating all specimen tests. It is shown that the developed method well captures the failure behavior for solid and shell elements. The predictions were better using shell elements because of the possibility to account for the bending ratio influence, what is not possible using solid elements.

Stress state dependent regularization

F. Andrade[1], M. Feucht[2] , T. Erhart[1] (DYNAmore)

[1]DYNAmore, an Ansys Company
[2]Mercedes-Benz AG

Simulating Structural Resistance of D&I Food Cans to Open Up Downgauging Potential

B. Liebscher[1], F. Knieps[1], I. Weinand[1]

[1]thyssenkrupp Rasselstein GmbH, Koblenzer Straße 141, Andernach, Germany

To reduce cost and increase the efficiency of D&I food cans, a lighter can with the same axial stability and paneling resistance is required. Axial stability depends on wall thickness, bead geometry (mainly bead depth) and tensile strength in the wall, whereas paneling resistance is a function of wall thickness, Young's modulus and bead geometry (mainly bead depth), with the bead depth having an opposite influence on paneling resistance and axial stability. FEA is used to find a bead geometry that satisfies both the paneling resistance and axial stability requirements. For a stable calculation of the paneling resistance, perturbation in the form of an eigenmode is required. The calculation time is significantly reduced by using an implicit solver with arc length method. When simulating axial stability, accurate modeling of the beginning of the flow curve is required. A weight reduction of 5% can be achieved by using next-generation high-strength D&I steel grades (e.g. rasselstein® D&I Solid).

1 Introduction

In the packaging steel market, demand for CO_2 and cost reduction goes along with downgauging ambitions to receive lighter cans. At the same time, there is a trend away from three-piece cans, which consist of a welded body, seamed bottom and lid, towards two-piece cans (also named D&I food can). D&I food cans are manufactured by draw and redraw of a cup and a subsequent ironing. This process has the advantage to avoid a welded body, which is more vulnerable regarding leakages. At the same time, a two- piece can line shows a higher profitability when operating at maximum capacity. Both, three- and two- piece cans must meet certain stability standards. In particular, a can must withstand a certain axial load and at the same time a certain paneling pressure to ensure a stable sterilization or pasteurization process. When aiming for a thickness reduction of the sheet metal, it must be ensured, that the defined stability requirements are kept. Thus, there is a trend towards higher strength materials. Materials as "rasselstein® D&I solid" offer a higher yield strength and a stronger hardening behavior, which results in a higher axial load resistance. However, the paneling pressure is not a function of the material strength but rather dependent on the can stiffness. This means, that sheet thickness, geometry (bead depth) and Young's modulus influence the resulting paneling pressure. Therefore, thickness reduction by using new generations of higher strength D&I steels always require the adaption of the can geometry. A higher bead depth leads to decreasing axial load while the paneling pressure increases. At this point, finite element analysis is of interest, to reduce cost intensive trial and error experiments. The general benefit of FE-simulation in the packaging steel industry was described by Köhl et al. fundamentally [1]. Both, simulation of axial load resistance and paneling pressure have not been investigated for D&I food cans so far. In a publication from Predictive Engineering Inc., the geometric nonlinear behavior of cylinders [2] was already shown. In addition, the need to pertub the geometry by using eigenvalue buckling mode shade was also demonstrated. Whang et al. simulated already axial load and paneling resistance for beaded cans and showed the optimization potential. They also noticed the need to consider eigenvalues for the simulation of paneling resistance [3]. In addition, a good overview on simulating structural resistance of metal container was given. However, only three-piece cans were considered in their investigation. Especially, to enable the use of FE-simulations in case of axial load resistance for D&I food cans the precise description of the yield strength in the can wall is crucial. Concerning the characterization of packaging steel for finite element analysis, Knieps et al. pointed out different characterization approaches to enable a suitable strategy for packaging steel [4,5]. However, the investigations only focused on sheet metal behavior and not on mechanical properties in the can wall. To enable a prediction of the mechanical properties in the call wall, either the whole process including the ironing must be simulated, or the mechanical properties must be measured directly from the can wall. FE-simulations of the ironing process were shown in general by Dankert et al. [6] and in detail for tinplate products by Nam and Han [7]. However, the simulation of the ironing process leads to high computational cost and is not able to consider the aging behavior afterwards which results from lacquering. Thus, aim of this work is to enable a precise simulation of axial load resistance and paneling pressure for D&I food cans. The results will be used to point out the downgauging potential by using D&I Solid material with an optimized bead geometry.

2 Literature

[1] M. Köhl, F. Knieps, I. Weinand, B. Liebscher, Effiziente Produktentwicklung in der Verpackungsstahlindustrie durch Nutzen von FE-Methoden, in: 33. Aachen Stahlkolloquium 2022.
[2] Linear and Nonlinear Buckling Analysis and Flange Crippling, Engineering Mechanics White Paper 2012.
[3] J. Wang, Design optimization of rigid metal containers, Finite Elements in Analysis and Design 37 (2001) 273–286. https://doi.org/10.1016/S0168-874X(00)00043-3.
[4] F. Knieps, M. Köhl, M. Merklein, Local Strain Measurement in Tensile Test for an Optimized Characterization of Packaging Steel for Finite Element Analysis, KEM 883 (2021) 309–316. https://doi.org/10.4028/www.scientific.net/KEM.883.309.
[5] F. Knieps, B. Liebscher, I. Moldovan, M. Köhl, J. Lohmar, Characterization of High-Strength Packaging Steels: Obtaining Material Data for Precise Finite Element Process Modelling, Metals 10 (2020) 1683. https://doi.org/10.3390/met10121683.
[6] J. Danckert, The Residual Stress Distribution in the Wall of a Deep-Drawn and Ironed Cup Determined Experimentally and by FEM, CIRP Annals 43 (1994) 249–252. https://doi.org/10.1016/S0007-8506(07)62206-9.
[7] Jaebok Nam, Kyung Seop Han, Finite Element Analysis of Deep Drawing and Ironing Process in the Steel D & I, in: Materials Science.

Numerical study on a new forming method for manufacturing large metallic bipolar half plates

David Briesenick[1], Maxim Beck[1], Kim Rouven Riedmueller[1], Mathias Liewald[1]

[1]Institute for Metal Forming Technology, University of Stuttgart

Abstract

Fuel cell technology offers a sustainable power supply solution for heavy-duty vehicles, aviation and shipping as well as stationary application. Manufacturing of metallic bipolar plates (MBPP) as a key component of fuel cells is nowadays one of the main topics in production-based research and processing industry. One reason for this is that although stamping of thin stainless-steel foils enables an economic large-scale production of metallic bipolar half plates, tooling and press technologies required for embossing and shear cut operations are highly demanding and thus continuously being developed. Particular challenges are posed by the embossing of the complex flow field structure, which can cause forming defects and pronounced springback phenomena. Moreover, multiple embossing operations and high press forces are needed to plastify the metal foils as homogeneously as possible and thus form the desired channel geometry. Accordingly, future demand for large metallic bipolar plates is likely to exceed the limits of current press technologies in terms of force capacity and stiffness. Similarly, rolling is also not suitable for forming such large plates due to severe wrinkling and springback. Therefore, this paper proposes the so-called TIP-Form as a new forming process with a tilting forming kinematic of one or both tool halves as well as a crowning of the active surface. In this regard, a numerical study is presented, in which the new forming approach to decrease the force demand and improve the formability when embossing bipolar plates was investigated. Explicit forming simulations of a flow field section were performed using LS-DYNA and forming forces as well as formability were evaluated. Compared to conventional embossing, the controlled tipping and crowning of the punch could reduce the simultaneous acting contact zone between tool and foil and therefore maximum forming forces have been reduced. In addition, thinning of formed channels decreased, as the tipping forming kinematics enabled a beneficial material flow. The promising numerical results motivate for further research work and emphasis the development of a new press technology with tipping ram or table for the manufacturing of large metallic bipolar plates.

Higher-Order 3D-Shell Elements and Anisotropic 3D Yield Functions for Improved Sheet Metal Forming Simulations: Part I

Maximilian Schilling[1], Tobias Willmann[1], Alexander Wessel[2,3], Alexander Butz[2], Manfred Bischoff[1]

[1]University of Stuttgart, Institute for Structural Mechanics,
Pfaffenwaldring 7, 70550 Stuttgart, Germany
[2]Fraunhofer Institute for Mechanics of Materials IWM,
Woehlerstrasse 11, 79108 Freiburg, Germany
[3]Technical University of Munich, Chair of Metal Forming and Casting,
Walther-Meissner-Strasse 4, 85748 Garching, Germany

1 Abstract

Sheet metal forming simulations are crucial in various industries, such as automotive, aerospace, and construction. These simulations are commonly carried out using Reissner-Mindlin shell elements, which involve certain simplifying assumptions about zero normal stress in shell normal direction and cross-sectional fibers remaining straight during deformation [1]. Because of this, the material model needs to be modified and no three-dimensional material model can be used. However, in critical forming situations such as bending with small radii relative to the sheet thickness, these assumptions do not hold, resulting in inaccurate simulation results. To address this issue, a higher-order 3D-shell element that incorporates a full three-dimensional constitutive model and that can account for cross-sectional warping and higher-order strain distributions has been developed [2].

First findings on the benefits of using higher-order 3D-shell elements for accurately modeling sheet metal forming processes were presented in [3]. The objective of this study is to expand upon this work by assessing the accuracy of simulations utilizing the higher-order 3D-shell element for critical sheet metal forming processes. Results of simulations with the higher-order 3D-shell elements are compared to experimental data and results obtained from simulations with solid elements and Reissner-Mindlin shell elements. It is demonstrated that simulations with higher-order 3D-shell elements provide more accurate predictions in sheet metal forming processes than the standard modeling approach, including but not limited to stress.

Furthermore, we aim to support the efficient utilization of the higher-order 3D-shell element by identifying situations in which the additional deformation modes of this element are beneficial, and in which application of a standard shell element suffices. To achieve this, we analyze the influence of its higher-order deformation modes on the strain for parameter alterations in benchmark problems. To aid the modeling decision, mesh studies are conducted to quantify the influence of the element size on the results quality. Lastly, a comparison of numerical efficiency of different element formulations is given, showing the high efficiency of higher-order 3D-shell elements compared to solid elements.
This contribution is part one of a two-part series that aims to present recent improvements of sheet metal forming simulations through a combination of higher-order 3D-shell elements and anisotropic 3D yield models. Part I focuses on the assessment of higher-order 3D-shell elements, while Part II investigates the effect of anisotropic 3D yield models with respect to the in-plane and out-of-plane behavior on sheet metal forming simulations. Together, these contributions aim to provide a comprehensive overview of the latest advances obtained in a joint research project at the Fraunhofer IWM in Freiburg and the Institute for Structural Mechanics at the University of Stuttgart.

2 Literature

[1] Fleischer, M.: „Absicherung der virtuellen Prozesskette für Folgeoperationen in der Umformtechnik". Thesis, Technical University of Munich, 2009.
[2] Willmann, T., Wessel, A., Beier, T., Butz, A., Bischoff, M.: "Cross-Sectional Warping in Sheet Metal Forming Simulations", 13th European LS-DYNA Conference, 2021.
[3] Schilling, M., Willmann, T., Bischoff, M.: „3D-Shell Elements for Improved Prediction Quality in Sheet Metal Forming Simulations", 16th LS-DYNA Forum, 2022.

Forming II

Higher-Order 3D-Shell Elements and Anisotropic 3D Yield Functions for Improved Sheet Metal Forming Simulations: Part II

Alexander Wessel[1,2], Maximilian Schilling[3], Tobias Willmann[3], Alexander Butz[1], Manfred Bischoff[3]

[1]Fraunhofer Institute for Mechanics of Materials IWM,
Woehlerstrasse 11, 79108 Freiburg, Germany
[2]Technical University of Munich, Chair of Metal Forming and Casting,
Walther-Meissner-Strasse 4, 85748 Garching, Germany
[3]University of Stuttgart, Institute for Structural Mechanics,
Pfaffenwaldring 7, 70550 Stuttgart, Germany

Abstract

This two-part series focuses on the industrial application of higher-order 3D-shell elements and anisotropic 3D yield functions in sheet metal forming simulations. In the second part, the effect of plastic anisotropy with respect to the in-plane and out-of-plane behaviour on sheet metal forming simulations is analysed. To this end, parameters of the anisotropic 3D yield function Yld2004-18p were identified by a crystal plasticity modelling approach for an AA6014-T4 aluminium alloy. Different loading conditions related to the plane and full stress state were carried out to study the plastic anisotropy with respect to the in-plane and out-of-plane behaviour. The results of the crystal plasticity simulations were utilised to identify parameters of the Yld2004-18p yield function considering three different data sets. The resulting parameter sets for Yld2004-18p were then applied to a sheet metal forming simulation of a generic car body part. All sheet metal forming simulations were carried out using higher-order 3D-shell elements. The results of this numerical study demonstrate that the plastic anisotropy concerning the in-plane behaviour has a higher relevance than the out-of-plane behaviour for the sheet metal forming process studied. Additionally, the results indicate that setting the parameters of Yld2004-18p associated to the out-of-plane behaviour to their isotropic values is a reasonable assumption for the sheet metal forming process analysed.

Simulation of Sheet Metal Forming – New Developments

Dr. M. Fleischer, C. Babel, M. Guru, Dr. J. Strauß-Ehrl

[1]BMW AG, Knorrstrasse 147, 80788 München

1 Introduction

The use of finite element (FE [12], [16]) simulations to conduct a virtual validation of the forming process for sheet metal parts has been introduced in the mid 1990s and is state of the art in the automotive industry today. Two challenging tasks for determination of feasibility of a tool design and its process parameters [17] are the prediction of the material behavior during the forming process and the springback of the final part [2,3,4].

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

As a result, the increase in data size and level of detail of the FE models poses a challenge for the future simulation systems and their application [5].

2 Forming simulation at the BMW Group – State of the art

Since the mid of the 1990s, forming simulation is state of the art in the tool development process and there are a lot of use cases today.

2.1 Simulation in the tool development process

The design process of a deep drawing tool is supported by FE simulation systems in many applications.

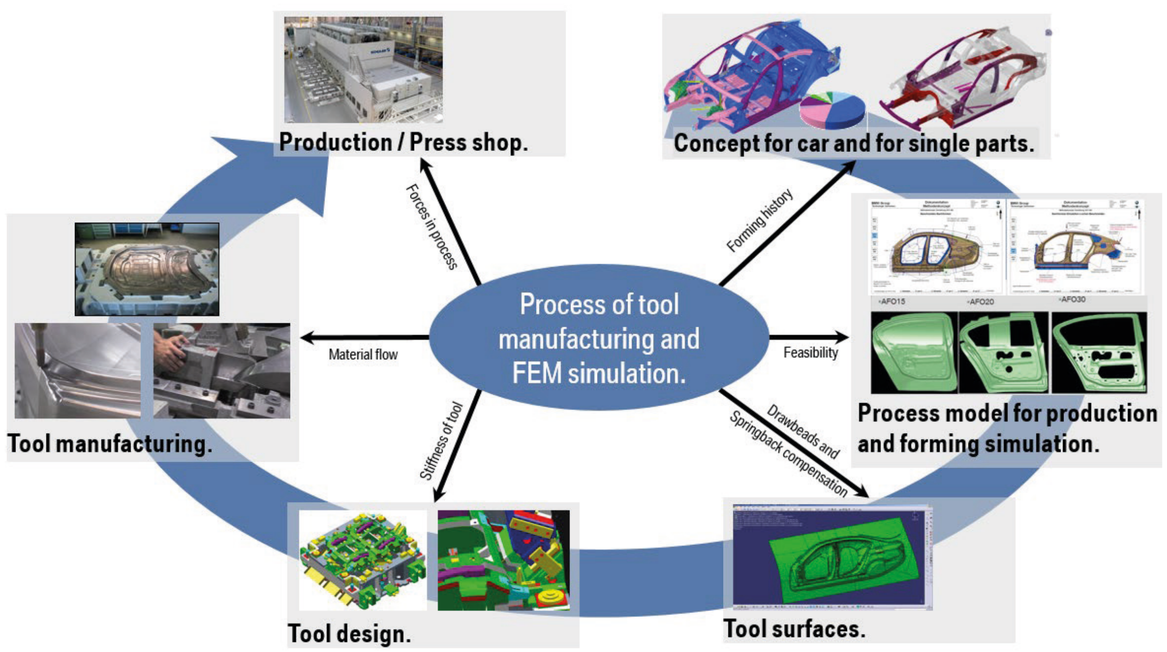


Fig.1: Application of simulation in the tool development process – [5].

In the design process of the car body, the forming simulation supports the simulation of passive safety with the material history from the production process to give a better prediction of the structural behavior of the car body in crash simulations. The main usage of the forming simulation is to model the production process itself and to conduct design studies for feasibility. When a good geometrical design for the forming tools is found, the next step is the springback simulation and its geometrical compensation. One more use case for the simulation is the prediction of stiffness of the tool to support the CAD engineer. More key results from the forming simulation are the forces that occur in the production process in the press line and the material flow during the deep drawing. All these applications of the FE simulation support the engineers in the design process.

2.2 Software concept

To deal with all the challenges in the design process, a general software setup is used for forming simulation. The input consists of geometries, material data and process parameters. These are all part of the classical pre-processing. The application was realized with BMW's own software systems to provide a process orientation to the forming process in graphical user interfaces [2,3,4].
This approach enables every user to setup a metal forming simulation with the FE solver LS-DYNA [1,9]. In the next step, the solution is computed by the solver and the results can be analyzed.

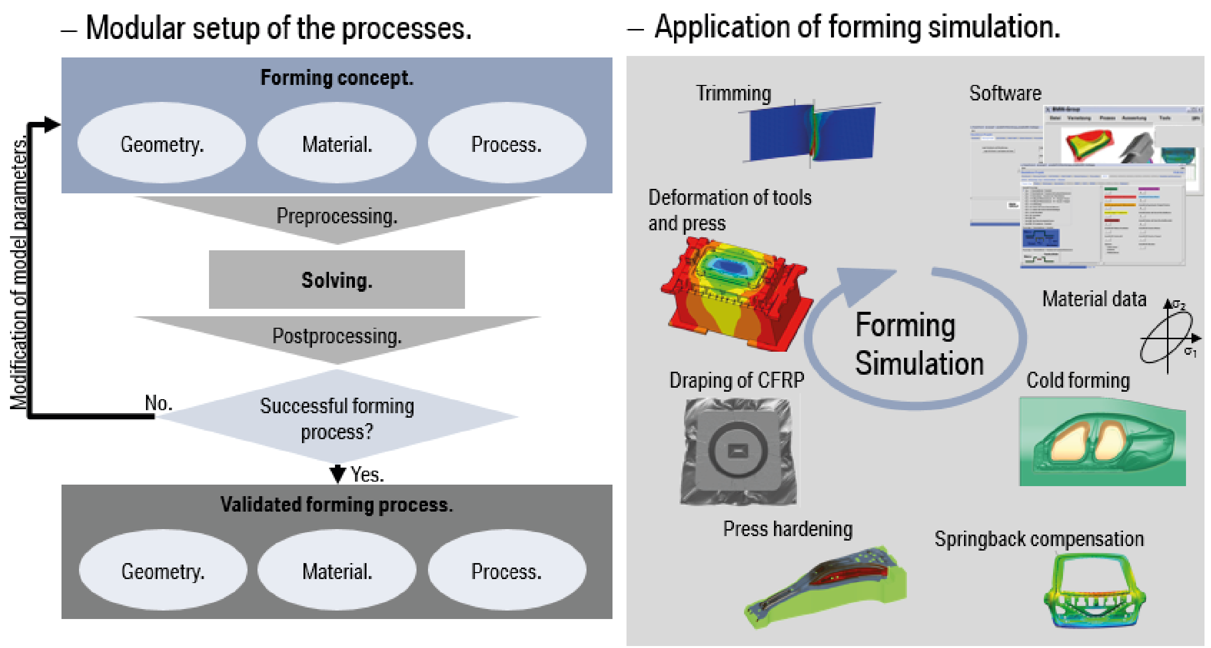


Fig.2: Software concept – [5].

This kind of simulation is used in a lot of cases in the engineering process, whereas cold forming simulation of automotive body parts is fundamental. After the cold forming simulation, the tools are springback compensated to reduce the final deviation from the target geometry. Further applications are indirect press hardening [13], draping of CFRP [14,15], calculation of the deformation of tools and the press at the end of the forming operation and the trimming.

2.3 Assumptions and limitations

The state-of-the-art simulation models for sheet metal forming have a lot of assumptions and restrictions [15]. For example, assuming all the tool surfaces as well as the whole press as a rigid body, the simplification of the displacement curve for the punch, the evaluation of necking and fracture with the FLD – limited to linear strain paths and flat geometries, etc.

One big point is the assumption on the behavior of the sheet metal blank as a shell body and the implementation of the FE shell elements with linear strain tensors through thickness with constant volume.

3 New developments

In the following, a short overview of why we need new shell elements with the capability to use 3D material models is given.

In classical deep drawing operations for body parts of a car, the usage of drawbeads is common. In some cases, two drawbeads are used in combination. Then, we've got six radii and then the draw in radius of the die.
All these radii are relatively small compared to the thickness of the blank, so that the assumption of a linear strain tensor in the shell elements is not valid.
Because of this the state of stress is not calculated in a correct way and in consequence, the prediction of spingback is not accurate.

The demand for a new shell element, with a nonlinear strain tensor over thickness is shown.

The EFB-Project “3D-Blechmodellierung” developed new shell Elements. The results on springback of a fender from a prototype deep drawing tool is shown, by using the new 3DBM13 Element in combination of a 3D yield model.

4 Summary

Shell element, rigid tools and the FLD have been state of the art for more than 10 years in evaluation of the forming simulation for the sheet metal blanks. In this presentation, the results of a new shell element in combination of 3D yield models are shown.

5 Literature

[1] Livermore Software Technology Corporation, LS-DYNA KEYWORD USER'S MANUAL VOLUME I R12, 2020.

[2] Meinhardt, J., Lipp, A., Ganser, M., Fleischer, M., Aspekte der Simulation Blechumformung im industriellen Umfeld, LS-DYNA Forum 2007, Deutschland.

[3] Meinhardt, J., Lipp, A., Fleischer, M., Neue Prozesse im Bereich Simulation Blechumformung, LS-DYNA Forum 2009, Deutschland.

[4] Fleischer, M., Panico, T., Meinhardt, J. Lipp, A., Anwendung der Simulation in der Technologie Umformen, LS-DYNA Forum 2011, Deutschland.

[5] Fleischer, M., Lipp, A., Meinhardt, J., Hippchen, P., Heinle, I., Ickes, A., Senner, T., Usage of LS-DYNA in Metal forming, Europäische LS-DYNA Konferenz 2015, Deutschland

[6] Fa. GNS, OpenForm, Europäisches LS-DYNA Forum 2013, England.; www.gns-mbh.com

[7] Fleischer, M., Sarvas, J., Grass, H., Meinhardt, J., Umformsimulationen, Schnittstellen und Prozesse, LS-DYNA Forum 2016, Deutschland

[8] Fleischer, M., Sarvas, J., Grass, H., Meinhardt, J., Forming simulation, meta language and input decks, LS-DYNA Conference 2017, Austria

[9] http://lstc.com/, 2022

[10] Livermore Software Technology Corporation, LS-DYNA KEYWORD USER'S MANUAL VOLUME 2 Material Models R12, 2020.

[11] Livermore Software Technology Corporation, LS-DYNA KEYWORD USER'S MANUAL VOLUME 3 Multi-Physics Solvers R12, 2020

[12] BELYTSCHKO, T., LIU, W.K., MORAN, B., Nonlinear Finite Elements for Continua and Structures. Wiley, 2008

[13] Hippchen, P., Merklein, M., Lipp, A., Fleischer, M., Grass, H., Craighero, P., Modelling kinetics of phase transformation for the indirect hot stamping process, Key Engineering Materials, Vol. 549, pages 108-116, 2013

[14] Senner, T., Kreissl, S., Merklein, M., Meinhardt, J., Lipp, A., A modular modeling approach for describing the in-plane forming behavior of unidirectional non-crimp-fabrics, Production Engineering, Volume 8, Issue 5, pp 635–643, October 2014

[15] Senner, T., Kreissl, S., Merklein, M., Meinhardt, J., Lipp, A., Bending of unidirectional non-crimp-fabrics: experimental characterization, constitutive modeling and application in finite element simulation, Production Engineering, Volume 9, Issue 1, pp 1–10, February 2015

[16] Wagner, M., Lineare und nichtlineare FEM, Springer Verlag, 2017

Experience with Crash Simulations using an IGA Body in White

Frank Bauer[1],Tian Yugeng[1], Lukas Leidinger[2], Stefan Hartmann[2]

[1]BMW AG, Germany
[2]DYNAmore, an ANSYS Company

1 Abstract

With the IGA (Isogeometric Analysis) technological approach, among other things, the transfer processes from CAD to CAE can be simplified in the future and false predictions due to discretization effects can be reduced.
In recent years, IGA and the toolset has increasingly developed into an setup that comes close to industrial use.
In order to test the use of IGA in industrial environments, a body in white that was previously modelled with „classic" FE was also created in IGA and installed as a so-called hybrid model in an entire vehicle crash simulation. For this purpose, the CAD data used as base for the FE model creation, was now used to directly create IGA surfaces.
The aim of implementing a body in white in IGA was, on the one hand, to look at the processes in terms of usability, automation capability and implementation quality and, on the other hand, how hybrid crash simulations behave in terms of computing time and stability.
In order to see different design effects in crash simulations, a front crash and a side crash were carried out and compared with previous FE models.
The presentation shows the entire process, from geometry conversion to overall vehicle simulation, and explains the findings in comparison with the FE model.

2 Keywords

Isogeometric Analysis (IGA), Automotive Crash Simulation, Shells, CAD/CAE integration

Systematic assessment of isogeometric sheet metal forming simulations based on trimmed, multi-patch NURBS models in LS-DYNA

Christoph Hollweck[1,2], Lukas Leidinger[3], Stefan Hartmann[3], Liping Li[4], Marcus Wagner[2], Roland Wüchner[1]

[1] Technische Universität Braunschweig, Institute of Structural Analysis
Beethovenstraße 51, D-38106 Braunschweig, Germany

[2] Ostbayerische Technische Hochschule Regensburg, Laboratory Finite-Element-Methods
Galgenbergstraße 30, D-93053 Regensburg, Germany

[3] DYNAmore GmbH, an ANSYS Company
Industriestr. 2, D-70565 Stuttgart, Germany

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

Isogeometric sheet metal forming simulation is a numerical simulation technique that is used to predict the behavior of sheet metal parts during the forming process [1] and tries to tighten the link with the Computer Aided Design (CAD) description. This technique uses the isogeometric analysis (IGA) approach, which combines the well-established framework of traditional finite element analysis (FEA) and the power of non-uniform rational B-splines (NURBS). In contrast to the approach in a "classical" FEA framework, IGA directly uses the ansatzspace of the CAD geometry for analysis, which opens the possibility to work directly on the exact geometry. Furthermore, the smoothness of the NURBS basis functions results in a more accurate simulation [2].

A powerful method to reduce the computational effort is adaptive mesh refinement, that has been developed and optimized for sheet metal forming applications over several years for standard Finite Elements. However, it remains an open question how an efficient local adaptive mesh refinement strategy can be implemented for complex industrial sheet metal forming simulations based on trimmed NURBS models, which are typically the description in Boundary Representation (B-Rep) CAD-models [3]. First investigations for explicit dynamics have been made in [4].

In this contribution, a detailed comparison between FEA and IGA sheet metal forming applications is conducted. The state of the art for FEA and IGA will be contrasted and the need for an efficient adaptive mesh refinement strategy will be discussed. The goal of our research is to develop an efficient adaptive mesh refinement strategy for isogeometric sheet metal forming simulations in LS-DYNA. This will contribute to closing the efficiency gap between IGA and FEA in explicit dynamics, accelerate the product development process and enable the application of IGA in industrial sheet metal forming simulations.

Literature

[1] S. Hartmann and D.J. Benson, Sheet Metal Forming with IGA in LS-DYNA, 15th International LS-Dyna Conference, 2018.

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

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

[3] Matthieu Occelli. Explicit dynamics isogeometric analysis : lr b-splines implementation in the radioss solver. Mechanics [physics.med-ph]. Université de Lyon, 2018. English. (PhD-Thesis).

Towards the Solution of Cross-Talk in Explicit Isogeometric B-Rep Analysis

Zeyu Lian[1], Lukas Leidinger[2], Stefan Hartmann[2], Roland Wüchner[3]

[1]BMW AG
[2]DYNAmore, an ANSYS Company
[3]TU Braunschweig

Abstract

Driven by the increasing need for seamless integration between Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE), Isogeometric Analysis (IGA) emerged with the groundbreaking research conducted by Hughes et al. in 2005. Unlike standard Finite Element Analysis (FEA) that typically employs C^0-continuous Lagrange polynomials as basis functions, IGA utilizes smooth spline-based basis functions, the same as the ones used to describe the CAD geometries. The usage of consistent basis functions offers the potential to bridge the gap between CAD and CAE. Industrial applications of IGA yet face a significant challenge. Although higher-order spline-based basis functions are undoubtedly beneficial in terms of enabling higher regularity of the solution field and larger time step sizes, this comes at a cost that each basis function spans over a larger support domain comparing to FEA. Therefore, a pair of spline basis functions may have overlapping supports even when the underlying control points are distinctly non-adjacent. Consequently, small-scale discontinuities within NURBS models, such as evolving discontinuities arising from element deletions or stationary discontinuities (e. g. notch, hole in the initial geometry) represented by trimming curves, and their corresponding mechanical characteristics cannot be correctly captured with a coarse mesh. In such situations, elements effectively remain interconnected across the discontinuity, violating the traction-free boundary condition. This phenomenon is referred to as Cross-Talk. It is widely recognized that the prevention of Cross-Talk can be achieved through a fine spatial discretization, particularly in areas where the length scale of the discontinuities is critically small. However, this approach faces limitations in the context of crash due to the reduction of the stable time step size. This contribution introduces a novel and effective solution that eliminates Cross-Talk, namely Control Point Duplication. The effectiveness of this approach will be demonstrated through a series of numerical examples.

Latest ANSA developments for IGA modeling

Lambros Rorris[1], Ioannis Chalkidis[2]

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

Abstract

Isogeometric Analysis (IGA) is maturing fast and offers certain advantages over classic FEA that makes it useful for large deformation dynamic analyses such as Crash applications in the automotive industry. 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 focus of our efforts 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. While single patch approach has been very successful, automatic multi patch creation has been the target of our latest efforts. Existing CAD or automatically recognized CAE features could potentially be an effective way to define multiple patches. In terms of user productivity, enhanced visual representations of the IGA parts with better performance and more accurate rendering will be introduced. Furthermore visual aids for several metrics like Jacobian, distortion distance from original geometry and control points cross-talking are added. Mapping vector fields like thickness, initial stress etc. is possible with the addition and support of the new respective LS-DYNA keywords. Tailor blanks and multi material parts are made possible using special modeling techniques as well. Recent LS-DYNA developments in the area of trimmed IGA solid modelling have shown great potential for modern cast parts. Preparation of the needed NURBS volume and the corresponding volumetric data though is not a trivial task and development efforts are taken in this direction. Support of unstructured spline technologies through BEXT format both for shell and solid parts is added throughout the program ensuring integration of the respective technologies in current workflows.

Incremental Forming Simulation of Dimples for Solar Mirror Supports using Isogeometric Analysis

Wade Evans[1], Johannes Pottas[1], Lukas Leidinger[2], Amit Nair[2], Fabian Knieps[3], Benjamin Liebscher[3], John Holmes[4], Joe Coventry[4]

[1]Timestep Pty Ltd.
[2]ANSYS
[3]thyssenkrupp Rasselstein GmbH
[4]The Australian National University

1 Introduction

Stamped supports are used as structural components in curved mirror facets for concentrating solar power (CSP) applications. A novel support with dimple-like protrusions, which is suitable for high-volume and low-cost production of high-precision mirror panels, has been developed at the Australian National University. Single point incremental forming (SPIF) is used for prototyping of mirror supports and may be adapted for medium-volume production for pilot-scale CSP plants. The envelope of design parameters for incrementally formed parts is limited by the maximum wall angle which can be achieved without causing the material to rupture. Isogeometric analysis (IGA) is evaluated in this study as a means of simulating the SPIF process to predict strain, thinning and displacement during SPIF. IGA and conventional finite element analysis (FEA) are compared to determine the relative modelling effort and computational cost, and simulation results from both approaches are compared with experimental results to assess the accuracy of the simulation methods.

2 Method

2.1 Experiment

A blank of rasselstein® TS 275 steel with 0.45 mm thickness is clamped in a specimen holder. The blank is incrementally formed by a hemispherical forming tool moving along a computer numerically controlled (CNC) toolpath. Strains at the lower surface of the specimen are measured in real-time using digital image correlation (DIC). The specimen geometry is chosen such that the wall angle increases with forming depth to induce failure. The experiment is continued until failure of the specimen is observed.

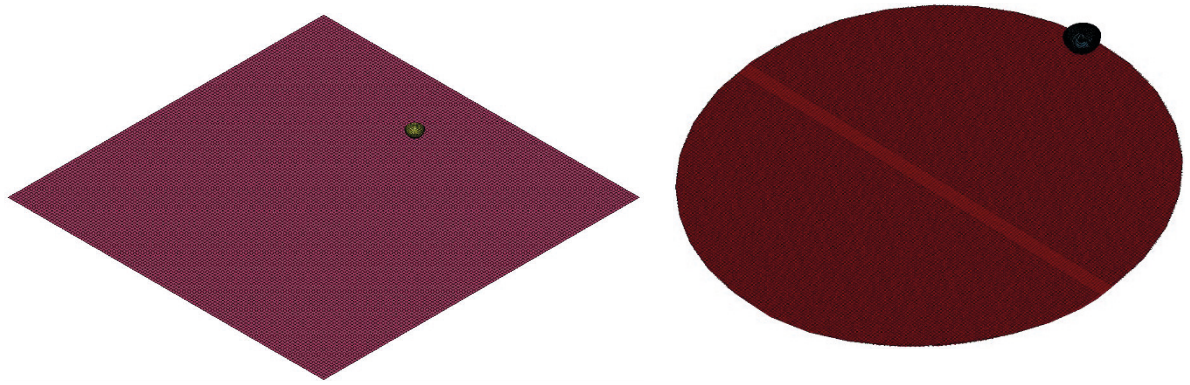
(a) (b)

Fig.1: (a) a partially formed SPIF specimen and (b) the underside of the specimen with speckle coating for DIC.

2.2 Simulations

The forming process is simulated with LS-Dyna explicit using conventional FEA and IGA approaches. In the IGA simulations, only the blank is modelled as an IGA part, whereas the tool is modelled as a

rigid FEA part (hybrid model). The *MAT_BARLAT_YLD2000 material model is assigned to the blank in both simulation approaches, without a failure criterion.

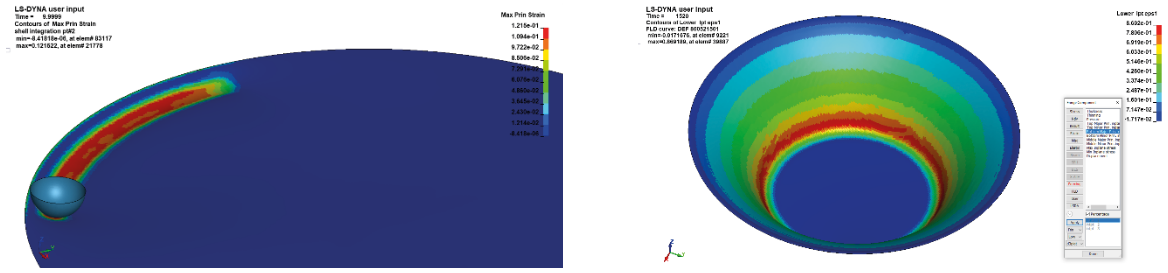


(a) (b)

Fig.2: The (a) conventional FEA model and (b) the IGA model.

3 Results

Results from the FEA and IGA simulations are presented, and challenges regarding pre- and post-processing of IGA results are discussed. Experimental results are presented with major strains exceeding 100% in some cases.



(a) (b)

Fig.3: IGA strain results (a) at initial contact between the tool and blank and (b) at completion of the forming process.

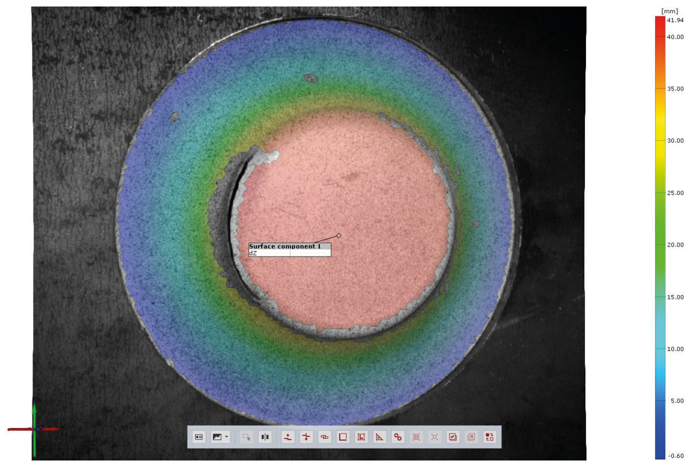


Fig.4: Measured Z-displacement at rupture mapped onto the lower surface of the specimen.

A Numerical Investigation of the Ballistic Performance of the Ceramic Composite Armor against EFP Threats

M. Bartu Ünal[1], M. Emin Akca[2]

[1]Nurol Teknoloji, Ankara/Türkiye
[2]Nurol Teknoloji, Ankara/Türkiye

1 Introduction

The increasing destructiveness of explosive-formed projectiles (EFPs) in modern warfare has posed significant challenges to developing effective armored solutions that incorporate advanced ceramics as key components, offering enhanced protection against high-velocity-formed projectiles. [1] In other words, Explosively Formed Penetrators (EFPs) pose a significant threat to military vehicles, necessitating the development of advanced armor solutions to counteract their destructive potential. Thus, analysis and simulation research is crucial for the armor system against this threat. In addition, since EFP tests are very expensive and time-demanding, performing these experiments with analysis and simulation provides significant cost and performance efficiency. In this study, we present an analysis of the composite armor system developed by integrating Nurol Technology [2] ceramics against the EFP threat using LS-DYNA [3] software.

2 Methodology

This study focuses on optimizing the fluid-structure interaction between EFPs (Explosively Formed Projectiles) and ballistic protection platforms based on simulations computed via LS-DYNA and LS-OPT software, utilizing Structured-ALE (Arbitrary Lagrangian Eulerian) with multi-material groups in 3D as the methodology for finite element analysis and simulations of EFPs. Structured-ALE uses a structured mesh with regular connectivity, allowing for a better representation of the deformation of the liner material during EFP formation. One of the main advantages of using Structured-ALE for EFP simulations is its ability to accurately model the complex material behavior involved in the formation of EFPs, including high strain rates and temperatures, as well as the interaction between the liner and explosive charge. Additionally, Structured-ALE allows for better control of mesh quality, which is critical for accurately capturing the deformation of the liner material. [4] A comprehensive numerical simulation was conducted using LS-DYNA, a widely recognized finite element analysis software known for its accurate representation of complex material behavior and high-fidelity simulations.

The composite armor configuration comprised multiple layers, with Nurol Technology ceramics strategically incorporated to exploit their exceptional hardness and fracture toughness properties. LS-DYNA allowed us to simulate the dynamic response of the armor system under EFP impact, capturing the complex interactions between the EFP and the composite layers.

The finite element method, implemented in LS-DYNA software, was used to investigate various parameters, including the ceramic layer thickness, orientation, and placement within the composite structure or the thickness, i.e., the number of the layers, of the structural polymer materials. Performance metrics such as penetration depth, residual velocity, and damage distribution were assessed to quantify the armor's effectiveness against the EFP threat.

Protective Structures II

3 Results

Preliminary results indicate that the integration of Nurol Technology ceramics significantly improves the armor's performance in resisting EFP penetration. The ceramic layers demonstrated exceptional capability in disrupting and deforming the EFP jet, effectively dissipating its kinetic energy. Furthermore, the positioning and orientation of the ceramics within the composite structure played a crucial role in redirecting and fragmenting the EFP, thereby reducing its overall penetration capability.

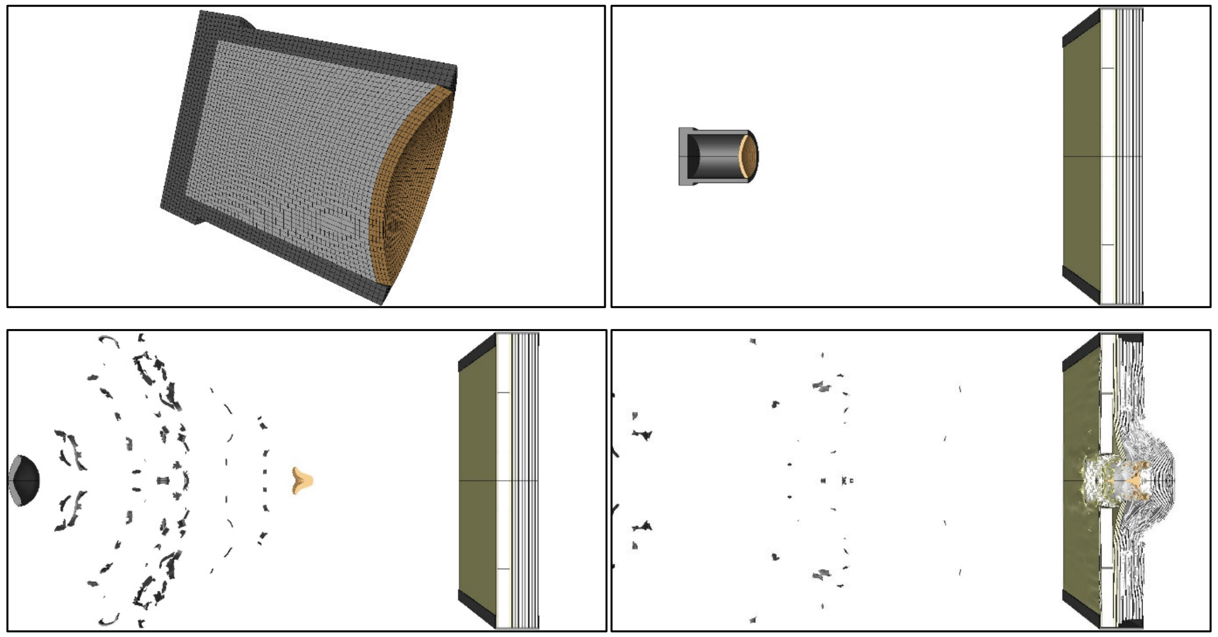


Figure 1. Section view of casing, explosive, and shape liner and images from the simulation at 0, 90, and 300 µs

4 Conclusion

The findings of this study demonstrate the performance of the ceramic composite armor against EFP threat. The numerical simulations performed using LS-DYNA provide valuable insights into the behavior of composite armor systems and offer a cost-effective approach for optimizing armor configurations without the need for extensive physical testing. Additionally, the study will focus on the implementation of these findings in real-world applications, ensuring enhanced protection for military vehicles against EFP threats.

5 Literature

[1] J. J. Morrison, P. F. Mahoney, and T. Hodgetts, "Shaped charges and explosively formed penetrators: background for clinicians," *J R Army Med Corps*, vol. 153, no. 3, pp. 184–187, 2007, doi: 10.1136/JRAMC-153-03-11.

[2] "Nurol Teknoloji." https://www.nurolteknoloji.com/ (accessed May 12, 2023).

[3] J. O. Hallquist, "LS-DYNA ® Theory Manual," 2006, Accessed: May 12, 2023. [Online]. Available: www.lstc.com

[4] "Introduction to Structured ALE | Livermore Software Technology Corp." https://ftp.lstc.com/anonymous/outgoing/hao/sale/ (accessed May 12, 2023).

Modeling Aluminum Honeycomb Under High Velocity Impact

Murat KAMBEROGLU

Roketsan

Abstract

Honeycomb is a family of a geometrically engineered products, offering many beneficial structural and functional design solutions by means of significant mechanical strength in specific spatial directions along with inapproachably low density. Beside its possession as core material in composite industry, remarkable energy absorbance behavior under destructive compressive loading is the reason of being utilized in many areas involving in impact and crash scenarios. This study focused on numerical modeling of such conversion of energy by compaction (up to 90%) of Al-honeycomb in solid continuum model, under impulsive loading induced by accelerated plate whose velocity is above 1000 m/s and impact area of 30 cm². Solution approach (Finite element, element-free methods), discretization inputs (element type and density) and MPP decomposition methods were investigated so as to minimize both simulation time and output deviation. While keeping the material model and inputs constant, pressure on donor stationery target were compared with both experimental findings and macro-scale modeling in which Aluminum foils have been directly modelled with shell elements. Determination of the best modelling approach offered both significant increase in solution efficiency and more reliable predictions in large aerospace designs.

Investigation of Improvised Explosive Device Effects on a Section Hull of Armored Military Vehicle

İsmet Kutlay ODACI, Samet Emre YILMAZ, İlker KURTOĞLU

FNSS Defense Systems Inc.
Ankara / TÜRKİYE

1 Abstract

Military vehicles and their occupants in conflict zones face a significant risk from improvised explosive devices (IEDs). Simulating IED risks on armored military vehicles requires employing various modeling approaches. However, to ensure the accuracy and effectiveness of these approaches, it is crucial to accurately transfer the explosive load onto the vehicle structures. This study aims to address this critical point by developing a methodology for selecting the appropriate vehicle components for load transfer and evaluating the proximity of the analysis model to live fire test results.

To investigate the efficiency of the established methodology, two numerical models representing different regions of a complete vehicle hull were constructed, mirroring the ones used in live fire tests. Test data obtained from the Hybrid III dummy, along with the plastic deformations occurring in the hull and subsystems, were compared with the analysis results. The findings revealed consistent outcomes between the test data and analysis results, validating the accuracy of the methodology.

The results emphasize the significance of accurately selecting the structures onto which the blast load is transferred during the modeling phase. Such precision plays a crucial role in designing military vehicles that meet structural integrity requirements and ensure occupant protection. This study contributes to enhancing the understanding of IED risks and provides valuable insights for optimizing military vehicle design and occupant safety measures.

Blast Mitigation Seat Simulations Using LS-DYNA®

Buğra Balaban, Ahmet Mete Sabah

Tempa Engineering & Commerce Ltd.

1 Abstract

Military vehicles are exposed to mine and explosive loads in operational conditions and the vehicle must have the appropriate protection level to prevent personnel injuries. Blast mitigation seats are one of the most important systems for personnel protection. In this study, mine blast and impact simulations were performed using the non-linear finite element code LS-DYNA® to examine the structural behavior of the blast mitigation seats. Blast simulations have been prepared considering the requirements of STANAG 4569-AEP55 VOL-2. Blast simulations were performed according to STANAG 4569-AEP55 VOL-2 requirements. Structured ALE method was used for blast simulations and the model consists of the ALE domain which includes soil, air and explosive definitions and the Lagrange domain for the 4x4 military vehicle. During the project, in the first step, crash tests and computational simulations were carried out using the detailed finite element model of the seat and LSTC Hybrid III 50th dummy. After the simulation results were compared with tests, the validated finite element model of the blast mitigation seat was used for the mine blast simulation in the second step. Consequently, the structural behavior of the blast mitigation seat was examined and measured dummy acceleration and forces were compared with the allowable levels which are defined in the STANAG 4569-AEP55 military standard.

***KEYWORDS:** Blast mitigation seat, mine blast, crash test, dummy

Recent Development of Incompressible Smoothed Particle Galerkin (ISPG) method and Its Applications in Manufacturing Process Simulations

Xiaofei Pan [1], Jingxiao Xu [1], Tung-Huan Su [1], C.T. Wu [1]

[1] Computational and Multiscale Mechanics Group, Ansys, Livermore, USA

1 Abstract

Many manufacturing processes, e,g., adhesive flow, laser welding, reflow soldering, thin film coating, involve free surface flow with strong surface tension effect, large variation of dynamic viscosity, phase changes and coupling of complicated structures, which bring challenges to conventional CFD solvers. Incompressible Smoothed Particle Galerkin (ISPG) [1-4] is Lagrangian particle-based method, and has emerged as an accurate, efficient, and robust solver for solving the incompressible Navier-Stokes equation. ISPG was first released in LS-DYNA in 2019 and has been successfully applied in simulation of surface tension-dominant free surface flow [2-4]. Recently, several major developments have been done for ISPG in LS-DYNA: 1), fully implicit Newton-Raphson iteration algorithm for ISPG to enable it to handle strong surface tension effect and non-Newtonian behavior with large variation of fluid viscosity more efficiently. 2), development of nodes-cloud based adaptivity makes ISPG more accurate in the coupling among fluids and structures and capable of simulations of more complicated fluid behavior, e.g., liquid blob effect. 3), development of thermal-related features for ISPG, e.g., temperature effects for Newtonian and Non-Newtonian material modeling, thermal transfer in fluid and coupling with structure.

In this paper we demonstrate how ISPG method with above new developments brings the insights to the different manufacturing processes such as adhesive flow simulation with considering the non-Newtonian fluids effects, the laser-welding simulation with modeling molten pool and phase changes due to the thermal transferring, and the solder bridging phenomenon of the large-scale solder reflowing process in IC packaging industry.

***KEYWORDS** ISPG, Incompressible, Surface tension, adhesive flow, reflow soldering, laser welding

2 Literature

[1] 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 Engineering, 121(17), 2020, 3979-4002

[2] Pan, X., Wu, C. T. and Hu, W.: " Incompressible Smoothed Particle Galerkin (ISPG) Method for an Efficient Simulation of Surface Tension and Wall Adhesion Effects in the 3D Reflow Soldering Process", 16th International LS-DYNA Users Conference, 2020

[3] Sengupta, A., Pan, X, etc.: " Predicting Solder Shape Evolution During Solder Reflow in Packaging Assembly Processes", NAFEM World Conference, 2021

[4] Pan, X., Xu, J., Lyu, D., Hu, W. and Wu, C. T..: “Introduction of implicit incompressible smoothed particle Galerkin method for solder reflow analysis”, USACM Thematic Conference on Meshfree and Novel Finite Elements with Applications (MFEM), September 25-27, 2022, Berkeley, California, USA

Recent developments in discrete element method (DEM)

Jason Wang, Mohammed Mujtaba Atif

Ansys Inc.

Abstract

DEM (Discrete Element Method) was originally proposed to effectively simulate granular models such as sand, clay, water, etc. Another approach is developed to account for the elastic and brittle behavior of materials. The method has been successfully implemented in the LS-DYNA to study cementitious, sintered, and rock materials such as concrete, fibers, rocks, etc. Explosives with the combination of rock, concrete, and other brittle materials are simulated using LS-DYNA and studied extensively by researchers. In such cases, the volume of the fragments and number of fragments would be beneficial to the researcher. To assess this, we have developed a system that computes the number of fragments, fragment volume, and number of bond links associated with the DEM particles. Users have the option of outputting the results described above, depending on the user control card by using **`*DATABASE_DEFRAGMENT`**. Additionally, DEM has been extended to simulate the likes of intumescent material through new keyword **`*DEFINE_DE_TEMP`**. The DEM formulation has been extended to include the temperature effect on the radius of DEM particles. The method takes the user's temperature with several inputs and expands the radius of the particle based on the temperature and coefficient of thermal expansion. The LS-DYNA/MPP decomposes the problem once based on its initial geometry. For good MPP parallel efficiency, the deformation of the geometry should not be too large to keep reasonable ratio between computation and communication time. However, most DEM problems are dealing with granular flow which involve mixing and large relative displacement between particles. The parallel efficiency degrades due to increasing data communication. A re-decomposition algorithm supports DEM features which can reduce computing costs by 30%. This algorithm would rearrange the grouping of the particles, thus avoiding the search for neighbors that are far away from the group.

Particle Methods

An enhanced discrete sphere bond model based on continuum mechanics

Mohammed Mujtaba Atif, Jason Wang

Ansys Inc.

Abstract

In the last three decades, DEM (Discrete Element Method) has been extensively developed and applied to solve many geotechnical and petroleum engineering problems. Recently, researchers successfully implemented the idea of bonded DEM to simulate the mechanical behavior of elastic and brittle materials. Several bonded DEM models have been proposed to study cementitious, sintered, and rock materials such as concrete, fibers, rocks, etc. Despite developments in DEM for simulating complicated particle behavior, there has been little success in creating DEM models that explain non-linear behavior. Due to these limitations, the bonded DEM model is not able to capture the wide range of industrial material behavior. Although the formulation has been refined by several researchers to capture certain materials, it does not capture numerous typical materials. To address this issue, a robust bonded DEM formulation is developed for the discrete spheres called a Heterogeneous Bond Model. The mechanical formulation is developed to handle various material behaviors based on continuum mechanics with the combination of bonded DEM formulations. The resulting bond model inhibits energy and linear momentum conservation laws, which can be adopted to allow damage between the bonded spheres. Similar to the bonded DEM models, the breakage of particles can be accounted for by breaking the link between the particles. The method has been examined for a variety of benchmark problems involving elastic, plastic, and brittle material characteristics. Overall, the simulation result demonstrates that the method is stable and works effectively for the various types of materials. The user can define this model by using **`*ELEMET_DISCRETE_SPHERE_VOLUME`** to define each discrete sphere. The required parameters for the Heterogeneous bond model can be defined using **`*DEFINE_DE_HBOND`**. Numerous material behaviors are taken into consideration by using the constitutive model, which can be defined by **`*MAT_`**.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Study on Analytical Verification Method for Dynamic Load PROFILE-based Joint Design

JunHyeok Kim[1], SangDeok Kim[2], Kyoung Taek Lee[2]

[1] HYUNDAI MOTOR COMPANY
[2] Korea Simulation Technologies

1 Abstract

Bolt fastenings are widely used as a means of joining structures in various industries, but they have the disadvantage of being vulnerable to self-loosening and preload loss.

When external forces are repeatedly applied to the joint, the preload of the bolt decreases. If this reduced preload cannot withstand the shear forces generated by external forces, it will lead to slippage between the parts and eventually bolt loosening.

In order to prevent bolt self-loosening, it is necessary to calculate the minimum bolt preload to prevent slip due to shear forces. To calculate this, prediction of the minimum preload for slip prevention is required, along with consideration of factors that contribute to loss of preload due to system assembly, plastic deformation, embedding, and tolerance.

To calculate the minimum bolt preload for slip prevention, it is necessary to consider the shear force generated by external forces, and this value is calculated using a simulation method based on FEA. Until now, the results of applying the equivalent load of external force acquired through the durability test to a static analysis model were utilized, so dynamic characteristics such as inertia and acceleration were not reflected, limiting the accurate calculation of the minimum preload for slip prevention.

This study aims to apply the dynamic load profile to the analysis model to simulate the behavior of the joints more closely to the actual phenomenon, so that various dynamic phenomena that cannot be found in the static analysis can be analyzed and used to calculate the minimum preload for slip prevention.

A study on the bolt modeling with pre-load for field application

Han deok hee[1], Kim dong hyeon[1], Kim young joon[2],

Lee kyoung teak[3], Ahn yi moon[3]

[1]DAS Co., Ltd. CAE Solution Team
[2]DAS Co.,Ltd. Future Mobility Lab
[3]KOSTECH Co.,Ltd. R&D Team

1 Abstract

When fastening a structure with a bolt, an axial force is generated by the tightening torque of them. This axial force acts as a friction force by the friction coefficient of the fastening part, and becomes a factor that directly affects the deformation of the fastening part. For this reason, there have been many studies on how to make the bolt models for applying preload and users are using various methods. What is common is the construction and evaluation of bolt models with preload requires a lot of working by user. So this study was conducted because it was necessary to easy and exact method for with preload bolt models.

The research consists of two main parts. The first main is to validate the proposed bolt modeling method. We used *initial_stress_section keyword as the main. The checklist is as follows: First, whether target preload is applied. Second, whether there is a problem in the progress of FEA while preloading is applied. Third, whether there is any problem with the configuration of the fastening part of the bolt. The second main is the easy pre-processing and post-processing. We made some rules and used them to automate pre and post processing. The commercial software used is LS-DYNA & Hypermesh.
The bolt modeling method in the pre-processing was made possible by simple selection through the GUI and we were able to have results by automatic when analyzing the results in post-processing.

By applying our research to the field, we were able to reduce the work time of making modeling by more than 90%, excluding human errors at pre-processing. At the same time, we were able to get the results intuitively in the post-processing process.

A new apporach for the modeling of point-shaped multisheet connections in LS-DYNA

Max Huebner[1], Tobias Graf[1], Markus Feucht[2]

[1]DYNAmore GmbH an Ansys Company
[2]Mercedes-Benz AG

1 Abstract

Point shaped connections like spotwelding and riveting are still one of the main joining techniques in car construction. Therefore the prediction of the behavior of the spotweld connections is very important for the overall crash analysis. One method for the simulation of point shaped connections like spotwelds in LS-DYNA is the use of ***MAT_SPOTWELD_DAMAGE_FAILURE/DAIMLER**. This method provides several different failure functions and allows the use of the ***DEFINE_FUNCTION** keyword by which the parameters of the connected sheet materials can directly be used for the calculation of the failure properties. This method provides a great variability and enables a very efficient handling especially if many parts made up of different materials of various sheet thicknesses are being connected in a model, as it is the case in full car crash models.

The special case of multisheet connections where more than two metal sheets are being joined together in one single connection has previously been modelled by connecting two parts with an individual connection element whilst the properties of this connection element depend only on the adjacent sheet parts. However this method seems to not capture some failure modes of these multisheet connections which were observed in experiments. Therefore the new keyword ***DEFINE_MULTI_SHEET_CONNECTORS** has been introduced which allows to handle up to 4 connected sheets. The failure behavior of the joining elements may hence be calculated based on the material and geometric properties of all connected sheets involved.

LS-PrePost support for *IGA keyword entities

Gunther Blankenhorn, Philip Ho
Ansys Inc.

PyDYNA - Integrating LS-DYNA into pythonic workflows

Raphael Heiniger
Ansys Inc.

Displacement based Simulation and Material Calibration based on Digital Image Correlation

Christian Ilg
DYNAmore GmbH, an Ansys Company

Efficient processing method for material card definition of AHSS and UHSS to predict fracture in crash analysis and its application to vehicle crash model

Kang Hee Lee[1], Dae Young Kim[1], ChulWoong Jun[2], Kyoungteak Lee[2]

[1]Hyundai Steel Co., Ltd.
[2]Korea Simulation Technologies Co., Ltd.

1 Abstract

Crashworthiness of the vehicle body is getting importance to meet the enhancing vehicle crash safety regulations. To improve the vehicle body crashworthiness, application of Advanced high-strength steels (AHSS) and Ultra high-strength steels (UHSS) are continuously increasing due to their superior strength than conventional high-strength steels (HSS). However due to the lower ductility of AHSS and UHSS, material fracture can occur during crash scenario and it affects the overall performance of vehicle. To prevent this problem, CAE tool has important role in design stage of the vehicle body. CAE material model is steadily increasing and many fracture models are developed to predict the material fracture phenomena in crash scenario. Especially, Generalized incremental stress state dependent model (GISSMO) is widely used to predict the material fracture in crash scenario. Material card for crash simulation needs plasticity curves at various strain rates to predict the material behavior at high speed deformation and GISSMO parameter to predict the material behavior at high amount of local deformation. However, due to many parameters and data has to be defined for each material cards, the definition process is very complicated and time consuming. So, efficient processing method is required to effectively define material cards for various steel grades applied to vehicle body structure.

In this study, a user interface program is developed and verified to efficiently process the material card definition for crash analysis. LS-OPT and LS-DYNA linked calculation method included in this process helps to define material cards with reliable and effective results. Material cards for AHSS and UHSS for vehicle body structure were defined and verified. 5 Types of coupon test specimens and some component test specimens were used. Finally, material cards were introduced in H-Solution EV concept vehicle model which developed by Hyundai-Steel and 3 crash load cases were implemented. Analysis result of 2 types of vehicle crash model which introduce different material grades for the major body parts help to evaluate the crashworthiness result of each material grades and helps to predict the material behavior in vehicle body crash scenario. According to this study, it can be shown that the material card for crash analysis can be efficiently defined with the process and helps to predict the performance of the vehicle body structure by applying it into crash scenario model.

ML in the Detection of Fractures for the Characterization of Materialcards

Bernhard Jilka[1], Dr. Martin Schwab[1], Seraphim Amvrazis[1]

[1]4a engineering GmbH

1 Abstract

In recent years 4a engineering developed a couple of new test methods to support the characterization of Materialcards. One of those test methods is the puncture test. The benefit of this test method is that we can easily acquire test data that is dominated by a bi-axial tensile load.
Another field of activity in the recent years has been the calibration of fracture parameters in material cards. To support that calibration 4a engineering is using highspeed imaging to get the correct point in time where the fracture occurs. This has been a manual task for the person that conducts the test.
As we strive to automate as much as possible, we need to automate the detection of the fracture time. Thermoplastic materials can have varying optical and mechanical properties it is almost impossible to write a conventional algorithm to reliably detect the point in time where fracture occurs.
This is where ML comes into play. We have successfully developed and trained a ML model to find the most likely point in time where fracture occurs in a puncture test.
In the presentation it will be shown that a deep learning ML model can predict the correct fracture time for a large variety of thermoplastics and similar materials.

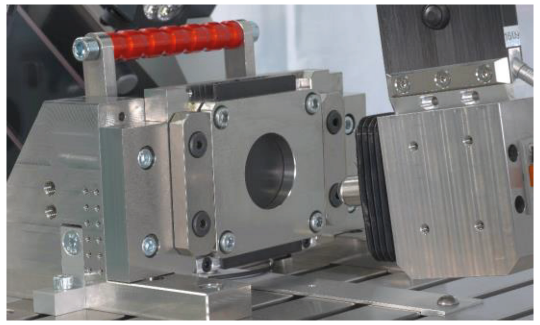


Fig.1: Puncture test

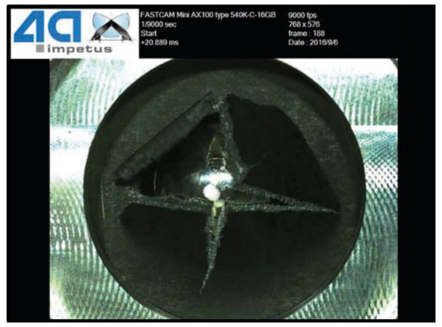


Fig.2: Highspeed video of a Puncture test

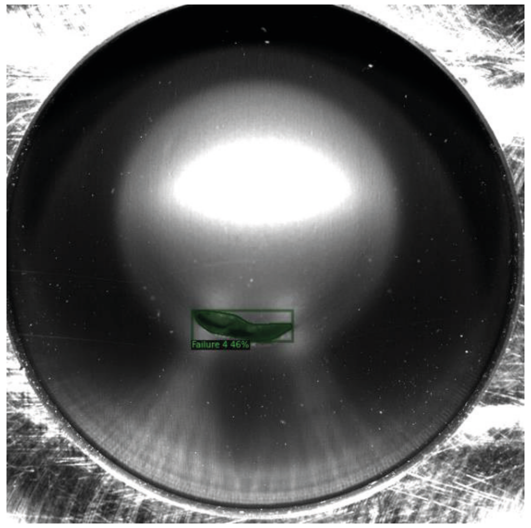


Fig.3: Ductile Material

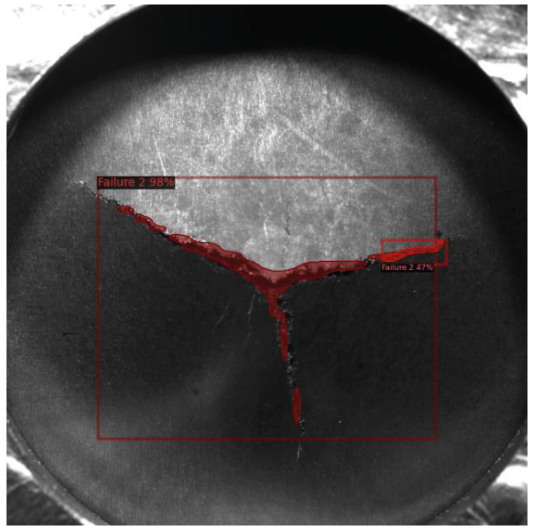


Fig.4: Brittle Material

Update on Workflow Based Material Calibration for Metals, Thermoplastics and Joining in d3VIEW with AI

Suri Bala[1]

[1]d3VIEW, Rochester Hills, USA

Abstract

Material models provide the basis for simulations. Calibrating accurate material models for LS-DYNA has long been a tedious task and requires substantial development time. The need for automation is greater today that ever before as the need for calibrated materials increases for quick valuation of complex designs. Conventional methods rely on manual efforts, prone to human errors, and can take up months of effort before the material models can be reliably used in simulations.

The current presentation will provide an overview of the Workflow based calibration in d3VIEW that has demonstrated substantial reduction of time and effort while increasing the accuracy of the calibrated models due to early use of ML/AI. This paper will also showcase real-world examples that includes calibration of metals, thermoplastics and joining materials such as adhesive, spotwelds, using workflows.

References

/1/ Experience with material and fracture modeling at Fiat Chrysler Automobiles (FCA), Dr. Anantharam Sheshadri, Hamid Keshtkar, Ashutosh Patil, Paul Du Bois[2 *] ,16th International LS-DYNA Users Conference, June 10-12, 2018, Dearborn, MI

A decoupled multiphysics approach for computationally efficient continuous induction welding simulation of CFRTPs

Thomas Hoffmann, Miro Duhovic, Peter Mitschang

Leibniz-Institut für Verbundwerkstoffe GmbH, Rheinland-Pfälzische Technische Universität Kaiserslautern-Landau, Germany
* Corresponding author (thomas.hoffmann@ivw.uni-kl.de)

1 Abstract

Continuous induction welding is a key process in the move towards the use of more carbon fiber reinforced thermoplastics (CFRTPs) in aircraft structures. One of the main targets is the assembly of longitudinal stringers in fuselage sections and ribs to skins in wing structures representing several hundred meters of joining. Due to the complex multiphysics nature of the process, simulation plays an important role, however present day simulation methods are just too slow to allow the necessary parameter studies. In addition, current continuous induction welding modelling approaches generally only make use of simplified macroscopic material models where laminate layup specific heating patterns cannot be modeled. More detailed mesoscopic models, where the actual laminate layup configuration is discretely modeled via orthotropic UD-layers, result in further increases in calculation times. In this work, a simulation methodology for continuous induction welding incorporating a mesoscopic electromagnetic material model with fast calculation times has been developed. To simulate continuous induction welding three physics solvers in LS-DYNA are used. The electromagnetic solver calculates the resistive losses in the laminate. These losses are then used by the thermal solver to determine the temperature distribution in the material. The mechanical solver calculates the relative movement of the laminate to the induction coil. In a fully coupled continuous induction welding simulation, the three solvers cycle through many calculation steps passing on information between one another leading to very high calculation times. The most computer intensive part of the entire calculation is the electromagnetic calculation. In order to minimize the computation time, a decoupled approach was developed here which can be described in two sequential steps. In the first step, a one-time electromagnetic calculation of the resulting heating pattern in a mesoscale partial model of the weld geometry is carried out. The heating values assigned to the elements of the simulation model are then transferred via a script to a user-defined volumetric heat source. In the second step, the volumetric heat source is used to carry out a continuous macro-level thermal-mechanical simulation of the induction welding process. Using this approach, simulation times for lab-scale weld geometries (400 mm x 75 mm x 4 mm) can be reduced from several days to less than 12 hours using desktop computing resources.

Multi-phase welding simulation, experimental validation and exploratory bending simulation of structural steel weldments

Tobias Girresser[1], Tobias Loose[2]

[1]Technologie-Institut für Metall und Engineering GmbH
[2]Dr. Loose GmbH

1 Abstract

To predict the load-bearing capacity of a welded joint, it is necessary to know the structure that will be formed. In recent decades, welding simulation has evolved and now offers the possibility to determine the required material properties after welding [1]. Computational welding mechanics (CWM) is a calculation method that can be used to calculate distortions, mechanical stresses, and strains as well as microstructure states and microstructure transformations in thermally joined components [2, 3].

The material models for CWM consider several phases, but not descriptions of the damage. The material model developed by [4], which is specially tailored to the requirements of welding and heat treatment simulation, is available as keyword ***MAT_GENERALIZED_PHASE_CHANGE (*MAT_254)**. It contains, among other things, the calculation of the microstructure evolution of up to 24 individual phases, whereby various phase change mechanisms are integrated for the phase transformations (e.g., Leblond, JMAK, Koistinen-Marburger, Kirkaldy). In welding simulation, the processes associated with phase transformations: heating, cooling and reheating in multi-layer weld seams, including the tempering effect, can thus be modeled.

For crash simulations, in which limit load capacities are also determined, the states calculated with CWM represent a starting point but cannot yet be used because the single-phase material models of the crash simulation are often inconsistent with CWM. This leads to uncertainties in the simulation results, especially in the prediction of load-bearing capacities.

In this talk, CWM with the material model ***MAT_254**, validation experiments and a virtual damage study of weldments will be presented. The software FabWeld® [5] is used to pre-process the welding simulation and processes the temperature-dependent material data from JMatPro® into the multiphase material model. Structural steels of different strengths are therefore considered. The validation experiments are carried out with a collaborative robot and a GMAW welding power source. They include the time-dependent recording of the welding parameters, the recording of the melt pool with a welding camera as well as thermography over a temperature range that takes the heat affected zone and the melted zone into account. The results will be considered with an exploratory simulation study of the damage to the welded joint in a virtual 3-point bending test.

2 Literature

[1] Goldak, J.A., Akhlaghi, M.: "Computational Welding Mechanics", Springer, Boston, MA, 2005, ISBN: 9780387232874
[2] Loose, T.: "Einfluß des transienten Schweißvorganges auf Verzug, Eigenspannungen und Stabilitätsverhalten axial gedrückter Kreiszylinderschalen aus Stahl", 2007
[3] Hildebrand, J.: "Numerische Schweißsimulation - Bestimmung von Temperatur, Gefüge und Eigenspannung an Schweißverbindungen aus Stahl- und Glaswerkstoffen", 2008
[4] Loose, T., Klöppel, T.: "A LS-DYNA material model for the consistent simulation of welding, forming and heat treatment", 11th International Seminar Numerical Analysis of Weldability, Seggau, Austria, 2015, https://dr-loose-gmbh.de/onewebmedia/2015-Seggau-Material.pdf
[5] Loose, T., Girresser, T., Goldak, J.A.: "Validation of Welding Structure Simulations", 13th International Seminar Numerical Analysis of Weldability, Seggau, Austria, 2022

Thermal Boundary Conditions for Simulation of a new forming process for Titanium (TISTRAQ)

Mathias Merten
DYNAmore GmbH, an Ansys Company

IGA technology validation for crashworthiness CAE analysis, including advanced plasticity and ductile fracture

Lluis Martorell[a], Pablo Cruz[a], Alejandro Domínguez[a], Adrián Ros[a]

Courtesy of: Andrew Hall[b], Patrick Kelly[b], Ilyasuddin Syed[b]

[a]Applus IDIADA

[b]Faraday Future

1 Abstract

Isogeometric Analysis (IGA) has gained considerable attention as a technology bridging the gap between CAD and CAE domains, promising a streamlined workflow. IGA presents a novel approach for incorporating design features as optimization variables during the entire design process, from pre-concept phases to final product release. IDIADA's work is focused on the IGA potential for producing more efficient and lightweight car body designs through improved accuracy in capturing crash deformation patterns and ductile fracture.

Before IGA integration into actual production, it is essential to validate the reliability and accuracy of IGA technology. To address this, a comprehensive validation study has been conducted by IDIADA, comparing IGA results with coupon and component testing, alongside state-of-the-art FE analysis results. The investigation specifically focuses on crash deformation patterns and material fracture, with particular emphasis on tackling challenges like the Cross-Talk effect.

This work presents ongoing research and efforts to find effective solutions to the identified challenges and to demonstrate the significance of IGA technology applied to car body design. The validation activities were carried out in collaboration with an Industrial Doctoral PhD program, in-house innovation initiatives, and actual production projects.

Enabling Productive Use of Isogeometric Shells in LS-DYNA

Lukas Leidinger[1], Stefan Hartmann[1], Frank Bauer[2],
Dave Benson[3], Attila Nagy[3], Liping Li[3], Marco Pigazzini[3], Lam Nguyen[3]

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

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

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

1 Abstract

Isogeometric Analysis (IGA) [1] is a novel Finite Element Analysis (FEA) technology based on splines known from Computer Aided Design (CAD). In an isoparametric sense, IGA uses the higher-order and higher-continuity spline basis functions, e.g. Non-Uniform Rational B-Splines (NURBS), not only to describe the model geometry, but also the solution field. This may yield a more accurate geometry description, higher solution accuracy and a larger time step in explicit analysis compared to conventional FEA.
With the goal to exploit these potential benefits in productive industrial use, the IGA capabilities in LS-DYNA have been successively extended and improved over the last few years with a focus on vehicle crash simulations and trimmed (multi-patch) NURBS shells [2]. A simple and convenient way to apply IGA in large-scale crash simulations is to replace certain components of existing FEA vehicle models with their IGA counterparts, resulting in hybrid IGA/FEA models. To enable such a 1:1 component exchange, LS-DYNA was enhanced to support the state-of-the-art FEA connection technology for spotwelds, seam welds, bonds and rigid bodies also for isogeometric shells. This includes a wide range of penalty- and constraint-based tied contacts and constraint definitions. As will be shown by Bauer et al. [3] at this conference, LS-DYNA is now able to run crash simulations of full vehicle models with hundreds of isogeometric shell components.
This contribution shall provide more information about the underlying developments and capabilities that were necessary to achieve this. Another crucial aspect to be discussed is the possibility to use the existing elasto-plastic material cards including damage and failure for isogeometric shells, ideally without modifications. Furthermore, recent developments such as initialization of isogeometric shells with mapped material history data, and the definition of multi-thickness and multi-material patches for tailor-rolled/-welded blanks are presented. Finally, an outlook is provided to a more sophisticated, feature-based IGA modeling approach for a tight connection between the CAD and the simulation model.

2 References

[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] Leidinger, L., Hartmann, S., Benson, D., Nagy, A., Li, L., Pigazzini, M., Nguyen, L., Bauer, F.: "Isogeometric Shell Components in Full Vehicle Crash Simulations: Hybrid Modeling", 16th LS-DYNA Forum 2022, Bamberg, Germany.
[3] Bauer, F., Yugeng, T., Leidinger, L., Hartmann, S.: "Experience with Crash Simulations using an IGA Body in White", 14th European LS-DYNA Conference 2023, Baden-Baden, Germany.

Trimmed IGA B-Spline Solids vs. Standard Tetrahedra Finite Elements

Stefan Hartmann[1], Lukas Leidinger[1], Frank Bauer[2],
Dave Benson[3], Attila Nagy[3], Liping Li[3], Marco Pigazzini[3], Lam Nguyen[3]

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

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

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

1 Abstract

The initiation of the numerical analysis methodology *Isogeometric Analysis* (IGA) dates to the paper by Hughes et al. [1] in 2005. Its main idea, compared to standard *Finite Element Analysis* (FEA) is to utilize the same shape functions used in the *Computer Aided Design* (CAD) environment for numerical analysis. In the last decade, the development of the IGA technology in LS-DYNA was mainly focused on thin-walled structures, using IGA shell elements based on B-splines and non-uniform rational B-splines (NURBS).
However, there is a trend towards more and more complex engineering parts, that cannot be classified as thin-walled structures. Its proper numerical analysis requires accurate solid-type finite elements. If the geometry of the part is rather complex, the current state of the art is to use low-order, linear tetrahedra-type solid finite elements, which are known to perform rather poorly.
In recent years the potential benefits of using higher-continuity B-Spline basis functions for developing solid-type finite elements have been successfully shown in prototypical developments by Meßmer et al. [2], [3]. This led to an increased interest from customers and thus to additional developments for solid-type IGA elements in LS-DYNA.
This paper describes the basic concept of trimmed IGA B-spline solid finite elements and its current capabilities, available in LS-DYNA. The influence of higher-continuity basis functions on the maximum critical timestep for trimmed finite elements in an explicit time integration scheme is explored to demonstrate the superior characteristics of using B-spline basis functions compared to linear Lagrange-polynomials. Numerical studies on a number of selected examples are presented, to compare the performance of these new type of solid elements with respect to standard, low-order, linear tetrahedra elements.
The paper closes with some conclusions on the current status and an outlook on future development activities.

2 References

[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] Meßmer, M., Leidinger, L, Hartmann, S., Bauer, F., Duddeck, F., Wüchner, R., Bletzinger, K.-U.: "Isogeometric Analysis on Trimmed Solids: A B-Spline-Based Approach focusing on Explicit Dynamics", 13th European LS-DYNA Conference 2021, Ulm, Germany.

[3] Meßmer, M., Teschemacher, T., Leidinger, L.F., Wüchner, R., Bletzinger, K.-U.: "Efficient CAD-integrated isogeometric analysis of trimmed solids", Computer Methods in Applied Mechanics and Engineering, Vol. 400, 2022, 115584.

Numerical simulation of shock events and associated response of satellite

LEGAUD Tess[1], VAN DORSSELAER Nicolas[1], LAPOUJADE Vincent[1], LEMAY Simon[2]

[1]Dynas+
[2]CNES

Abstract

The paper presents CNES' recent activities to improve shock events simulation and predict shock propagation in structures and equipment. The last activities presented here have been focused on shock test prediction by numerical analysis (i.e. virtual shock testing). The simulations were performed using LS-DYNA®, whereas the use of explicit non-linear computation codes is not common in the space industry to deal with spacecraft mechanical environments. Especially, the activities aimed at modelling physically the shock event generated by one of CNES' pyrotechnic test device and to predict the acceleration levels generated by this source on the structural model of a microsatellite. To do so, multiple intermediate steps had to be studied, beginning by the modeling of a simple sphere impacting a plate. The model complexity increased progressively to reach the modeling of a satellite vibrations induced by shock sources. In order to assess model predictability, all the performed tests were performed at various shock energies and compared to experimental results. This paper will present the results and comparisons with experimental SRS (Shock Response Spectrum) obtained starting with "simple" cases up to cases integrating complex structures and shock sources.

Designing shock absorbers for nuclear transport packages

Sean Duvall

Nuclear Transport Solutions

Abstract

Nuclear transport packages are required to withstand numerous tests before being granted a license to transport radioactive items. One of the requirements is that it withstands a 9m drop onto an unyielding surface. Nuclear Transport Solutions (NTS) are currently designing a large transport package which has limited external space for any shock absorbers, due to the limited rail gauge in the UK. This presentation takes us through the development process from hand calculations and LSDYNA analysis, to the current design which enables the 65Te package to drop from 9m with only 80mm of metallic shock absorber to absorb the impact energy.

Modeling net capture of an object in LS-DYNA

Aviner Shreiber, Lee-Hee Drory

Rafael Advanced Defense Systems

Abstract

Among other alternatives, one way of dealing with capturing a moving object, is to capture it in a hanged cage-like net. The capturing process should be reliable under various impact velocities and orientations of the body. In order to examine these multiple cases, a FEA model of the net and the body was created in LS-DYNA. This work is focused on the modeling, characterizing and calibrating of the net. In order to accurately represent the actual net's initial state and the capturing process, a few aspects should be considered: First, the beam-element-based model was constructed in MATLAB using a generative function, that allows easy generation of a multiple parameters dependent net, due to the need of flexibility in net dimensions, geometric properties, part allocation and the base unit cells of the net. Second, material properties of the hyper elastic unique net structure should be characterized by a series of tensile experiments followed by properly choosing LS-DYNA element and material formulation. Next, the modeled net should be folded in order to generate the initial net's state for the impact. Both folding and impact procedures require careful consideration and examination of different contact types and formulations, aerodynamic and mechanical forces, damping etc.

AI-based prediction of thermoforming, clamping and joining processes

Ingolf Lepenies
SCALE GmbH

On the Characterization and Calibration of Alloy Materials used in Additive Manufacturing Processes for Crashworthiness Applications

Vincent Suske[1], Aleksandar Bach[2], Michael Lutze[3], André Haufe[1]

[1]DYNAmore GmbH, Industriestrasse 2, D-70565 Stuttgart, Germany
[2]Ford-Werke GmbH, Henry-Ford-Str. 1, 70735 Köln, Germany
[3]AM Metals GmbH, Zoxy-Platz 1, 09633 Halsbrücke, Germany

Abstract

Thin-walled structures, filled with geometrically optimized lattice type structures to be used in crash management systems in the automotive industry have been proposed in various publications in recent years. Together with new additive production methods, namely selective laser melting (SLS) it is envisioned that their superior crashworthiness behavior will lead to lighter design. Especially for low volume vehicles these advantages might yield to broader application of the technology. This idea is followed in the public funded research and development project KI-LaSt.

In a first step, adequate alloys candidates that are producible within the SLS process and that show the required ductile fracture behavior need to be further developed. The present contribution discusses the various development steps of the alloy based on quick and simple testing after the time-consuming production. The main focus of the contribution is put on the full featured characterization and material parameter identification of the selected alloy. Furthermore, first structural simulations, that eventually will be used as basis or optimization runs will be presented.

AI-based prediction of the manufacturability of car body components in the CAD format

Ingolf Lepenies
SCALE GmbH

Mechanical characterization and automated macro-level modelling strategies of Li-ion Battery Cells

Jure Vinkovic[1], Simon Hudales[1], Martin Schwab[1], Harish Pothukuchi[1]

[1]4a engineering GmbH, Industriepark 1, 8772 Traboch, Austria

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.

This paper deals with the mechanical characterization and the macro level (homogenized) modelling of the jelly roll and the metal casing material. Static and dynamic mechanical characterization under different load cases (3-Point-Bending, Axial crush and Plane crush) were performed on two different cylindrical batteries – 18650 and 21700. The occurrence of the short circuit in the measurements was also monitored during the tests. Based on the experimental results, suitable material models within LS-Dyna are identified and parametrized using the software VALIMAT®. The workflow for the parameter identification is automated and standardized using the AUTOFIT function. The versatility of the VALIMAT® software solution will be demonstrated with the aid of user defined input decks to simulate the mechanical macroscopic battery response under different loading conditions.

Electric Vehicle and Battery Model with Thermal-Electrical-Mechanical Coupling

Chenxi Ling[1], Leyu Wang[1], Cing-Dao (Steve) Kan[1], Chi Yang[2]

[1] Center for Collision Safety and Analysis, George Mason University, Fairfax, Virginia, USA
[2] Department of Physics and Astronomy, George Mason University, Fairfax, Virginia, USA

Abstract

With increasing number of electric vehicles (EV) on public roadways, there is a need to develop a predictive and computationally efficient model to fully understand battery and electric vehicle crash safety. This paper presents a system solution to model the lithium-ion battery and EV crash performance by modeling and simulating the thermal runaway process of batteries with a thermal-mechanical-electrical coupled analysis. A representative cell model is first developed based on matching dimensions and the amount of material in a realistic battery cell. Each battery component is modeled separately with realistic material properties assigned to respective mechanical, electrical, and thermal material models. To improve computational efficiency, the electric circuit of the battery is modeled with the resistant solver and a user defined Randles Circuit Model is used to capture the circuit dynamic effect. The external short circuit test and punch test are used to calibrate the circuit dynamic response and mechanical property. It is demonstrated that the model is able to predict current density, OHM heating power and temperature distribution contours for each layer of battery component. The model also captures the local heat generation caused by coupling effect in an event that an active cell is punched. In the second part of this work, the battery model is integrated into a generic EV finite element model, which includes active battery cells connected in series with normal discharging current. A pole impact simulation is carried out to demonstrate the applicability of this model. In addition to predicting deformation and acceleration similar to mechanical-only crash analysis, this model also provides the heat generation and temperature distribution caused by impact loading and electric short circuit. This full vehicle with battery simulation model runs for 24 hours on a 32-core cluster for a crash event duration of 0.2s. This model demonstrates the feasibility of using thermal-electrical-mechanical coupling in EV crash simulation.

Abuse Characterization and Simulation of Battery Cells using a Layered Approach

David Poulard, Prakhar Amrute, Kevin Kong, Inaki Caldichoury, Vidyu Challa, Pierre L'Eplattenier

Ansys Inc.

Abstract

Safety is an important functional requirement in the development of large, energy-dense, lithium-ion (Li-ion) batteries used in electrified vehicles as they can catch fire if they have been damaged during car crashes. Simulations that predict the response of a Li-ion battery pack to abusive conditions can support analysis during the design phase and reduce the need for physical testing. A series of crush experiments on automotive grade Li-ion pouches cells have been performed by Ansys to obtain the numerical parameters needed for battery modelling. In a previous publication, benchmarks between the experimental and numerical results from LS-DYNA allowed the tune up of cell dependent parameters of the equivalent circuit models. The model used was a macroscopic model “batmac” that can mimic the complex electrochemistry happening in Li-ion batteries without having to model each layer that compose it (cathode, separator and anode), saving a significant amount of computation time. While it could reproduce well the experimental data in terms of voltage drop and temperature elevation, ANSYS would like to get a deeper understanding of battery modelling when all the layers of the cell has been represented. In this presentation, a refined battery cell model that represents each layer of the cell that was used during the experimental tests performed by Ansys will be presented. Such detailed model regarding the experimental results will be crucial in our understanding of the different local phenomenon that cannot be captured by the macroscopic model and will help improving the workflow that Ansys LS-DYNA offers for the battery simulations in full vehicle crash.

Modeling and calibration of a detailed micro-mechanical model for cylindrical batteries under large deformations

Skylar Sible[1], Nick Wagner[1], Nils Karajan[2], Wolfram Hahn[3], Julia Petro[4], Matthias Morak[5]

[1]Ansys, Inc.
[2] DYNAmore GmbH, an Ansys Company
[3] Hilti AG
[4] Polymer Competence Center Leoben GmbH
[5] 4a Engineering GmbH

1 Abstract

With the increasing utilization of Lithium-ion batteries in commercial products, the need grows for accurately simulating the failure and testing the safety measures of these batteries. Understanding the mechanical behavior of these cells is critical in predicting a cell failure or shortage resulting from physical damage, which could cause thermal runaway and other catastrophic consequences. It is proposed in this paper a methodology to characterize the mechanical behavior of a battery cell by combining models of the materials within the cell at microscopic level as well as the entire jellyroll structure at component level. A micro-mechanical model is utilized to describe the inter- and intra- layer mechanical responses within the single cell when subject to large deformation. A layered jellyroll mesh is created based on geometries obtained from CT-scan and other measurements, taking into consideration details such as active electrolyte layers, anode, cathode foils, and a separator. Material cards are calibrated using mechanical tests performed on deconstructed layer samples. This model shows promises in demonstrating the mechanical behavior of a battery cell and predicting any potential failure. Furthermore, it can be transformed into a homogenized material model at macroscopic level for any future Multiphysics abuse models.

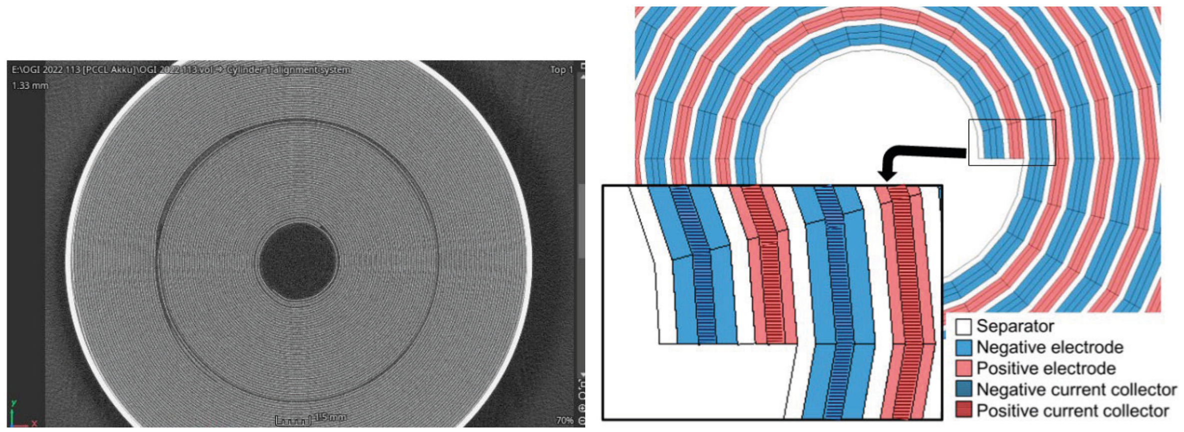


Fig.1: CT-scan of Internal structure of cylindrical Li-ion battery and the corresponding jellyroll mesh

Development of a Data-driven Surrogate Model for Scale-bridging in Battery Modelling Applications

Harshwardhan Dhumal[1], Tobias Aubel[1], David Koch[1], André Mielke[2], Karsten Keller[2]

[1]DYNAmore, an Ansys Company
[2]Institut für Statik und Dynamik der Luft- und Raumfahrtkonstruktionen (Stuttgart Universität)

Abstract

Multiscale computational modeling and simulations are required in many mechanical engineering applications. However, because of the enormous dimensionality of the solution space, solving for complicated multiscale systems remains computationally challenging. Machine learning (ML) has been continuously evolving as a viable alternative that can either accelerate, replace, or supplement conventional numerical methods. Recent research has shown that ML can solve various differential equations, such as a boundary value problem with an appropriate level of accuracy and computational effort. The current work focuses on the FE- simulation of the batteries with emphasis on modelling of the battery cells. It is often observed that it is a difficult and time-consuming process to account for the micro-structural response of the cells in full-scale simulations. Because of the computational inefficiency of incorporating detailed models in larger simulation models there is a high demand for the development of numerically less expensive models. In this contribution, we propose a multiscale strategy based on large data-sets and ML. Thereby a data-set of simulations is obtained at the micro-scale, by creating a detailed FE-model and simulating it along with the concept of Representative Volume Elements using the LS-DYNA software suite. After that, the data is used to train a ML-based surrogate model. Finally, this surrogate model can be linked back to LS-DYNA using a UMAT and can be imported in future for battery modelling and simulations.

A New Model Reduction Method for Vehicle Crash Simulation

Shinya Hayashi[1], Shinya Hiroi[1], Norio Shimizu[1]

[1]JSOL Corporation

1 Abstract

In this paper, a new model reduction method for vehicle crash simulation is presented, to aid in the early stages of automotive development.

When major design changes are required to satisfy product performance late in the development process, significant cost and time are required to implement them. This is a particular problem in automotive development which requires a large amount time and cost. To alleviate this, automotive manufacturers have adopted the concept of "front-loading" to identify problems early-on in the development process. “Front-loading” is defined as "the distribution of development costs or time in large proportions in the early stages of the design process”. The earlier issues in the production phase can be identified, the more efficiently and effectively development can be performed. Recent simulation technology has taken front-loading to a higher level, allowing for quicker decisions earlier in the vehicle design and concept phase. This allows designers and CAE engineers to arrive at better solutions in the earlier stages of product development.

Vehicle crash safety is one of the most important criteria in automotive design and incurs significant development cost. Simulation plays a major role in confirming and studying improvements in automotive crash safety performance, and contributes to a significant reduction in the number of physical crash tests. However, crash simulations themselves require expensive and lengthy analyses to faithfully reproduce the complex nonlinear behaviour of highly deformed vehicle body structures. In addition, these analyses are usually performed in the middle to late stages of design with detailed crash analysis models. An example in literature [1] has shown how vehicle body structures modelled using classical beam elements can be used to perform crash analysis at low computational cost in the early stage of design. However, the method presented requires dedicated material models and shows difficulty representing complex structural joints [2], such that the simplified model is quite different to the detailed crash model used in the mid to late stages of development.

JSOL Corporation has developed a new method for modelling automotive body structures to simulate crash analysis in the early stage of design that employs Hughes-Liu beam elements with arbitrary cross-sectional geometry. JSOL is working on a new modelling tool that can easily create these beams from existing detailed FEM/IGA shell structures or vice versa. This model reduction approach provides a seamless simulation workflow throughout the whole development process, enabling a more powerful "front-loading" approach to automotive development.

2 Literature

[1] L. Wu, X. Zhang, and C. Yang: “Research on Simplified Parametric Finite Element Model of Automobile Frontal Crash”, 6th International Conference on Computer-Aided Design, Manufacturing, Modeling and Simulation (CDMMS 2018)

[2] G. De Gaetano, D. Mundo, G. Vena, M. Kroiss and L. Cremers: “A study on vehicle body concept modelling: beam to joint connection and size optimization of beam-like structures”, ISMA Conference 2014

Parametric Projection-based Model Order Reduction for Crash

Mathias Lesjak[1,2], Prof. Dr.-Ing. habil. Fabian Duddeck[2],:

[1]BMW Group, Research and Innovation Center, Knorrstraße 147, 80788 Munich, Germany
[2]Technical University of Munich, TUM School of Engineering and Design, Arcisstraße 21, 80333 Munich, Germany

Abstract

Passive safety development of modern cars requires numerous simulations and hardware tests. These are necessary to investigate not only the behavior of the car at a specific test configuration but also to observe how deviations affect the crash design’s response. On the virtual development side, methods that investigate the system’s response to parameter variations are typically associated with many simulations. An approach to make the development process more efficient is Model Order Reduction (MOR). MOR enables multi-query applications by reducing the model’s complexity using already generated data. While different approaches for MOR exist, we decide for projection-based MOR as it still involves the problem’s physics. Using randomized algebra, it was shown that the required computations can be performed in reasonable time. However, this was only shown for reproductive examples. That is, the already known solution was reproduced by the ROM that was trained with the reference data. In contrast, parameters must be varied for the mentioned multi-query applications. Including parameter variations immensely increases the training data and the included variation. Using a single ROM is ineffective as the dimension of the ROM must be chosen large, which directly influences hyper-reduction. As this second reduction step is one of the two main mechanisms to achieve computational speedup in nonlinear MOR, other MOR approaches are required. We show the successful implementation and application of the local Reduced-Order Bases (ROB) approach for a crash box. The method can divide the whole training data into subregions, of which each region is of manageable size. The dimension of each local ROM is low, hence allowing for effective hyper reduction. In addition, we propose a hyper-reduction approach for path-dependent materials using a global reduced mesh with varying mesh weights. The presented results motivate the use of MOR and promote further research in the context of nonlinear solid dynamics.

From automatic event detection to automatic cause correlation

D. Borsotto, N. Abdelhady, C.-A. Thole

SIDACT GmbH

Abstract

Reaching and fulfilling several design and crash criteria during the development process is what makes the engineer adapt and redesign the simulation model over and over again. Ideally resulting in new simulation runs with in best case improved performance, matching the intention of the applied changes. For the more demanding case of unforeseen results which do not necessarily fit to the expectations of the actual changes, methods and a workflow are being introduced here, which allow to identify the root cause of this behavior. In a first instance every new simulation run is being added into an analysis database, which is continuously being used to compare new simulations against. Previous studies have already shown that this process can assist the engineer in automatically highlighting new behavior and pin pointing the engineer to the regions of interest. Rather than only highlighting the new behavior now a second phase is being triggered additionally. In this second phase the previously detected event is being isolated and analyzed against the gathered data of the development history. The analysis methods used are based up on the Principal Component Analysis, a reduced order modelling technique. This allows not only identifying structures in the data but also correlating deformation patterns against each other. Especially the latter one is of interest for an automated process, as it allows automatically detecting and suggesting possible root causes to the engineer. As an outcome of this process the engineer receives a list of correlating parts, so that he can focus on deriving a better engineering solution to achieve a deterministic behavior, rather than searching for the root cause of the event. To provide additional information about the type of cause, as for example failure or buckling, the identified parts are also forwarded to a classification prototype. This type of classification shall assist the engineer in deriving a possible design adaptation.

Full-field ROM generation with LS-OPT, LS-DYNA and Twin Builder

Nielen Stander[1], Anirban Basudhar[1], Valéry Morgenthaler[2], Christelle Grivot [2]

[1]Ansys LST [2]Ansys Digital Twin

1 Introduction

A methodology for creating fast surrogate models of LS-DYNA models is investigated. The method involves the integration of LS-OPT, LS-DYNA and Ansys Twin Builder. LS-OPT is used to generate and schedule multiple LS-DYNA simulations according to an experimental design. The dynamic field results are then gathered as input to create a Reduced Order Model (ROM) using Ansys Twin Builder. The paper presents some of the results obtained using this methodology.

2 LS-DYNA Histories and Fields using the LS-OPT Extractor

The need for creating inexpensive surrogate models arose from the application of Digital Image Correlation (DIC) to material calibration [1]. The calibration relies on the matching of full-field strains produced as LS-DYNA results to experimental results produced by Digital Image Correlation. LS-OPT can be used to minimize the field difference which results in a set of optimal material parameters. Since the test coupons are typically standardized and the large number of FE analyses required for the material parameter identification (using a fine mesh) can be time consuming, it was proposed to create a cheaper surrogate parametric model to replace the FE solver. For the same material model, the same surrogate model can then be used repeatedly across multiple test results. To create full-field models of strains or displacements, field extraction was added to the LS-OPT extractor functionality. Any full-field quantity can be specified by part for 3-D solids or shells. Stresses and strains are mapped to the FE nodes.

More recently the authors have also encountered a growing need for inexpensive surrogate models to replace full Finite Element analyses during the early stages of the vehicle design process. There is also a need for ROM components as substructures of the main FE models.

3 Ansys Twin Builder

Twin Builder uses Singular Value Decomposition (SVD) or Proper Orthogonal Decomposition (POD) in combination with advanced interpolation methods such as Kriging, Support Vector Regression and GARS to construct Reduced Order Models (ROM). The data required for building a full-field surrogate model can be 4-dimensional including time. Examples of such fields are dynamic displacements or strains.

4 LS-DYNA Example

Fig. 1 shows an example of the side impact of a vehicle B-pillar to which 7 design thickness variables and side impact velocity) have been assigned. The goal is to approximate the y-displacement field using LS-DYNA, LS-OPT and Twin Builder. Fig. 2 shows a representation (Twin Builder) of the reference FE model, the ROM model and the difference of a design in the proximity of the mean ROM error. Fig. 3 shows a plot of the maximum absolute error as a function of the number of training points used.

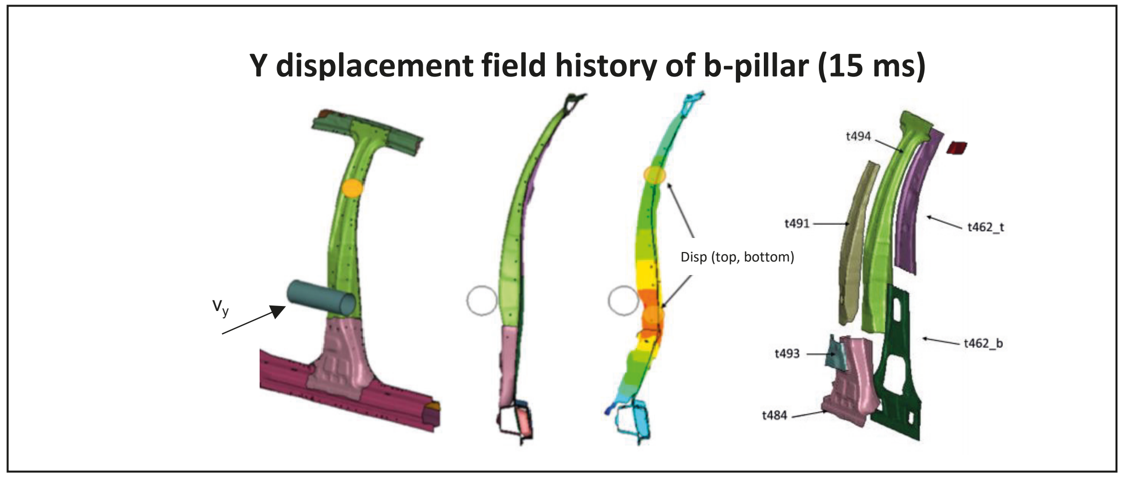


Fig.1: LS-DYNA FE model B-pillar intrusion. Y-displacement field history

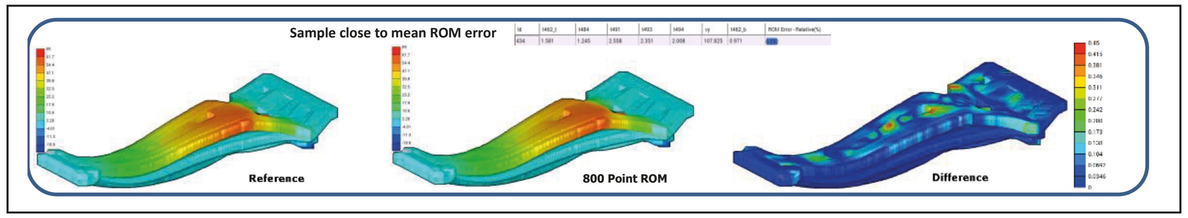


Fig.2: Reference FE model, ROM model and difference using a design in the proximity of the mean ROM error.

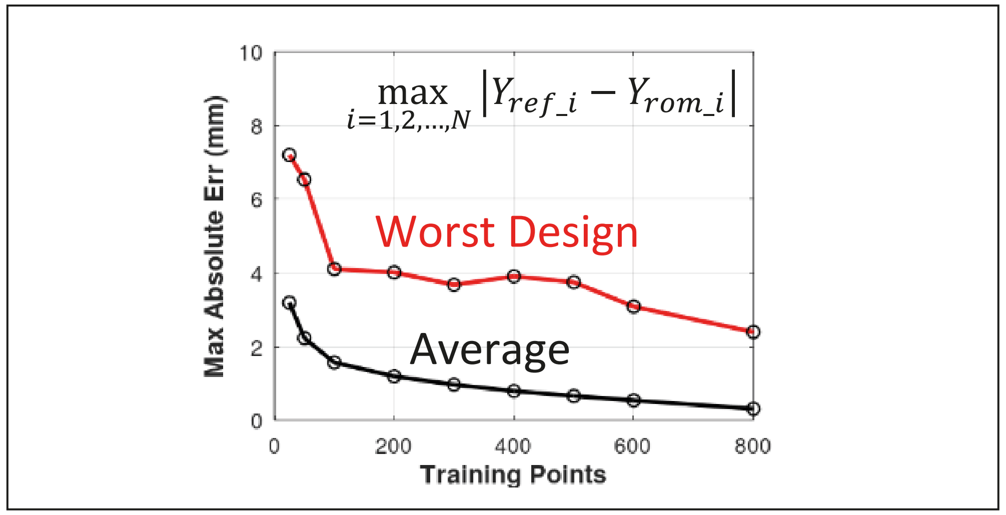


Fig.3: Maximum absolute error as a function of the number of training points used to construct the ROM.

5 Summary

A methodology for producing and evaluating ROM models of LS-DYNA non-linear dynamic FE analysis is demonstrated. The process uses LS-OPT to generate and run multiple simulations and export the results to Ansys Twin Builder. Twin Builder is then used to create a ROM model as an approximation of the full FE dynamic model. This example shows that reasonable displacement accuracy for a 7-variable 4-D model can be obtained using a few hundred simulations.

6 Literature

[1] Stander, N., Witowski, K., Ilg, C., Helbig, M., Koch, D. Application of Digital Image Correlation to Material Parameter Identification. Proceedings of the 12th World Congress on Structural and Multidisciplinary Optimization, Braunschweig, Germany, June 5-9, 2017.

[2] Kayvantash, K., Thiam, A., Ryckelynck, D., Ben Chaabane, S., Touzeau, J, Ravier, P. Model Reduction Techniques for LS-DYNA ALE and Crash Applications, Proceedings of the 10th LS-DYNA Conference 2015, Würzburg, Germany.

Reduced Order Model for enhanced EVAR Planning and navigation guidance

Monica Emendi[1,3*], Eirini Kardampiki[1,2*], Karen H. Støverud[2], Pierluigi di Giovanni[3], Sigrid K. Dahl[2], Aline Bel-Brunon[4], Victorien Prot[5], Marco E. Biancolini[1]

[1] Department of Industrial Engineering, University of Rome Tor Vergata, Italy
[2] Department of Health Research, SINTEF Digital, Norway
[3] HSL S.r.l, Trento, Italy
[4] LaMCoS CNRS UMR5259, INSA-Lyon, Villeurbanne F-69621, France
[5] Department of Structural Engineering, Faculty of Engineering, NTNU, Norway
*These authors equally contributed to the work

1 Introduction

Endovascular aneurysm repair (EVAR) procedure consists in the deployment of a stent-graft in the abdominal aorta to prevent blood flow in the aneurysm sac and thereby avoiding its rupture. It is a minimally invasive procedure: the insertion of a stiff guidewire straightens the iliac arteries to ease the passage of the stent-graft delivery system. This causes deformations of the aorta, leading to the need of acquiring intra-operative images with contrast agent to visualize the updated configuration of the vessel.
Towards the prediction of the deformations induced by the insertion of medical instruments inside the vessels, previous works [1] [2] have developed Finite-Element (FE) models to simulate the interaction between the aorto-iliac structure and the guidewires. However, the current bottleneck of the FE approach remains the relatively high computational time and exploitability by the clinicians.
To this aim, Reduced Order Models (ROMs) [3] [4] could be a valuable computational tool that can offer a new perspective on the clinicians' decision-making process, in compliance with the limited clinical time. This study proposes the build-up of a patient-specific pre and intraoperative interactive FE environment to predict the guidewire induced aortic deformation in function of some critical mechanical, morphological and clinical parameters.

2 Materials and Methods

2.1 FEM Set-Up

The FE analysis was carried out using the software Ansys LS-DYNA Explicit. The abdominal aorta was segmented from pre-operative CT images of the patient. The aortic wall was meshed with triangular shell elements and modelled as a linear elastic material. For the guidewire, beam elements were chosen assuming linear elasticity. The floppy tip of the guidewire was modelled with a gradually decreasing elastic modulus, ranging from 1 to 50 GPa [1]. A rigid introducer was included to avoid undesired movements of the wire outside the vessel.

2.2 Reduced order model Set-Up

Seven parameters were considered for this study: the elasticity of the aorta, the stiffness of the guidewire, two guidewire's insertion angles (on the sagittal and the frontal plane), the suprarenal and infrarenal neck angle, the tortuosity of the iliac artery. The last three morphological parameters were implemented via radial basis functions. The ROM was set based on a Design of Experiments scheme (Optimal Space Filling algorithm) with 300 samples. The obtained scenarios were evaluated and used to feed the ROM generation. The ROM build-up was performed on ANSYS Twin Builder, using the least number of scenarios that guarantee the least accuracy reduction.

3 Results

The calculation of the Digital Twin's snapshots took 125 CPU hours. The prediction error of the aortic deformation ROM was less than 6% or 0.3mm for all considered validation cases. This accuracy was considered sufficient for the clinical application. As depicted in Fig.1, the aortic wall stiffness and the guidewire's angle of insertion proved to significantly affect the motion of the abdominal aortic wall.

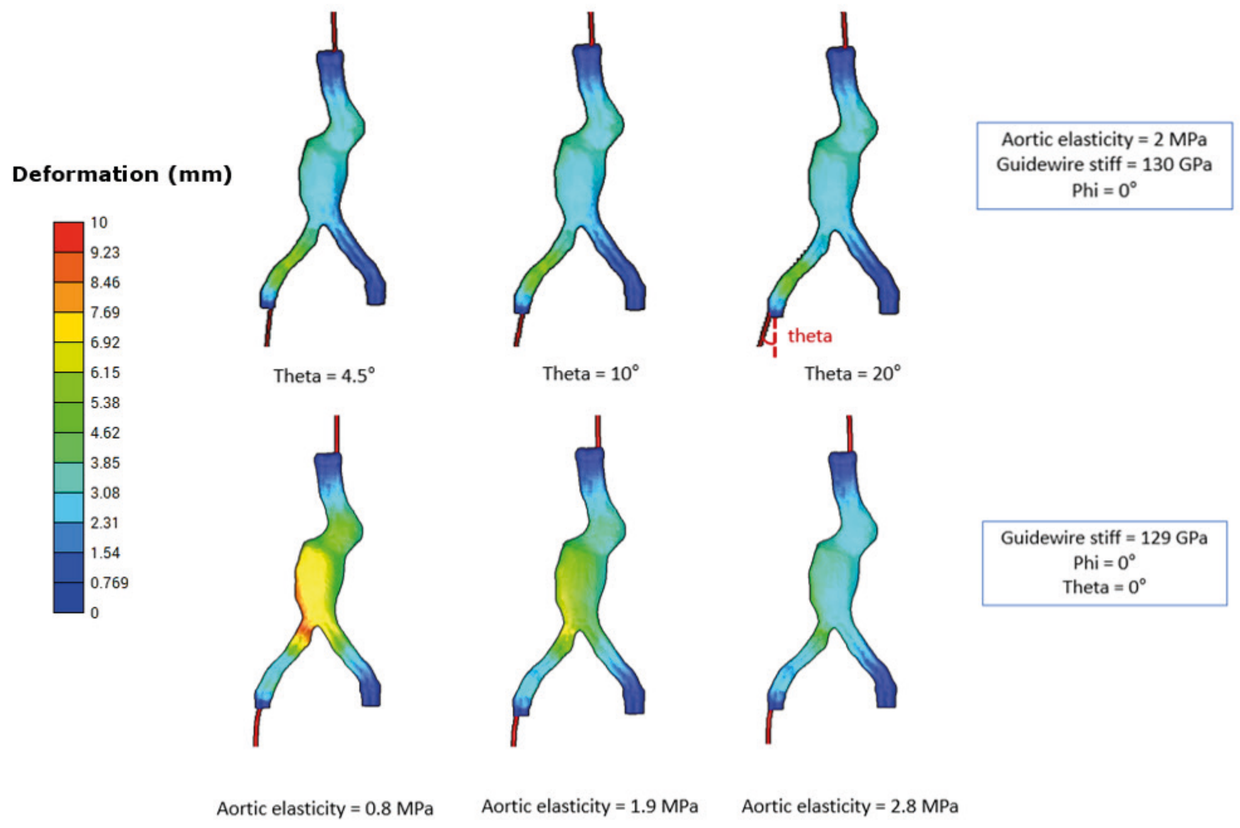


Fig.1: ROM deformation prediction in function of theta insertion angle (upper panel) and aortic elasticity (lower panel)

4 Discussion

A semi-automated Medical Digital Twin (MDT) pipeline to enhance the clinical workflow was successfully developed. The evaluation of the abdominal wall motion in function of the patient-specific aortic elasticity, the guidewire's stiffness and the clinicians' insertion method is satisfactory. The preliminary results of the ROM show that it is a promising surrogate model for FE simulations, despite its limited application to one patient's anatomy to date.

5 Summary

Starting from the high-fidelity FE simulation of the guidewire's insertion in a patient-specific vessel, we obtained a parametric ROM that allows the real-time calculation and visualization of the aortic deformations.

6 Literature

[1] Gindre, J. et al.: " Finite element simulation of the insertion of guidewires during an EVAR procedure: example of a complex patient case, a first step toward patient-specific parameterized models ", Int. J. Numer. Meth. Biomed. Engng., 31, 2015
[2] Mohammadi, H. et al.: "A numerical preoperative planning model to predict arterial deformations in endovascular aortic aneurysm repair", Annals of biomedical engineering, 46, 2018, 2148–2161
[3] Vega, J.M. et al.: "Higher Order Dynamic Mode Decomposition and Its Applications", Academic Press: Cambridge, 2021,1–28.

[4] Kardampiki, E. et al: "The Hemodynamic Effect of Modified Blalock–Taussig Shunt Morphologies: A Computational Analysis Based on Reduced Order Modeling", Electronics MDPI, 11(13), 2022

7 Acknowledgements

This work is funded by the European Union’s Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No. 859836, MeDiTATe: “The Medical Digital Twin for Aneurysm Prevention and Treatment”.

Simulating the hot press processing of structural thermoplastic foams

Stefano Cassola[1], Miro Duhovic[1], Maximilian Salmins[1], Peter Mitschang[1]

[1]Leibniz-Institut für Verbundwerkstoffe GmbH

Abstract

Thermoplastic foams allow the manufacture of lightweight parts with good thermal and acoustic insulation properties, particularly suited for aircraft interior and cabins structures. Such foams can be combined with skin layers of organic sheet materials (e.g. glass fiber (GF) polycarbonate (PC)) forming sandwich structures, enhancing the mechanical properties, but which unfortunately do not fulfil strict FST (Fire, Smoke and Toxicity) standards. An alternative approach uses the foam itself to create an integrated sandwich structure of an unmodified core and two skins of high density from the same material. In this work, structural foams were manufactured using a closed-cell polyethersulfone (PESU) foam. The thermoplastic foam was transformed into structural foams using a newly developed hot press process, which does not change the part weight. A shell element model was developed that allows the simulation of the hot press process for thermoplastic foams under non-isothermal conditions using LS-DYNA standard material models. The objective of the simulation model was to predict the final foam core and skin thicknesses that result from a hot press process to subsequently perform mechanical property (bending stiffness) calculations. During the hot press process, the foams are subject to a compressive force at elevated temperatures of up to 240°C. For the creation of the material model, it is therefore necessary to determine the temperature dependent compressive response by performing isothermal compression tests for PESU foams. The test were conducted at room and four elevated temperatures close to the glass transition temperature of PESU (T_g = 222°C). The final model was evaluated simulating the force-controlled, non-isothermal hot press process. Two different process conditions were simulated using a target pressing force of 500 N or 1000 N and a temperature of 230°C. The results of the simulations show that the usage of this shell model approach is a viable option to simulate the temperature dependent compression of foams using standard LS-DYNA material models. However, the accurate prediction of final foam core and skin thicknesses resulting from the force-controlled hot pressing process remains a challenge due to latencies in the process control that are currently not implemented in the simulation model.

Modelling structural foams with VALIMAT® under mixed shear-compressive loading in LS-DYNA

Benjamin Hirschmann[1], Christoph Schober[1],

[1]4a engineering GmbH, Industriepark 1, 8772 Traboch, Austria

4a engineering GmbH developed the software solution VALIMAT®, which helps the user to calibrate material models in a wide variety of solvers including LS-DYNA to measurement data with a reverse engineering process. For some material models automated workflows were defined[1)]. In this paper, this AUTOFIT function is extended to generate validated material models for structural foams in an automated process.

The conventional material card generation approaches generally utilize static and dynamic compression tests to characterize the mechanical behavior of foams. The drawback of this approach is that such a conventional material card would be able to predict the deformation mechanisms occurring in foam structural parts in a limited manner. This includes mainly load cases where the component is under a compressive load and the description of a tensile side in the load curve enables for predictions in tension / bending load cases.

In this paper, some of these conventional material models in LS-DYNA: **`*MAT_163`** & **`*MAT_083`** and their limitations with respect to the prediction of a mixed shear-compressive loading are discussed in detail. An alternative approach using a **`*MAT_SAMP-1`** coupled with a damage curve including compensation is proposed as a possible solution to deal with such scenarios.

References

[1] Hirschmann, B et. al: "Automated material model generation in VALIMAT – AUTOFIT & AUTOFAILUREFIT", 16. LS-DYNA Forum 2022, 11-12 October 2022, Bamberg

Simulation of thermoplastic 3D-printed parts for crash applications

Mathieu Vinot[1], Lars Bühler[1], Tobias Behling[1], Martin Holzapfel[1], Nathalie Toso[1]

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

1 Abstract

Keywords: additive manufacturing, thermoplastic material, numerical simulation, cohesive zone model

The rapid development of additive manufacturing techniques in aeronautic and automotive industry opens new possibilities in the design of metallic or composite parts compared to traditional subtractive processes. In particular, 3D printing allows the design of complex parts with a high lightweight potential through optimal use of material along the load paths. In the composite field, various printing techniques emerged in the last decade such as Selective Laser Sintering (SLS) or Fused Deposition Modelling (FDM) [1]. On the downside, 3D printing is confronted to the large influence of process parameters on the geometrical and optical quality as well as on the mechanical properties of the manufactured structures [2]. Moreover, simulation techniques with finite-element methods are still at their very beginning and improvements should be achieved to predict structural performances in crash applications.

Fig. 1: SEAM manufacturing system from DLR

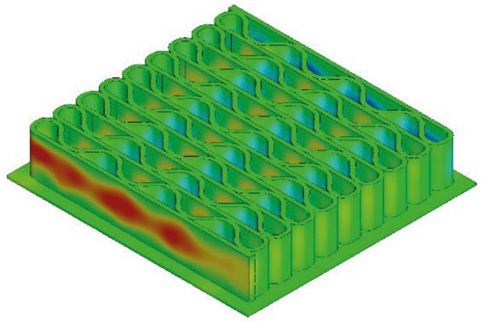


Fig. 2: Deformation of a 3D-printed structure under compressive loading

This paper focuses on the characterization and simulation of structures manufactured with the Screw Extrusion Additive Manufacturing (SEAM) technique presented in Figure 1. The material investigated is a combination of PA6 resin reinforced with 30% carbon fibres in weight. The mechanical behaviour under various loadings (tension, compression, shear) is first investigated in different material directions relative to the building direction with help of coupon specimens extracted from printed structures. The obtained properties are then fed in orthotropic and anisotropic material models **`*MAT058_LAMINATED_COMPOSITE_FABRIC`** and **`*MAT157_ANISOTROPIC_ELASTIC_PLASTIC`** coupled to **`*MAT_ADD_GENERALIZED_DAMAGE`** and the respective performances studied by reproducing numerically the experimental tests. This contribution illustrates potential numerical solutions to recreate the mechanical behaviour between the printed layers and gives an insight into their advantages and drawbacks. Finally, an application of the suggested approach to generic honeycomb structures is presented and the gaps in simulation technique are analysed.

2 References

[1] A. Jandyal, I. Chaturvedi, I. Wazir and M. I. U. H. A. Raina, “3D printing – A review of processes, materials and applications in industry 4.0,” Sustainable Operations and Computers Vol. 3, pp. 33-42, 7 October 2021.

[2] M. Pivar, D. Gregor-Svetec and D. Muck, “Effect of Printing Process Parameters on the Shape Transformation Capability of 3D Printed Structures,” Polymers Vol. 14, 29 December 2021.

Computational Modeling of the TPU Shock Absorber using Ls-Dyna

Ahmet Mete SABAH, Buğra BALABAN

TEMPA Engineering & Commerce Ltd.

Abstract

In this study, various analysis and test activities are presented, which were carried out for the development of shock-mitigating floor mats. These impact-absorbing floor mats are produced from hyperelastic materials and are designed to absorb high-amplitude, short-duration shock loads in defense or civil applications. Throughout the study, the ***MAT027** model is examined for its suitability in modeling hyperelastic materials in both static and dynamic analyses, conducted using the non-linear finite element code LS-DYNA®. The ***MAT027** model accurately describes the behavior of hyperelastic materials and is often preferred for this type of material. To correctly apply this model, specific parameters pertaining to the hyperelastic material must be determined. This study primarily focuses on the determination of these material parameters through tests conducted on samples made of Thermoplastic Polyurethane (TPU) material. In the initial phase of the study, a literature review concerning TPU material was conducted, and material parameters were obtained using the test data presented in this study. The material parameters were then optimized using LS-Opt® to achieve the best material behavior. Following the determination of material parameters, dynamic simulations were performed using LS-DYNA®, and the simulation results were subsequently compared with experimental data. With the material parameters in hand, the second phase of the study involved the design of cellular structures with various geometric shapes. The force-displacement graphs of these newly designed shapes were analyzed through dynamic analyses.

Keywords: Thermoplastic Polyurethane, Hyperelastic materials, Cellular solids, Shock absorber

Optimized light weight structures design using MAT187 card for improved accuracy

Vinod Kavalakkat[1], Somasekhar Bobba[2], Vamsy Godthi[1]

[1]High Performance Plastics India Private Limited,EOU
[2]SABIC Specialties, USA

Abstract

To design light weight structures efficiently and to accurately predict its performance using computer aided engineering (CAE) simulation has been a challenge for many automotive OEM's. The designs are not able to live up to the expectation in real-world tests as the CAE predictions are not always accurate. The accuracy of the predictions primarily comes from the material card that has been used and more importantly from the data that has been used to generate the material card. In event of lack of data most engineers tend to use data available in public databases which are often incomplete and/or inaccurate. Apart from material data, the inclusion and prediction of material failure sometimes plays a key role in enabling accurate predictions and CAE to physical test co-relation. The current work aims to bridge this gap and show a use case scenario. The work done in this paper explores the use case of a MAT 187 card to accurately to predict structural performance of an automotive component, namely the B-pillar. The study uses the IIHS side impact protocol to evaluate the material model and its prediction capability. A generic vehicle FEM model is used for the evaluation. A lightweight metal plastic hybrid (MPH) B-pillar structure is designed to replace the incumbent metal B-pillar with an aim to reduce weight of the vehicle. A comparison is also done against a simpler material model MAT24, which is primarily used to predict behaviour of metallic structures. The paper articulates the difference in prediction result that can be achieved by using the accurate material model.

Parametric optimization of cellular materials through LS-OPT

Alessandro Giustina[1], Ivan Colamartino[2], Paolo Franzosi[1], Marco Anghileri[1], Marco Virginio Boniardi[2]

[1]Department of Aerospace Science and Technology, Politecnico di Milano,
Via La Masa, 34, Milano, 20156, Italy
[2]Department of Mechanical Engineering, Politecnico di Milano, Via La Masa, 1, Milano, 20156, Italy

Abstract

Recent advancements in additive manufacturing technologies and the effort for researching lightweight solutions in energy absorption fields or for specific structural properties led to an arising interest in the development of cellular materials. One of the most valuable and unique aspects of these types of metamaterials is the possibility of customizing the geometrical parameters of the cells responding to different base materials, loading conditions, and performance requirements. However, the topology optimization of cellular materials is a non-trivial problem, especially considering the complex dynamics during impact for energy-absorbing applications. Not many analytical tools exist for the prediction of their performance and often they are based on weak assumptions on their behaviour. LS-TaSC, the topology and shape computation tool included in LS-DYNA suite, does not allow to create and control cellular structures, with the repetition of base units of cells. On the other hand, LS-OPT has the capability of setting up an optimization problem interfacing with user-defined programs in order to create the input files for LS-DYNA solver. The aim of our study is to develop a custom workflow for the optimization of cellular materials exploiting the LS-OPT interface, for any given parameters, even if not related to any variables specified in LS-DYNA keyword cards. In this context, an external subroutine was implemented in python environment, allowing to generate and mesh cellular geometries, one for each set of design point parameters, and performing optimization tasks on the performance indexes, given as output responses of the finite element analyses. Three relevant cellular geometries are considered as case studies: Stochastic-Voronoi, Negative Stiffness Honeycomb, and auxetic lattice structure, using as independent variables parameters both related to the geometry and to the base material of the cells. This novel process could become a useful tool for the design of cellular materials for several applications, allowing also to spot interesting statistical performance trends.

Numerical Structural Design and Optimization of Free-Form Hydrogen Vessels in the Context of Metal-Organic Frameworks

Markus Bellmann[1], Ruben Krischler[1], Ruben Czichos[1], Peter Middendorf[1]

[1]Institute of Aircraft Design, University of Stuttgart, Stuttgart, Germany

Abstract

Development regarding storage solutions for hydrogen is crucial to enable its widespread adoption as a sustainable energy carrier especially in the mobility and transportation sector. The application of Metal-Organic Frameworks in carbon fiber composite wound pressure vessels leads to a reduction in operating pressures and allow both a cost and CO2 footprint reduction by enabling glass fiber as a valid material choice and the exploration of free form tank designs in order to better utilize challenging design spaces in automotive vehicles. This study explores the capabilities and limitations of these tank designs using numerical multi stage optimization in LS-OPT in conjunction with LS-DYNA, BETA CAE, MATLAB and Python for the fully automated, detailed optimization of the geometry and laminate of these tanks. Encountered challenges regarding the automated creation and partitioning of design spaces and ideas for computational effective optimization by reduction of design variables are discussed. Furthermore, optimization strategies for the sizing of the composite laminate for different geometries and in compliance with required load cases are explored. Constraints by the winding manufacturing process are addressed by the development of different numerical methods for the calculation of geodesic winding paths. Some applied simplifications and methods are discussed and potential solutions provided. The results of this research confirm the functionality and modularity of the created software tool and show potential for improved design space usage for free from hydrogen tanks especially for geometrically complex design spaces.

Structural Optimization with the Incremental Equivalent Static Load Method for Nonlinear Dynamic Responses

Hong Dong[1], Brian Watson[1], Juan Pablo Leiva[1]

[1] Vanderplaats Research & Development, Inc. DBA OmniQuest

1 Abstract

This paper presents an efficient approach for optimizing structures under dynamic impact loading conditions. We introduce an improved method called the Incremental Equivalent Static Load Method that enhances the accuracy of the original ESL method. In the original ESL method, equivalent static loads are computed based on the initial geometry and nonlinear displacement results from a nonlinear analysis software. With the Incremental ESL method, we update the stiffness matrix at selected time steps using deformations from a base time step. This enables us to compute and apply equivalent static loads based on incremental displacements for ESL loadcases, resulting in a more precise capture of geometric and material nonlinearity.

The Incremental ESL method demonstrates superior performance by producing better approximations of structural responses and showcasing faster convergence for highly nonlinear analysis problems. To facilitate its practical implementation, we have incorporated the Incremental Equivalent Static Load Method into the OmniQuest GENESIS software, which seamlessly integrates with the existing OmniQuest ESLDYNA software and the ANSYS LS-DYNA software.

To illustrate the effectiveness of the approach, we present several examples of optimization results. These examples use the LS-DYNA software and include the sizing and topometry design of a sill and floor structure under pole crash conditions, topology optimization of a rocker beam cross-section subjected to pole crash scenarios, and topology optimization of a vehicle body experiencing multiple dynamic loading conditions. Through these examples, we demonstrate the ability of the Incremental Equivalent Static method in improving structural performance and optimizing designs under complex dynamic loading conditions.

The method discussed here shares similarities with another improved ESL method named difference-based ESL (DiESL) method, which also uses displacement steps to perform optimization. However, unlike the DiESL method that modifies FE meshes, our implementation is done inside the structural optimization program, GENESIS, and it does not require using multiple models, making it more practical and easier to use.

Overall, this paper demonstrates the significance and benefits of the Incremental Equivalent Static Load Method for structural optimization in nonlinear dynamic responses.

***KEYWORDS**

Equivalent Static Load Method, Incremental ESL Method, topology optimization, sizing optimization and topometry optimization

From topology optimization to 3D geometryfor a BiW subjected to multiple, crash and static, load cases

Kaloudis Alexandros[1], Malotka Alexander[2], Bikos Georgios[3], Nikoglou Nikolaos[3]

[1]BETA CAE Systems International AG
[2]BETA CAE Deutschland GmbH
[3]BETA CAE Systems SA

Abstract

The objective of this presentation is to briefly describe the distinct steps followed during a process regarding the structural optimization of a BiW starting from a given design space. The load cases under consideration are multiple, both crash and static, and the topology optimization calculations are performed simultaneously for all of them by one solver. The results of these calculations are interpreted and transformed into actual 3D geometry based on parametrized cross-sections using an automated process. Their parametrization enables the automatic sizing of them both during the original geometry creation and during a parametric optimization that could take place at a next step, for further improvement of the first interpretation.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Worst-Case Topology Optimization

Imtiaz Gandikota, Willem Roux, Guilian Yi

ANSYS Inc.

Abstract

For topology optimization to be useful in industrial design it must be able to meet general design criteria such as the design code requirements. An example of meeting these essential design code requirements is that of the design of a vehicle hood, for which one must consider the impact of a pedestrian's head against the hood, various static loads, constraints on the natural frequencies, and the minimum weight of the hood. Specifically difficult to handle are the multiple head impact locations specified in the Euro NCAP pedestrian testing protocol, because of the high computational cost. This paper accordingly shows how to conduct the worst-case design of the hood for multiple head impact locations, considering both the head injury criterion and the deflection of the hood.

Coil Winding Simulations of Electrical Machines

Stavroula Stefanatou[1], Johannes Heydenreich[1]

[1]Brose Fahrzeugteile SE & Co. Kommanditgesellschaft, Würzburg

Abstract

The winding of coils on stator teeth is a central process in the manufacture of electrical machines. The quality of the windings and the associated copper fill factor are important factors for the efficiency of electrical motors. While the copper winding process was manual labor some years ago, this process has meanwhile been completely automated and has been taken over by machines and robots. What is still left for manual labor is the setup of the machines for series production. This can be quite time consuming and costly. In order to support this setup process, fully understand it in all its details and speed it up, BROSE has used LS-Dyna to develop simulation models for the setup of new coil windings. These models meanwhile show a good balance between simulation speed and accuracy and comparisons with winding samples from the sample shop and production machines show good matching. This presentation shows simulation models, results, comparisons with measurement and outlines alternative options for model building with their advantages and disadvantages.

Prepreg forming, curing and structural analysis for an aero engine component

Dennis Wilhelmsson[1], Jesper Eman[2], Vivekendra Singh[2], Anders Bernhardsson[3], Mats Landervik[3]

[1]GKN Aerospace Sweden AB
[2]RISE Research Institutes of Sweden
[3]DYNAmore Nordic

Carbon fibre composites have the potential of reducing weight and thereby the carbon footprint of an aero engine component due to the high strength and stiffness of the material relative to its weight. In this paper, a process simulation chain, consisting of forming, curing and structural simulations, is proposed. The demonstrator is an outlet guide vain (OGV) which is part of an electric fan aero engine. The objective of the simulations is to facilitate the optimization of the guide vain design and production process early in the design phase by considering the properties resulting from the process in the predictions of the part performance in the use phase. Concretely, forming and curing simulations are used to get a better understanding of the process and predict fibre angles and shape distortions to provide improved accuracy in the subsequent structural simulations.

The OGV is produced by a novel double vacuum forming (DVF) process. In the process, a full stack of prepreg plies is placed on a rubber membrane. The stack is then formed and consolidated in vacuum at a material specific temperature after which it can be demoulded and moved to the curing tool. For the forming simulation, the single plies are characterized in terms of bending and shear behaviour to be represented by shell elements using the *MAT_249 material model. Furthermore, the interply friction is tested at different pressures, sliding velocities, temperatures and sliding directions relative to the fibre directions in the neighbouring uni-directional (UD) plies. In the simulation, these dependencies are included in the Mortar contact (*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_ORTHO_FRICTION) [1] describing the ply interaction. From the forming simulation, the resulting mesh and fibre orientations are exported. To have a 3D representation of the plies in the subsequent curing and structural simulations, the shell mesh resulting from forming is inflated in the thickness direction. The ENVYO software [2] is used to transfer the material history variables containing the fibre orientation from the shell forming mesh to material directions in *ELEMENT_SOLID_ORTHO in the solid mesh for curing analysis. To analyse the curing, a viscoelastic material model is used, [3].This material model has been implemented as a user defined subroutine and computes cure induced shape distortions and residual stresses together with cure and temperature evolution. Resulting nodal coordinates and residual stresses are subsequently transferred to the structural simulations.

For the structural simulations, three mechanical load cases have been identified to simulate the ultimate failure of the OGV. To do that, a user defined material model is used [4]. It is a 3D homogenized model with large deformation formulation to account for damage growth in a unidirectional (UD) composite ply. The fibre kinking response is accounted by the fibre kinking theory formulated in a finite deformation framework. The nonlinear shear response of the ply is pressure dependent and is formulated by combining damage and friction on the fracture plane.

To summarize, advanced material models within three different disciplines has in this project been joined together to create a new method for manufacturing simulations of composites and the subsequent consideration of these in structural assessment.

An illustration of the simulation chain is shown in Figure 1-1.

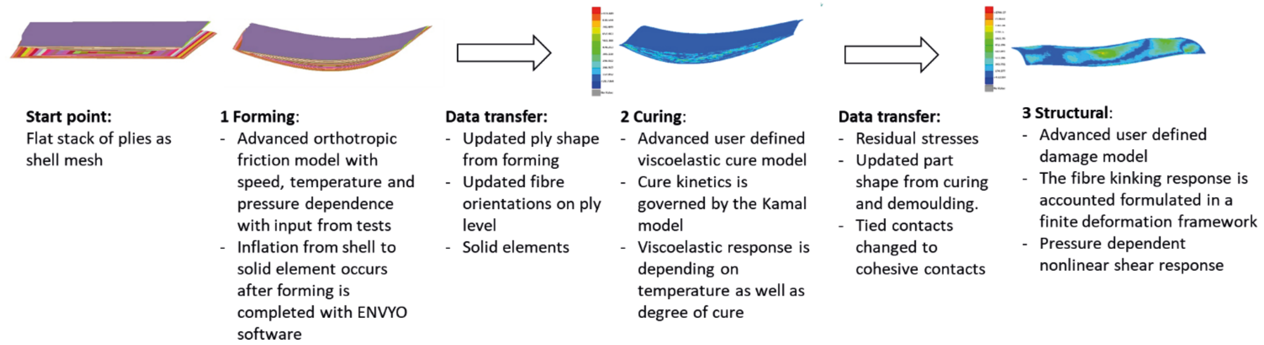


Figure 1-1: Sequential simulations of a composite vane in LS-Dyna.

1 Literature

[1] Kumaraswamy, S., Dutta, A., Bernhardsson, A., Landervik, M., Åkermo, M.: “On Interply Friction in Prepreg Forming Simulations. 13th European LS-DYNA Conference, 2021.

[2] Liebold, C., Haufe, A.: “Process2product Simulation: Closing Incompatibilities in Constitutive Modeling and Spatial Discretization with envyo®“, 15th International LS-DYNA Conference, 2018.

[3] Saseendran, S.: "Effect of Degree of Cure on Viscoelastic Behavior of Polymers and their Composites”, PhD dissertation, 2017.

[4] Costa, S., Zrida, H., Olsson, R., Herráez, M., Östlund, R.: “A unified physically-based finite deformation model for damage growth in composites”, 161, 107103, Composite: Part A, 2022.

Process simulation in LSDYNA from the viewpoint of a materials supplier: towards an integrated approach for performance and process

Christos Derdas, Michael Klotz, Raoul Abas

Henkel AG & Co KgaA

Abstract

Process induced deformations (like readthrough) are an issue always considered during product development for adhesives and reinforcements that are used on automotive substrates that are visible. The electrification and lighweighting megatrends drive significant visible panel thickness reductions and at the same time for the increased use of adhesives and localized reinforcements which essentially makes understanding the curing process and its effects vital for getting it “right the first time”. In order to achieve this understanding and to serve customer’s requests for process simulation data cards , Henkel AG & Co KgaA has been for the past years developing significant knowhow on process simulation using *MAT_277 and *MAT_307. The present paper aims to present Henkel AG &Co KgaA’s capabilities for process simulation of various reinforcement and adhesive products. A public domain finite element model of an automotive door is used to perform initially a lightweighting study. Then the effects of curing of the products used is investigated, along with the relevant measures to minimize process effects on the visible panel of the door. The study encompasses the description of the methods, the numerical stability studies conducted, the process effects as derived by the FE model of the door after curing of the adhesive, along with the measures -either application wise or product characteristics wise – that can be taken to reduce process-induced effects. Finally, conclusions are drawn and the mechanism leading to such deformations is described and commented on.

Die Attach Process using Adaptive ISPG in LS-Dyna

Ayush Kumar[1], Vivek Pawar[2]

[1]Ansys Germany GmbH
[2]NXP Semiconductors

Abstract

The die-attach process is a crucial step in electronic packaging, where semiconductor chips (dies) are securely bonded onto substrates (e.g., lead frames or printed circuit boards). The process typically involves applying an adhesive or solder material to join the die and substrate. It ensures electrical connectivity, dissipates heat, and protects the delicate semiconductor components. Precise die-attach (DA) techniques are vital to guaranteeing the reliability and performance of electronic devices, as improper bonding can lead to connection failures and reduced overall functionality of the packaged components. A series of experiments are required to obtain full coverage of the DA material while optimizing bleed out. To reduce the experimental trials, a 3D model is developed and simulated using Adaptive **ISPG (Incompressible Smooth Particle Galerkin)** technology in LS-Dyna. An Ansys Mechanical ISPG Plugin (ACT Extension) was developed to leverage Mechanical’s user-friendly model-setup and solving environment for LS-Dyna solver. During the simulation displacement-based boundary condition is used to conduct an analysis of reaction force with varying Die velocities. A non-Newtonian fluid model (**Carreau Model**) is used for the epoxy material where the viscosity varies with the current shear rate which impacts the reaction force on the Die. The simulation results are validated against experimental data at NXP. The Validated model is further used to investigate the effect of different dispensing volume, dispensing pattern and die dimensions on coverage of die attach material in a subsequent study. The result of this parametric study can provide more insight into the die attach process and it can be used as an initial process parameter to reduce design of experiments.

Recent Development of Multiscale Methods in LS-DYNA

Wei Hu, Xiaofei Pan, Bo Ren, Yong Guo, Jingxiao Xu, Dandan Lyu, Haoyan Wei,
Yuxi Xai, Michael Su, C.T. Wu

Ansys Inc.

1 Abstract

This talk provides a review of recent LS-DYNA developments for multiscale modeling. The goal of our development is to integrate manufacturing information, which may include deformed geometry/warpage, location-dependent material properties, microstructures, defects, residual stress, and other sub-scale details, with the dynamic structural analysis across multiple length scales, and to have less need of experimental tests in product design cycle time. Two multiscale methods are introduced in this talk.

Deep Material Network (DMN) [1], released in R14, is a data-driven method for bridging location-dependent micro-meso scale material properties via physics-based machine learning algorithm for the dynamic structural analysis of injection molded short-fiber reinforced composites. The on-going DMN development for polycrystal metals will also be mentioned.

Two-scale Co-simulation [2], first released in R13, is a joint simulation of coupled standalone solvers bridging meso-macro scale via various coupling algorithms (contact, constraint, and immersion) when fine mesh parts or joints (rivets, bolts, spot welds, solder balls, etc.) are presented in the sub-scale level of a system model. A synchronized time stepping scheme was developed to speed up and stabilize the co-simulation in applications like impact, drop test and crashworthiness.

Finally, an overview on three multi-physics manufacturing simulation methods used in multiscale modeling are also provided. They include (a) Incompressible Smoothed Particle Galerkin (ISPG) method [3] for free surface reflow soldering, adhesive, coating, and capillary flow simulations, (b) Smoothed Particle Galerkin (SPG) method [4] for explicit dynamic analysis of material fabrication processes involving severe deformation and material removal such machining, blanking, tapping, screwing, riveting, drilling, and grinding. (c) Adaptive EFG/FEM method for simulating fabrication processes like forging and rolling that involve severe deformation but without material removal.

2 References

[1] H. Wei, C.T. Wu, W. Hu, T.H. Su, H. Oura, M. Nishi, T. Naito, S. Chung, L. Shen, (2023) LS-DYNA machine learning-based multiscale method for nonlinear modeling of short-fiber-reinforced composites, J Engrg Mech, 149 (3), 04023003.

[2] W. Hu, X. Pan, D. Lyu, A. Srivastava, S. Shah, C.T. Wu (2020) A two-scale approach for the drop shock simulation of a printed circuit board package considering reflowed solder ball geometries, IJ Multi Comp Engrg, 18 (4), 455-476.

[3] X. Pan, C.T. Wu, W. Hu, (2020) A semi-implicit stabilized Particle Galerkin method for incompressible free surface flow simulations, IJ Num Meth Engrg, 121 (17), 3979-4002.

[4] X. Pan, C.T. Wu, W. Hu, Y. Wu (2019) A momentum-consistent stabilization algorithm for Lagrangian particle methods in the thermo-mechanical friction drilling analysis, Comp Mech, 64 (2), 451-465.

Dynamic Explicit SPH Simulation and Material Characterization of Road Tankers using LS-DYNA

C. Robb, G. Abdelal[1], P. Mckeefry[2], C. Quinn[2]

[1]Queen's University Belfast
[2]Crossland Tankers

Abstract

Crossland Tankers is a significant manufacturer of bulk-load road tankers from Northern Ireland. 30,000 litres of liquid are carried over long distances and varying road conditions. The effect of sloshing within the tank can significantly impact the driveability and lifespan of the tanker. As part of this project with Crossland Tankers, we will develop a model using LS-DYNA to investigate the applications as part of the design process.
Multiphysics simulations are now more emphasised than physical tests due to environmental, financial, and time requirements in the competitive manufacturing industry. Thus, developing models to predict and analyse the system's behaviour accurately is imperative.
Fluid simulations are typical when studying baffle design but only consider the dynamics within the tank. This project aims to create a general-purpose model of a road tanker that captures road, tyre and suspension dynamics and the effect on the chassis. With this model, different driving conditions can be applied, and the effects on various components can be studied.
This paper describes the process of using OASYS PRIMER as a pre-processor for DYNA to build a Fluid-Structure Interaction (FSI) model with Smooth Particle Hydrodynamics (SPH) used to model the fluid and Explicit Dynamic FE techniques to model the chassis. The model will simulate a tanker undergoing emergency braking and other worst-case scenarios to identify areas that experience high fatigue levels due to liquid sloshing. The model was validated using extensive road testing carried out on a tanker using acceleration and strain data by Resonate Testing Ltd.
Crossland uses 304 stainless steel sheet metal in the construction of their tank, and high cycle fatigue and tensile testing were carried out to produce load curves to characterise the material using *MAT_123 (MODIFIED_PIECEWISE_LINEAR_PLASTICITY).
In this paper, we describe (i) The process of creating a Multiphysics simulation of a road tanker. (ii) The process of material characterisation for S304 steel. (iii) The testing was undertaken on a road tanker to validate the model.

Numerical Dynamic Characterization of a Xenon Satellite Propellant Tank employing Discrete Element Spheres

PIERROT Thomas[1], GUILPIN Antoine[1], LEGAUD Tess[1], LAPOUJADE Vincent[1], CHAMBE Jean-Emmanuel[2], CHARLOTTE Miguel[2], GOURINAT Yves[2]

[1]Dynas+
[2]ISAE-SUPAERO

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 supercritical 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 supercritical 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 supercritical 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 supercritical or cryogenic fluid and larger applications. The project combines numerical, analytical 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 three main steps: - 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. - Identify an analytical methodology that should enable scaling from experimental device shape to real cryogenic fluid tank.

ICFD Fluid Structure Interaction Modeling for the Seismic Evaluation of a Nuclear Pool

Nicolas Van Dorsselaer[1], Vincent Lapoujade[1], Morgan Le Garrec[1], Jean-Philippe Tardivel[2]

[1]Dynas+, 5 avenue Didier Daurat - 31400 Toulouse France

[1]ORANO Projets, 23 place de Wicklow - 78180 Montigny-le-Bretonneux France

Abstract

With the evolution of the modeling techniques and the computational power available, new problems can be addressed by numerical simulation. The ability of LS-DYNA to perform efficiently Multiphysics simulations allows users to go further and further in modeling to assess the classic dimensioning methods used for nuclear facilities design.

This paper relates to the seismic behavior of the civil engineering of a pool by an analysis method based on a real modeling of the water and its interaction with the structure. The objective is to compare the results obtained in fluid / structure interaction with the results obtained with a classic modeling of water by added masses calculated using the Housner method.

The fluid / structure interaction model created combines a modeling of the water by the Incompressible Computational Fluid Dynamics solver of LS-DYNA and a modeling of the structure by the implicit mechanical solver of LS-DYNA. This modeling takes advantage of the strong coupling capabilities between the two solvers to successfully simulate fluid movements with pressure oscillations in the depth of the pool and waves on the surface due to seismic loading.

Recent Developments of the EM-Module in LS-DYNA

Lars Kielhorn[1], Thomas Rüberg[1], Jürgen Zechner[1]

[1]TAILSIT GmbH, Graz, Austria

Since 2017, TAILSIT maintains a close collaboration with Ansys/LST. Our partnership focuses on the enhancement of LS-DYNA's electromagnetic (EM) solver module which is based on a coupling between Finite Element (FEM) and Boundary Element Methods (BEM).
A significant speed increase was achieved with the implementation of the monolithic FEM-BEM coupling solver, which has served as a basis for all the following improvements:

- modeling of ferromagnetic materials,
- permanent magnets,
- the evaluation of electromagnetic forces and
- voltage driven simulation.

All these enhancements of the EM solver make LS-DYNA highly suited for Multiphysics problems, e.g. the design of magnetic latches, haptic engines and motors as well the simulation of metal forming or inductive heating processes.

The latest development for LS-DYNA's EM module relates to the latter application, where alternating currents (AC) are used in the process of induced heating. Until now it was necessary to model at least one sinewave of the excitation using a large amount of time steps. To this end, such a simulation required either using very small timesteps or the concept of micro (resolving the AC current) and macro (resolving the model's dynamics) time steps. The newest enhancement allows it to represent AC sources directly in the frequency domain. Using this feature and the fact that no air needs to be meshed, the modeling of, for example, translational or rotational hardening processes or the dimensioning of flux concentrators is now possible in LS-DYNA in a more efficient way.

In this presentation, we give an overview of TAILSIT's contributions to the EM module with emphasis on the latest frequency domain implementations. We present benchmark results (Fig. 1 & 2) and discuss potential improvements for the near future.

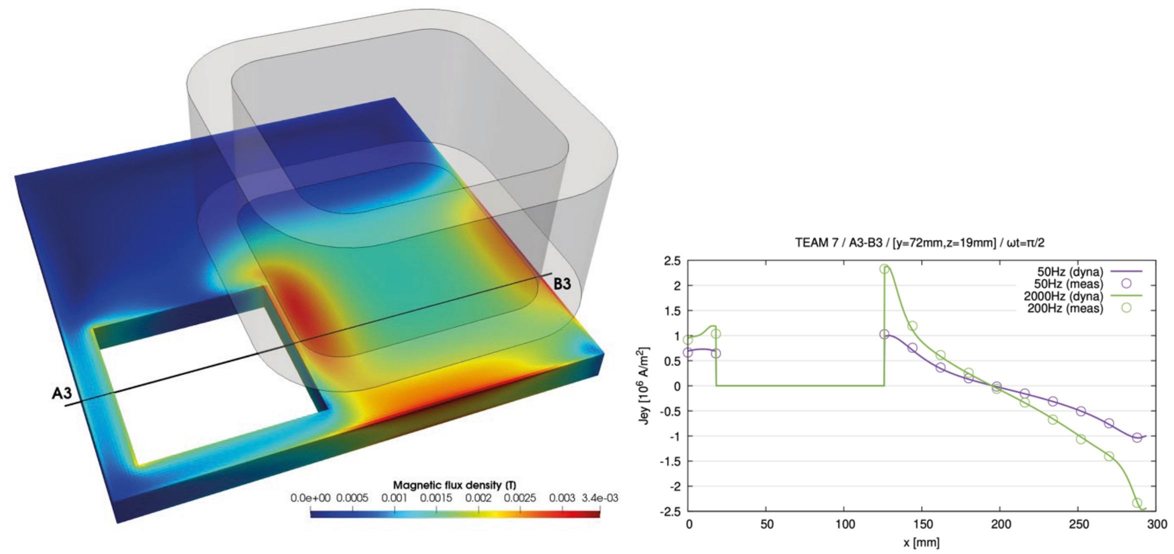


Fig.1: COMPUMAG TEAM 7 problem - magnetic flux density and measure line A3-B3 (left), comparison of the calculated (dyna) induced currents with measurements (meas) along the line A3-B3.

Improvements of LS-DYNA ICFD's two-phase level-set solver

Zlatko Solomenko, Facundo Del Pin, Inaki Caldichoury, Rodrigo Paz, Peggy Huang

Ansys Inc.

Abstract

Interface-capturing techniques are used in a variety of multiphase flow applications, e.g., free-surface flows, microfluidics, capillary flows. Among them, the level-set method is attractive as it gives a simple representation of the interface – the level-set scalar field is continuous, and the interface is generally chosen to be the zero level-set. The level-set function that is used here is the signed distance to the interface. That function evolves in time as it is transported by the flow velocity. As the flow velocity is not uniform in general, the level-set function may cease to be a distance function. Therefore, it must be reinitialized while maintaining the actual zero level-set. Numerically, a smooth level-set function is wanted. Perturbations in the level-set scalar field might yield unphysical oscillations at interfaces otherwise. In two-phase flow simulations, the solve of Navier-Stokes equations yields the flow velocity that transports the level-set function, which may need be reinitialized after a few time steps. Numerical methods that are deployed to solve those problems must be chosen meticulously. Depending on the type of flow, advanced numerical techniques must be used to avoid unphysical motion of interfaces. Several benchmark tests will be presented to assess the solver accuracy on flow types for which numerical simulation is challenging, e.g., flows with surface tension, flows with high viscosity and density ratios, free-surface flows.

COMPARING PLATE-LEVEL BLAST ANALYSIS USING ALE, S-ALE AND CONWEP METHODS

Ahmet Salih Yilmaz[1], Barkin Sonmez[1], Ulas Ozcan Erkan[1]

[1,]Research and Development Department, Anadolu ISUZU, Çayırova, 41435 Kocaeli, Türkiye

1 Introduction

The simulation analyses of the explosion were performed using three different methods: LBE (Lagrangian-based Eulerian), PBM (Particle-based Method), and ALE (Arbitrary Lagrangian-Eulerian). The reference point cloud data obtained from the scanning process after the explosion tests conducted on a 12 mm thick M400 steel plate indicated a deformation of approximately 12.5 cm at a Y-directional distance between the measured points. Based on this measurement, the explosion analyses were simulated using the aforementioned three methods. The scenario involved the detonation of 6 kg of TNT beneath the structure.

*ALE *S-ALE *CONWEP *FINITE ELEMENT

2 Test

2.1 Plate Level Burst Test

As stated in Stanag 4569 level 2, 6 kg of tnt was buried in the soil from a certain distance by opening a pit at the depth specified in the standard. The prepared torzo was brought to the test area and the explosion test was carried out by adding weights on the torzo so that the plate would not fly due to the explosion.

Fig.1: Real Test Pictures

2.2 Material testing

With the Split Hopkinson Bar test, the compressive stress-strain behavior of the materials at different deformation rates is determined, as well as the impact properties at low speeds and the wave transition in multi-layer materials. When combined with static velocity experiments, structural equations can be created that can be used to model materials in impact events with numerical methods or programs.

Fig.2: Split Hopkinson Bar Test Pictures

3 Analysis Methods

3.1 Ale / S-Ale

The Arbitrary Lagrangian-Eulerian (ALE) method is a computational technique used in fluid dynamics and mechanics to solve problems with moving boundaries or deformable domains. The ALE mesh refers to a computational mesh that can adapt and deform to accommodate the motion of the underlying domain. Unlike the Lagrangian approach that tracks the material motion or the Eulerian approach that uses a fixed mesh, the ALE method combines both by allowing the mesh to deform while maintaining a fixed computational framework. This approach is particularly useful for problems involving large deformations, complex geometries, or fluid-structure interactions. By using the ALE method and mesh, simulations can accurately capture the motion and deformation of objects or materials, enabling realistic and reliable analysis in various scientific and engineering fields.

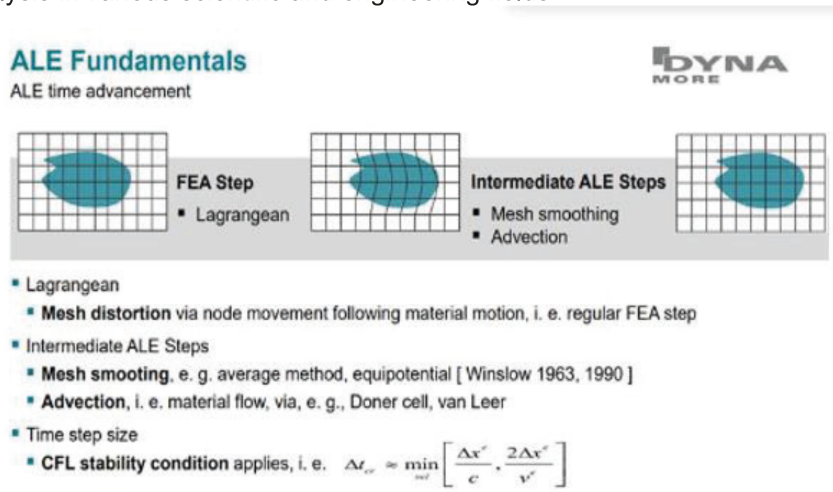


Fig.3: Ale Formulation

3.2 Conwep

The CONWEP blast method is an established engineering tool used to estimate the effects of explosive detonations on structures and individuals. It employs empirical relationships and mathematical models to assess key blast wave parameters, including peak pressure, duration, impulse, and phase duration. By considering factors such as explosive yield, distance, and environmental conditions, the method provides an estimation of the potential damage and effects caused by the blast. Widely applied in military, civil engineering, and security contexts, the CONWEP blast method aids in evaluating structural vulnerabilities, human injury risks, and developing protective measures against explosive threats.

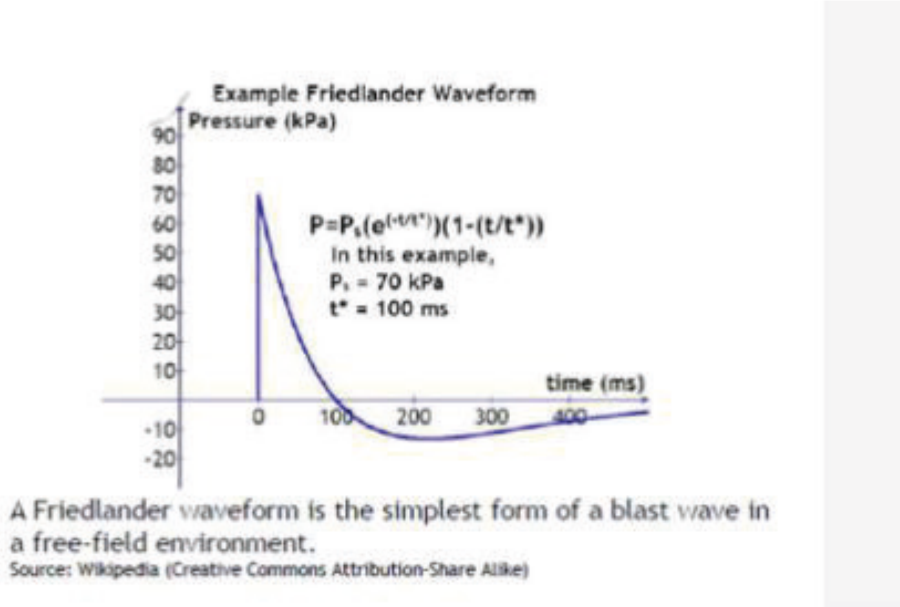


Fig.4: Conwep Formulation

4 Figure and Tables

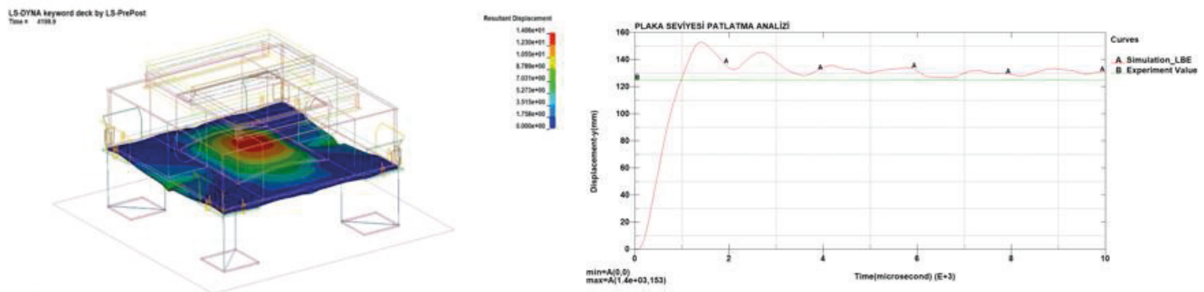


Fig.5: Conwep Method Virtual Analysis Result

Blast / Explosives

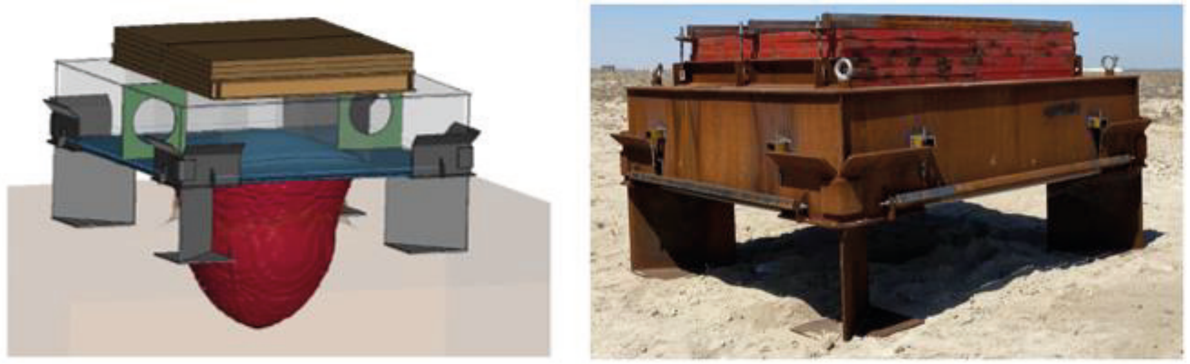


Fig.6: Real Torzo Picture with Ale Method Virtual Explosion

5 Summary

The plate deformation values were measured on the plate through explosion tests conducted for Stanag 4569 Level 2 blast testing. Subsequently, the deformed plate was scanned and converted into CAD data.

A mathematical model of the material used for Stanag 4569 Level 2 blast analysis was developed. Split Hopkinson bar tests were performed to characterize the material at high velocity. As a result of these tests, the Johnson-Cook stress and damage models for the material were obtained.

The obtained mathematical model was assigned to the finite element model of the plate using the Johnson-Cook material model.

Blast analyses for Stanag 4569 Level 2 was conducted using the prepared finite element model with different methods such as CONWEP, ALE, and S-ALE methods. The results were compared against the deformed scanned plate.

During the blast analyses, attention was given to comparing the different methods with the actual test results and analyzing the computation times.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

A Comparative Study of Modeling Approaches for External Structures in Mine Blast Simulations of an Armored Military Vehicle

İsmet Kutlay ODACI, Samet Emre YILMAZ, İlker KURTOĞLU

FNSS Savunma Sistemleri A.Ş.
Ankara / TÜRKİYE

1 Abstract

External structures are known to be critical in ensuring the protection of occupants in military vehicles during mine blast events. There are variety of modelling approaches that can be employed to represent external structures in mine blast simulations of armored military vehicles. This study aims to present an accurate configuration considering the modelling efforts and tight project schedules by comparing different modeling techniques applied to external structures, such as add-on armor plates and other external subsystem components. A whole vehicle finite element model is utilized for an on-going research and development project to evaluate the effectiveness of these modeling approaches by comparing simulation results with live fire test data of Hybrid III dummy and plastic deformations of the hull structure. The findings emphasize that the modelling approach of not only primary protective structures but also other external components significantly contributes to better representation of the tests. Configurations featuring accurately modeled external structures demonstrate improved accuracy in occupant safety assessment. The outcomes of the study contribute to enhancing the efficiency and reliability of the conceptual design phase by providing faster and relatively reliable finite element solutions, specifically in terms of representing external structures in the simulations.

Blast / Explosives

Correlative Approach to Mine Blast Effects via Conducting Real Test Campaigns and Simulating in LS-DYNA

I. OZCAN[1], B. ISIKER[1]

[1]Nurol Makina ve Sanayi A.Ş.

Abstract

In this study, pressure data were collected by conducting free field blast tests with explosives placed inside the steel pot to verify the explosive model. Free field blast analysis was performed using the Structured Arbitrary Lagrangian-Eulerian (S-ALE) method in the LS-DYNA® software under the same boundary conditions, and the pressure values obtained from the test were compared. Plate tests were performed in consideration of the verified explosive model with explosives placed inside the steel pot. Tests were carried out for 3 different designs, which consist of flat plate, twisted plate, and plate with deflector. Elastic and plastic displacement measurements were taken during the tests. LS-DYNA® software was used to perform analyses using the Johnson-Cook material model obtained from Split Hopkinson bar tests for plate materials and the S-ALE method. The effect of the distance between the plate and the explosive, the behavior of the source during the explosion, the effect of plate geometry, and the comparison with analysis results were investigated as a result of the plate tests.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Modeling of Directional Focused Fragmentation Charge (DFFC) – Investigation of Different Approaches

İsmet Kutlay ODACI, Samet Emre YILMAZ, İlker KURTOĞLU

FNSS Savunma Sistemleri A.Ş.
Ankara / TÜRKİYE

1 Abstract

The aim of this study is to examine the effects of explosion-accelerated clusters of projectiles, which is in literature referred as Directional Focused Fragmentation Charge (DFFC), on target armor structures. The primary challenge in this study is to develop an accurate model for the explosive and fragments configuration, since the scenario involves a close-range explosion and fluid-structure interaction (FSI) due to the direct contact of fragments with the explosive. To find an appropriate and stable solution to this challenge, various techniques are explored for modeling both the explosive and the cluster of fragments.

In the modeling of explosive, two different approaches are considered: the structured Arbitrary Lagrangian-Eulerian (S-ALE) method [1] and the **`*PARTICLE_BLAST`** (PBM) approach [2]. In terms of modeling the cluster of fragments, the classical Lagrangian approach and the discrete element method are utilized in combination with explosive modeling techniques. Each of these combinations of methods have their own advantages and limitations which will also be discussed during the presentation of the work.

To evaluate the effectiveness of these modeling techniques, the dispersion of fragments on the target plate is compared with the one obtained in field tests. After comparison of the results, it is observed that utilizing the S-ALE approach for modeling the explosive and using the classical Lagrangian approach for the cluster of fragments yields a stronger correlation with the dispersion observed in the experiments than PBM method combined with discrete element method.

2 References

[1] Chen, H.: "LS-DYNA® Structured ALE (S-ALE) Solver", 14th LS-DYNA® International Users Conference, 2016, Detroit, USA.

[2] Teng, H., Wang, J.: "Particle Blast Method (PBM) for the Simulation of Blast Loading", 13th LS-DYNA® International Users Conference, 2014, Detroit, USA.

Creating Machine-Learning-Friendly Training Data from Crash Simulation Data

Sarah Zenne[1], Joachim Sprave[1], Markus Stoll[2]

[1]Mercedes-Benz AG
[2] Renumics GmbH

1 Abstract

We present a method for generating training data from FEM meshes based on element properties. Learning at the element level is often indicated when larger structures, especially parts, are too inhomogeneous in size or geometry. At the element level, quality metrics and other information are gathered from the neighborhoods of elements, just as convolutional neural networks for image processing gather information from the neighborhoods of pixels. But in general, neighborhoods of elements are not as well structured as pixels in images. Instead, they form irregular graphs which cannot be processed by standard NN architectures directly. There are two approaches to make irregular graphs machine-learning-friendly: specialized NNs such as Graph NNs, and preprocessing of data. While Graph NNs seem to be a natural choice, data preprocessing has still its advantages in overall simplicity and explainability. In this presentation/paper, we will highlight these advantages. Together with the method we present a Python library that allows CAE engineers to extract machine-learning-friendly data from LSDYNA keyfiles as well as simulation results with a minimum amount of coding. This library streamlines the process of preparing mesh data for machine learning and facilitates the implementation of various machine learning models for mesh quality evaluation. The data generation and subsequent usage by machine learning methods is shown by the example of learning mesh quality assessment from labelled training data. Results for this data that already have been published are used as a baseline for a comparison of model accuracy.

Explorative ML-based approach to adaptively select design variable ranges for CAE crashworthiness optimization tasks

D. Steffes-lai[1], R. Iza-Teran[1], M. Pathare[1], J. Garcke[1]

[1] Fraunhofer SCAI, Schloss Birlinghoven 1, 53757 Sankt Augustin

1 Abstract

Exploratory analysis of many simulations in crashworthiness analysis by dimensionality reduction provides a convenient way to detect similar behavior between simulation results as well as to detect outliers. In addition, a sensitivity analysis to identify the most influential input variables and surrogate modelling on a reduced input basis is state of the art for individual components in CAE vehicle development. We propose to improve the optimization task by the use of this exploratory machine learning-based approach. A DOE with only few simulations builds the basis of the explorative iterative workflow. Sensible input variables are identified along the low dimensional embedding coordinates for different groups of parts of the vehicle structure. Next, surrogate models are constructed for the sensible variables. An adaptive sampling strategy to identify additional simulation runs based on the expected improvement of the surrogate model is applied to improve the prediction accuracy. Finally the objective function of the optimization is adapted taking into account the sensible quantities. This approach will lead to an improvement of the predicted quantity of the surrogate models and thus of the optimization results compared to classical surrogate modeling. The optimization might not get stuck in local minima and need fewer iterations, as the input data are pre-selected and thus already more homogeneous. The approach is demonstrated on a vehicle frontal crash analysis of the Toyota Yaris for multiple design changes in selected components, simulated with LS-Dyna.

Machine Learning / AI I

Innovating feasible Design with Machine Learning through the Combination of Topography Optimization and Ansys Forming

Sebastian Stahn, Kang Shen

Anys Inc.

Abstract

The field of design and engineering is constantly evolving, driven by the need for more efficient and innovative solutions. One such advancement is the combination of topography optimization and Ansys forming, powered by machine learning. This cutting-edge approach offers a unique opportunity to revolutionize the way we design and validate feasible designs.

By leveraging machine learning algorithms, designers can optimize product performance through topography optimization techniques. This process involves identifying the most efficient distribution of material within a given design space, resulting in lighter, stronger, and more cost-effective structures. The integration of Ansys forming further enhances this process by simulating sheet metal forming.

The combination of topography optimization and Ansys forming not only enables engineers to create innovative designs but also ensures their feasibility in real-world applications. By validating these designs using simulations, design engineers can assess factors like structural integrity, manufacturability, and performance under various loading conditions.

Machine learning plays a crucial role in this innovative approach by continuously improving the optimization algorithms based on historical data and feedback loops. It enables designers to explore a vast design space quickly while considering multiple constraints simultaneously.

With this powerful combination at their disposal, engineers can push the boundaries of what is possible in design-driven industries such as automotive, aerospace, consumer goods, and more. Ansys forming integration with topography optimization allows companies to remain competitive while producing innovative and feasible products.

In conclusion, the synergy between topography optimization and Ansys forming powered by machine learning represents a significant leap forward in designing feasible yet highly innovative products. This approach not only saves time but also offers immense potential for cost savings without compromising on performance or quality. As technology continues to advance rapidly in these fields, embracing these advancements will be crucial for staying competitive in an ever-evolving market landscape.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Machine learning using a hybrid quantum-classical algorithm

Maximilian Spiegel[1], Sebnem Gül-Ficici[1], Ulrich Göhner[1]

[1]University of Applied Sciences Kempten, Bahnhofstraße 61, 87435 Kempten, Germany

1 Abstract

The industrial sector uses artificial intelligence (AI) in many ways. I.e. anomaly detection to identify and examine abnormal behavior of machines, such as voltage and current fluctuation. To develop self driving cars AI is used to perform segmentation of the environment to navigate the vehicle and make decisions, preferably in real-time.
Quantum computers are already being used for special machine learning processes, achieving, in some instances, better results than a regular machine learning algorithm. This paper will elaborate on the upsides of a machine learning model consisting of a hybrid between a quantum machine learning (QML) algorithm and a classical machine learning algorithm.

KEYWORDS: Artificial Intelligence, Hybrid Quantum Machine Learning

2 Project Objectives

For a quick start in artificial intelligence, the following reference can be consulted for more details [1]. For an introduction on quantum mechanics take a look at [2] and for quantum computing refer to [3].
Machine learning, especially deep learning, requires an extensive data record in order to be trained appropriately. Processing large amounts of data, however, is expensive, due to its highly time consuming and computing-intensive nature. The added value of this trained model lies in its ability to lower cost through an early detection of errors. Efforts required for both, training and evaluation, could possibly be diminished notably by using a quantum computer.
Data measured by sensors are usually formatted in a way that allows regular computers to interpret them. Even if the data was an analog measurement it will still be stored as a digital bit string. However, quantum computers are unable to interpret bit strings, therefore necessitating a data encoder before handing over the data to a quantum algorithm. Such a hybrid quantum-classical algorithm is often referred to as a variational quantum algorithm [4]. In this paper the Python libraries Qiskit by IBM and Pytorch have been used to build a hybrid QML.
The standard machine learning network pre-processes the input data. Its output is then transmitted to the QML network.
As mentioned the data is encoded as a binary string and needs to be converted. To complete the data encoding, the machine learning's output will be passed on to a quantum feature map. This quantum feature map then converts a binary string into quantum states that the quantum algorithm is able to work with. After the data has been converted into quantum states, they are forwarded to the "ansatz" which represents the weights of our "artificial quantum neurons". After the algorithm reached a conclusion a different problem arises; the data presentation now has to be reversed to compare the conclusion and adjust the weights within the neural network (see Fig. 1).

Machine Learning / AI I

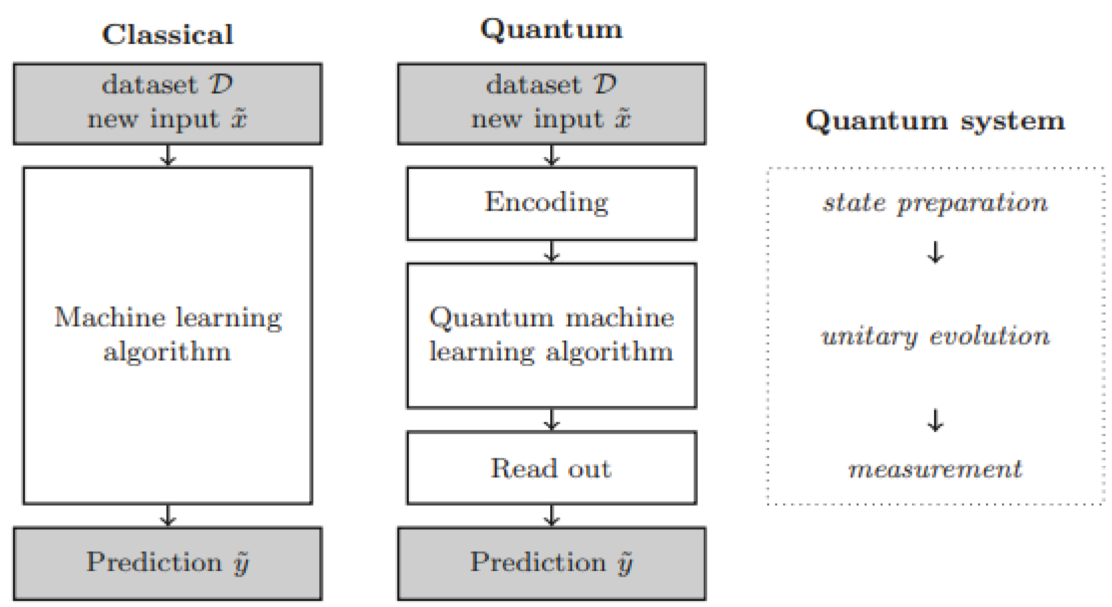


Fig.1: Comparison between a classical machine learning algorithm (left) and a quantum machine learning algorithm (middle). In our case, the prediction $\vec{y}$ of the classical algorithm is used as the input $\vec{x}$ of the quantum algorithm. Figure source: [3]

3 Summary

The performance of the complete algorithm is being discussed and the compute time of the data encoding process is being evaluated for different numbers of qubits. In the evaluation process, not only the results are considered, but also the formula for data encoding is taken into account. Using the qiskit library we encode the original data using a "PauliFeatureMap". The following section is citated from [5] to visualize the mathematics behind the data encoding.
The "PauliFeatureMap" is a Pauli Expansion circuit that will transform our data, $\vec{x} \in \mathbb{R}^n$, given by the classical machine learning network, where n is the „feature-dimension" (that is equal to the number of qubits used), into the quantum state

$$U_{\Phi(\vec{x})} = \exp\left(i \sum_{S \in I} \phi_S(\vec{x}) \prod_{i \in S} P_i\right)$$

S is a set of qubit indices that describes the connections in the feature map, I is a set containing all these index sets, and $P_i \in \{I, X, Y, Z\}$. Per default the data-mapping ϕ_S is

$$\phi_S(\vec{x}) = \begin{cases} x_i \text{ if } S = i \\ \prod_{j \in S} (\pi - x_j) \text{ if } |S| > 1 \end{cases}$$

4 Literature

[1] Russell, S.J.: "Artificial intelligence a modern approach", third edition, 2010.
[2] Griffiths & Schroeter, D. J. & D. F.: "Introduction to quantum mechanics", third edition, 2018
[3] Schuld & Petruccione, M. & F.: "Supervised learning with quantum computers", Vol. 17, 2018
[4] Caro & others, M. C..: "Encoding-dependent generalization bounds for parametrized quantum circuits", Vol. 5, 2021, 7 - 8
[5] IBM: "Qiskit Pauli Feature Map", https://qiskit.org/documentation/stubs/qiskit.circuit.library.PauliFeatureMap.html#qiskit.circuit.library.PauliFeatureMap, 2023, accessed on 2023-06-05

Fluid-structure interaction simulations of mechanical heart valves with LS-DYNA ICFD

Mariachiara Arminio[1], Dario Carbonaro[1], Sara Zambon[1], Rodrigo Paz[2], Facundo Del Pin[2], Umberto Morbiducci[1], Diego Gallo[1], Claudio Chiastra[1],

[1]PolitoBIOMed Lab, Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Italy
[2]Ansys, Inc.

Abstract

Introduction: Aortic valve disease has a high impact on patients' lives. Treatment often involves replacing the native valve with a prosthesis, and surgical mechanical prosthetic valves are typically preferred for younger patients. In recent years, fluid-structure interaction (FSI) modeling of prosthetic valves has emerged as an effective approach for the biomechanical study of these devices. In this context, this work presents a framework for FSI simulations of mechanical aortic valves using LS-DYNA ICFD solver. Methods: An idealized geometrical model of the aortic root and a geometrical model of a mechanical bileaflet aortic valve resembling St Jude Medical Regent valve (Abbott Laboratories, IL, USA) were created using HyperMesh (Altair Engineering, MI, USA) and SolidWorks (Dassault Systèmes, FR), respectively. The valve model was integrated into the aortic root by positioning the valve cuff in supra-annular position. Using HyperMesh, the fluid domain wall and valve leaflets were discretized with shell elements, and valve frame and cuff were discretized with tetrahedral elements. FSI simulations were performed using Ansys LS-DYNA R14.1 (Ansys Inc., PA, USA), coupling the ICFD solver to the implicit structural solver. The gap closure feature (*ICFD_CONTROL_GAP) was used to prevent flow across the valve during diastole. A physiologic transvalvular pressure gradient waveform and a steady zero pressure were applied to the inflow and outflow sections of the fluid domain, respectively. The no-slip condition was applied to the wall of the fluid domain and to the leaflets. Aortic root wall was assumed rigid. Results: The proposed FSI framework appropriately reproduced the kinematics of valve leaflets and the main hemodynamic features. In particular, the three-jets configuration of the flow during systole due to the valve bileaflet design was successfully captured, as well as the low blood velocity values along late diastole. The evaluated valve effective orifice area was equal to 2.16 cm2, meeting the requirement of the standard ISO 5840:2021. Discussion: The current findings demonstrate that LS-DYNA ICFD solver adequately captures mechanical aortic valves related fluid dynamics, highlighting the capacity of the solver in handling high-displacement FSI scenarios. In the future, this approach will be utilized to further characterize the fluid dynamics of these devices and to assess their performance replicating the in vitro tests recommended by ISO 5840:2021.

Biomechanics I

pyheart-lib: a python library for LS-DYNA multi-physics cardiac simulations

Martijn Hoeijmakers, Karim El Houari, Wenfeng Ye, Pierre L'Eplattenier, Attila Nagy, Dave Benson, Michel Rochette

Ansys Inc.

Abstract

Physics-based computer simulations of the heart is gaining rising interest for optimizing the design of medical devices and for its treatment prediction and planning. LS-DYNA offers a powerful framework for modeling cardiac electrophysiology, mechanics and fluid dynamics, as well as the coupling between the three physics. However, its wider adoption is hindered by several requirements among which: knowledge in cardiac function in health and pathology, expertise in numerical simulation, appropriate right modeling choices for the target application, availability of realistic heart geometries. In this paper, we present a free to use python package that allows for the generation of physiologically accurate heart models in an automatic and modular fashion. The architecture is organized in an abstract form that allows users to easily choose between the different physics, anatomical chambers of interest and parameters of interest and export the LS-DYNA keyword files ready for simulation. We also introduce the relevant heart modeling features that are available in LS-DYNA and present two exemplary models generated by the package: a full electrophysiology heart model and a bi-ventricular mechanical model.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Manufacturing, crimp and deploy simulations of Transcatheter Aortic Valve Replacement

Nils Karajan[1], Facundo Del Pin[2], Marco Sensale[2], Nick Wagner[2], Thomas Weckesser[2]

[1]DYNAmore GmbH, an Ansys Company
[2]Ansys, Inc.

1 Motivation

In general, the design of a stent frame for transcatheter valve replacements is a challenging task as manufacturing has an influence on the final shape of the device and a complicated fluid-structure interaction load case is required for useability analysis to predict the correct stresses and strains for fatigue life assessment. Typically, one distinguishes between balloon-expandable frame designs mostly made of stainless steel or self-expandable frame designs made of nitinol shape memory alloy. The focus of this contribution is on self-expandable frames to demonstrate manufacturing and laboratory load cases as well as crimp and deploy simulations for useability load cases. The latter will be presented by Facundo Del Pin after this presentation where the use-case is illustrated by a fluid-structure interaction simulation of the deployed device in an aortic root anatomy. The mockup geometry is based on the Evolute-R System of Medtronic, which is frequently used in literature [1,2].

2 Simulation workflow

Often, the design idea for stent frames originates from a 2d flat pattern which is mapped onto a cylinder. During manufacturing, a laser is cutting the stent structure from a nitinol tube. The shape setting from the small tube to the final device is then done in several expansion and annealing steps to keep the strains low. Once the device is brought in shape, the new design idea is usually tested in some laboratory test setups and metrics are recorded to compare the performance of the design ideas. Due to the complexity of the simulations, a deployment into a patient specific artery is not yet done in industry on a routine basis but is currently only carried out by research institutes. The goal of this and the subsequent presentation by Facundo Del Pin is to simplify these simulations and work toward connecting them in an automated workflow. To aid the deployment in patient specific anatomies, an interactive stent positioning tool will be presented. Once in place, such a workflow will allow design engineers in industry to quickly assess design ideas under real-life loading conditions.

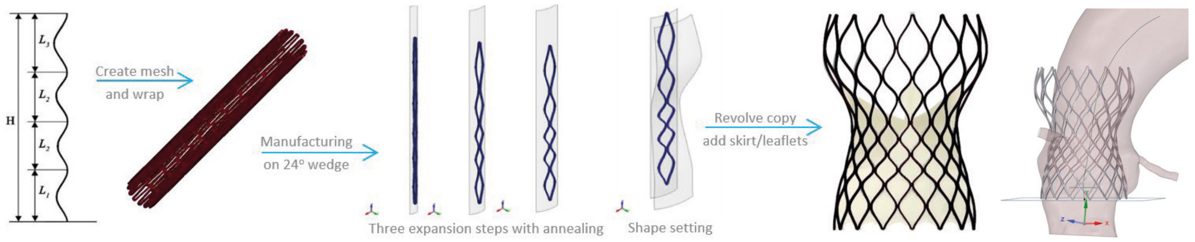


Fig.1: From the flat-pattern design idea to the device as well as positioning in an anatomy.

3 Literature

[1] Luraghi, G.; Migliavacca, F.; Rodriguez Matas, J.F.: The Effect of Element Formulation on FSI Heart Valve Simulations. Proceedings of the 12th European LS-DYNA Conference 2019.

[2] Luraghi, G.; Migliavacca, F.; Garcia-Gonzales, A.; Chiastra, C.; Rossi, A.; Cao, D.; Stefanini, G.; Rodriguez Matas, J.F.: On the Modeling of Patient-Specific Transcatheter Aortic Valve Replacement: A Fluid–Structure Interaction Approach. Cardiovascular Engineering and Technology 2019.

Streamlining prosthetic heart valve simulation, from stent positioning to fluid structure interaction

Facundo Del Pin[1], Rodrigo R. Paz[1], Nils Karajan[2], Peggy Huang[1]

[1]Anys Inc.
[2]DYNAmore GmbH

Abstract

LS-DYNA, a comprehensive Multiphysics solver, offers an efficient, unified platform for prosthetic heart valve simulation. From stent crimping and anatomically tailored deployment to advanced fluid structure interaction (FSI), LS-DYNA's capabilities streamline the complex process of prosthetic heart valve analysis. The solver's robust FSI allows for detailed hemodynamic evaluation, critical in predicting valve-associated thrombosis risk. LS-DYNA's advanced post-processing tools further enhance this evaluation. Residence time computation and stagnation area/volume identification facilitate thrombosis risk assessment and offer comprehensive insights into valve performance. This data allows design exploration and comparative evaluations, enabling the possibility of valve optimization. The advanced contact features of LS-DYNA provide a way to create complex assemblies of leaflets, skirts, stent, and anatomy/conduit walls with any other supportive structure that is needed to match experimental or clinical setups. Contact is a critical aspect of valve simulation that can be tackled efficiently using LS-DYNA structural solver and FSI. In this paper the state of the art in solver features will be presented to provide a step-by-step workflow that can be used in a practical real world case study for the evaluation of prosthetic heart valves.

Development of Effective Taylor-Quinney Coefficient Tables of *MAT_224 for Aluminum 2024-T351

Chung-Kyu Park[1], Kelly Carney[1], Paul Du Bois[1], Cing-Dao Kan[1]

[1]Center for Collision Safety and Analysis, George Mason University, Fairfax, VA, USA

Abstract

The LS-DYNA constitutive material model *MAT_224 was developed for the analysis of high-speed impact problems, e.g., a real turbine engine fan-blade release event. The thermal softening in *MAT_224 implemented the function to calculate the temperature increase induced by plastic work but ignored a heat conduction effect because it was assumed that a metallic material would be deformed under an adiabatic condition in such high-speed impact problems. The temperature rise in *MAT_224 is estimated by the Taylor-Quinney Coefficient (TQC) which describes the percentage of plastic work converted into heat energy. Furthermore, *MAT_224 input parameters for Aluminum 2024-T351 (Al2024) have been developed with the series of test data and the referenced physical TQC constant of Al2024, and they showed well-correlated ballistic impact simulations. However, these constant physical TQC in the current material models of *MAT_224 could be problematic for the impact analysis of a full engine structure where the strain rate ranges from quasi-static to extremely high rates because the constant physical TQC could not represent the material thermal softening process in low and intermediate-rate deformation properly without considering heat conduction. So, an effective TQC that considers heat conduction at different rates is required. In this research work, the effective TQC table of *MAT_224 for Al2024 was developed to replace the current constant TQC. The methodology to create the effective TQC table was developed by using a two-step approach. In Step 1, the thermal-structural analyses of tensile tests were conducted to verify the referenced TQC values and generate the additional temperature-strain curves at additional rates which were not covered by the physical tensile tests. In Step 2, the structural-only analyses of tensile tests were conducted to calibrate the effective TQC values at all the rates for the effective TQC table and validate the calibrated effective TQC table.

Numerical Modelling of Sheet Metal Damage in LS-DYNA using GISSMO

Erik Stålbrand[1], Axel Hallén[2]

[1]Alfa Laval Technologies AB
[2]DYNAmore Nordic AB

Abstract

In many engineering applications, as well as in sheet metal forming, there is little interest in studying material behavior until complete fracture. However, when damage and failure of the material is required in order to achieve the purpose of an analysis, it is crucial to understand and predict the material behavior from initial deformation until fracture. Forming limit diagrams are commonly used to estimate sheet metal formability but fall short in the sense that they only predict whether or not the material will fracture, but do not describe the fracture behavior. A damage model on the other hand, describes the material degradation until complete fracture.

Modelling of the damage evolution of a sheet metal under deformation is a useful tool within the field of engineering and materials science and can provide information regarding material capabilities and the underlying physical principles of sheet metal forming. With the help of damage modelling, engineers can predict the likelihood of material damage and minimize the need for physical testing as well as time and material waste.

In this paper, a strategy for numerical simulation of cutting during concurrent pressing of stainless steel sheet metal is investigated while accounting for the stress triaxiality and Lode angle parameter. An inverse modelling approach is used, where both a material model and a damage model are developed. The damage model is developed using the GISSMO damage model (Generalized Incremental Stress State dependent damage Model). The fracture strain is defined in the stress triaxiality and Lode angle parameter space as a surface, and experiments using different specimens are conducted in an attempt to cover the space of possible stress states. The modified Mohr-Coulomb fracture criterion is used to predict fracture strain in a variety of stress states. The simulation software LS-DYNA is used for numerical modelling and the software LS-OPT is used in conjunction with LS-DYNA to identify damage parameters for use in the GISSMO model.

The results show good agreement with experimental load case data but due to issues with stress state characterization of some of the conducted experiments, further validation is necessary before the damage model can be used in practice. The stress states experienced by a sheet metal under deformation were ultimately found to be widely spread in the stress triaxiality and Lode Angle parameter space, whereas the material tests were limited to a relatively small region. The results from this work show a strong potential for the inverse modelling approach to model the evolution of damage using GISSMO. It was concluded that, in order to accurately describe the material behavior during cutting, experimental data from a wide variety of stress states is necessary and the results from this work highlight that the resulting damage model is highly dependent on the accuracy of the conducted experiments.

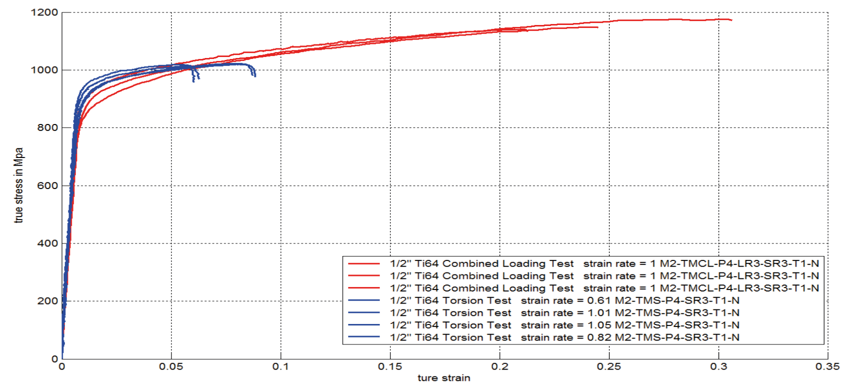
A first shot at tabulated 3D orthotropic failure application to Ti-6Al-4V

Paul Du Bois[1], C.K. Park[2]

[1]Forming Simulation Technologies, Northville, USA
[2]GMU, Fairfax, USA

Abstract

The Center of Collision Safety Analysis at George Mason University has made an intensive and ongoing effort on material and failure modeling of Ti-6Al-4V in the context of the FAA's aircraft catastrophic failure prevention program since 2006. Some of the previous work included tabulated isotropic failure models (MAT_224), asymmetric plasticity with associated flow (MAT_224_GYS) and asymmetric orthotropic plasticity (MAT_264) . Much of this work was presented in previous LS-DYNA user conferences on both sides of the Atlantic. Two constant tendencies in our work are regularization and tabulation. Input data are tabulated rather than fitted to some analytical expression in order to achieve a maximum flexibility and allow to fit a maximum number of datapoints with the best possible accuracy. Regularization comes in as a 'necessary evil' to counter the inevitable mesh dependency in numerical models once the material exhibits localization of plastic strains , which will always happen as we approach failure in ductile metals.
Extensive experimental work performed at the Dynamic Mechanics of Materials Laboratory of the Ohio State University has shown directional dependency of the failure strains in both tension and compression tests. It is however the results of torsion tests on hollow cylinders that constituted the main motivation for the current work. The differences in failure strain measured in cylinders that were cut in the thickness direction of the plates versus cylinders that were cut with axes parallel to the plate were indeed very significant and ignoring them in industrial applications (as we have done so far) would leave any numericist uncomfortable. It was therefore decided to extend our current orthotropic plasticity model (MAT_264) with the capability of simulating orthotropic failure. Tabulated orthotropic failure is currently available in LS-DYNA as an option in MAT_GENERALIZED_DAMAGE (eGISSMO) for the plane stress case. It requires the user to input 3 failure curves giving the failure strain in function of stress triaxiality resulting from experiment performed on samples such that the direction of the first principal plastic strain is under 0, 90 or 45 degrees to the material rolling (or extrusion) direction. Due to the assumption of orthotropy a fourth curve equal to the 45 degree curve is internally generated for the 135 degree direction. The current project can thus be considered a 3D generalization of this plane stress model.



The generalization to 3D is somewhat complicated mainly because the transformation between the material system and the principal plastic strain system under plane stress conditions requires a rotation defined by a single angle (around the axis normal to the plane of the plate). In the full 3D case, we have a rotation defined by 3 (successive) Euler angles or 3 Bryan angles. In our approach we have chosen to work with Bryan angles applied in the order X-Y-Z. This choice has led to an

approach were the plastic strain rate tensor is decomposed in 12 components and the incremental damage is determined for each of these components. A single incremental damage is then computed from the failure strains attributed to each component and the coefficient of each component after decomposition of the plastic strain rate tensor

$$\dot{d} = \dot{\varepsilon}_p \sqrt{\frac{\left(\frac{a_{000}}{\varepsilon_{00}^{fail}}\right)^2 + \left(\frac{a_{100}}{\varepsilon_{00}^{fail}}\right)^2 + \left(\frac{a_{001}}{\varepsilon_{90}^{fail}}\right)^2 + \left(\frac{a_{101}}{\varepsilon_{90}^{fail}}\right)^2 + \left(\frac{a_{010}}{\varepsilon_{tt}^{fail}}\right)^2 + \left(\frac{a_{110}}{\varepsilon_{tt}^{fail}}\right)^2 + \left(\frac{b_{121}}{\varepsilon_{45bc}^{fail}}\right)^2 + \left(\frac{b_{021}}{\varepsilon_{45bc}^{fail}}\right)^2 + \left(\frac{b_{102}}{\varepsilon_{45ab}^{fail}}\right)^2 + \left(\frac{b_{120}}{\varepsilon_{45ca}^{fail}}\right)^2 + \left(\frac{b_{020}}{\varepsilon_{45ca}^{fail}}\right)^2 + \left(\frac{b_{002}}{\varepsilon_{45ab}^{fail}}\right)^2}{(a_{000})^2 + (a_{100})^2 + (a_{001})^2 + (a_{101})^2 + (a_{010})^2 + (a_{110})^2 + (b_{121})^2 + (b_{021})^2 + (b_{102})^2 + (b_{120})^2 + (b_{020})^2 + (b_{002})^2}}$$

The paper shows a detailed derivation of the model as well as the results of verification and validation simulations that were performed to ensure it's proper functioning.

Consideration of inhomogeneous material properties of aluminium cast parts in crash simulations

P. Hager, P. Henn
Porsche AG

Fatigue assessment of an adhesively bonded EV battery enclosure, using LS-DYNA implicit tools

David McLennan, Michael Magnier

Arup

Abstract

Adhesively bonded aluminium structures are becoming increasingly popular within the automotive industry. Bonded connections are continuous, and therefore can avoid the stress concentrations which arise in discrete connections such as spotwelds, rivets or bolts, and thus have the potential to perform better from a fatigue perspective. However, bonded structures have their own challenges to analyse, particularly for predicting fatigue life, where limited data exists in the public domain. Using a generic electric vehicle (EV) battery enclosure as the case study structure, this paper demonstrates that LS-DYNA implicit solvers can perform all China regulation GB38031 mechanical vibration tests, which comprise a mixture of random vibration and fixed sine wave load cases. Using finite element (FE) results from analyses of these vibration load cases, a method developed by Sousa et al is adopted, where adhesive fatigue performance is predicted by first calculating an “effective stress” for each element in the model. To obtain a time-history of “effective stress”, a time-domain approach is taken using the LS-DYNA ***CONTROL_IMPLICIT_MODAL_DYNAMIC** keyword (a mode-based transient analysis, performed using modal superposition). This “effective stress” is then mapped onto an S-N curve (also derived from tests by Sousa), and the number of fatigue cycles from the test compared to the predicted number of cycles to failure for the adhesive. For a complete assessment of the battery enclosure, the aluminium fatigue performance is calculated separately using an equivalent LS-DYNA frequency-domain approach, with input from the power spectral density (PSD) of the mechanical vibration load cases and using keyword ***FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE.** The aluminium fatigue performance is predicted using the Dirlik method, which is embedded within LS-DYNA. It is shown that these LS-DYNA implicit tools provide a credible prediction of fatigue performance for adhesively bonded structures, and are a valuable design iteration tool, in combination with physical testing. It brings us a step closer to the one-code ideal, where a single LS-DYNA model is used for all implicit and explicit load cases, leading to a more streamlined CAE workflow.

Time-Domain Explicit Dynamic CAE Simulation for Brake Squeal

Gavin Song, Ford Motor Company

1 Abstract

Disc brake squeal is always a challenging multidisciplinary problem in vehicle noise, vibration, and harshness (NVH) that has been extensively researched. Theoretical analysis has been done to understand the mechanism of disc brake squeal due to small disturbances. Most studies have used linear modal approaches for the harmonic vibration of large models. However, time-domain approaches have been limited, as they are restricted to specific friction models and vibration patterns and are computationally expensive. This research aims to use a time-domain approach to improve the modeling of brake squeal, as it is a dynamic instability issue with a time-dependent friction force. The time-domain approach has been successfully demonstrated through examples and data.

NVH / Fatigue

A Study of Tied Contacts in Implicit Stress Analysis via a Fire Truck Tip-Over Simulation

Thomas Weckesser[1], Nils Karajan[2]

[1]Ansys, Inc.
[2]DYNAmore GmbH, an Ansys Company

1.Abstract

Tip-over simulations are critical in analyzing the structural integrity of firetrucks under various loading conditions. While setting up an initial assembly, it was noticed that different ways of modeling tied contact can result in variation in the simulation's convergence as well as the overall accuracy in the predicted stress. However, the influence of a single tied contact is not evident when used alongside numerous others in a large assembly. Therefore, a small tied contact example was developed in this paper to study eight different tied contact scenarios and determine the best practice for each case using available tied contact options in LS-DYNA®. A set of guidelines are thereafter summarized and implemented throughout many design iterations of the firetruck assembly. It was shown that the guidelines can help build more accurate and robust models.

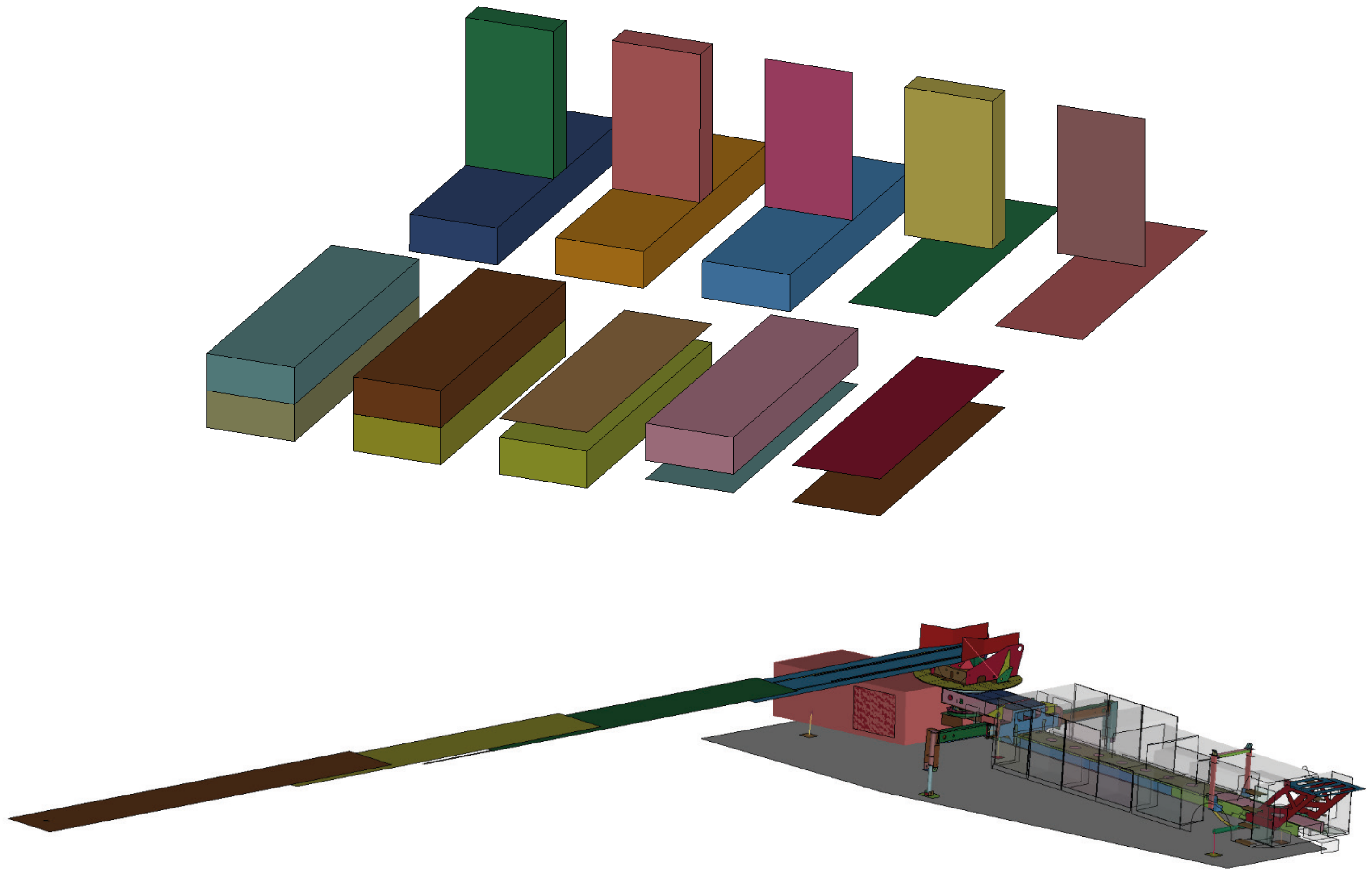


Fig.1: Example of the small tied contact simulation and the final fire truck tip-over simulation

Considering the Local Anisotropy in the Simulation Process Chain for Short and Long Fiber Reinforced Thermoplastics – Part I: Material Characterization

Harish Pothukuchi[1], Eike Reinhardt[2], Martin Schwab[1]

[1] 4a engineering GmbH, Industriepark 1, 8772 Traboch, Austria

[2] Dr. Ing. h. c. F. Porsche AG, Porschestrasse 911, 71827 Weissach, Germany

A huge number of short and long fiber reinforced thermoplastics play a decisive role in the automotive industry to ensure affordable lightweight design and availability in large quantities. The properties of these materials are especially highly influenced through the manufacturing process (typically injection molding for SFRT and LFRT). Over the last years, there is a strong industry interest to consider the manufacturing process induced local anisotropy inherent in these materials under a crash scenario.

This paper is the first part of a collaborative work undertaken with our partner Dr. Ing. h. c. F. Porsche AG that deals with the material testing with IMPETUS® and the material modelling calibration process for a *MAT_4A_MICROMEC with VALIMAT® in LS-DYNA. Besides the validation of anisotropic material deformation behavior, the calibration of the failure behavior is also considered in extensive detail. To obtain such a high-quality material card a workflow is developed, starting with molding adequate plaques for test specimens to characterize the underlying deformation and failure behavior at the coupon level under different loading scenarios up to final validation on component level.

Due to the short filling times, high speeds and pressures are a necessity in the injection molding process. This subsequently leads to the development of significant fiber orientations by the extensional and shear flows in the mold. Hence, material cards like the *MAT_4A_MICROMEC that can handle this manufacturing induced distribution of the fiber orientations will enable for an integrative simulation approach. The mapping of the fiber orientations from a process simulation to a structural simulation through FIBERMAP® among other approaches and the simulation process chain on component level are topics that will be discussed more in detail by our industry partner.

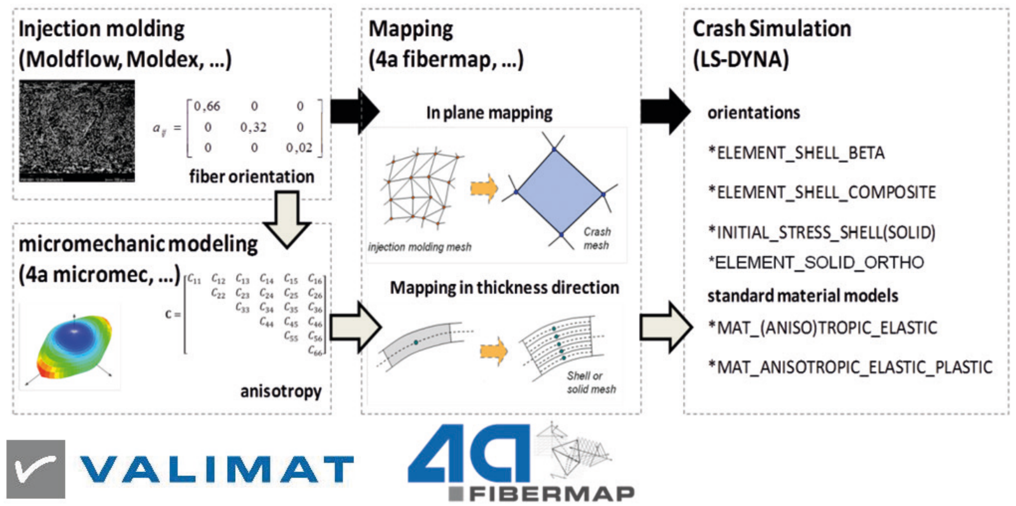


Fig.1: Available simulation process chain for injection molded parts.

Considering the Local Anisotropy in the Simulation Process Chain for Short and Long Fiber Reinforced Thermoplastics – Part II: Application within development

Eike Reinhardt[1], Harish Pothukuchi[2], Martin Schwab[2]

[1] Dr. Ing. h. c. F. Porsche AG, Porschestrasse 911, 71827 Weissach, Germany

[2] 4a engineering GmbH, Industriepark 1, 8772 Traboch, Austria

As part of the heavily hardware-reduced development in the automotive industry, a rapidly increasing number of validations are carried out in simulation. In addition, there is a wider use of structurally relevant plastic components - usually SFRT and LFRT. The property changes caused by the manufacturing process - triggered by local anisotropy - must be mapped in the simulation to generate predictable results. A major challenge for development is the availability of process simulation data.

In this second part of a collaborative work undertaken with 4a engineering GmbH, the use of integrative simulation is considered in an application-oriented manner. Various approaches are discussed regarding the mapping of fiber orientations to a structural mesh in vehicle crash applications. Furthermore, it must be estimated to what extent deviations are to be expected when using "generically created process simulations" compared to those based on optimized developed tool concepts.

The process chain from the generic - based on geometry information and in-house technological expertise - creation of process data to application in the whole vehicle simulation is shown. The influences of material model, mapping method and generic process data approach are discussed.

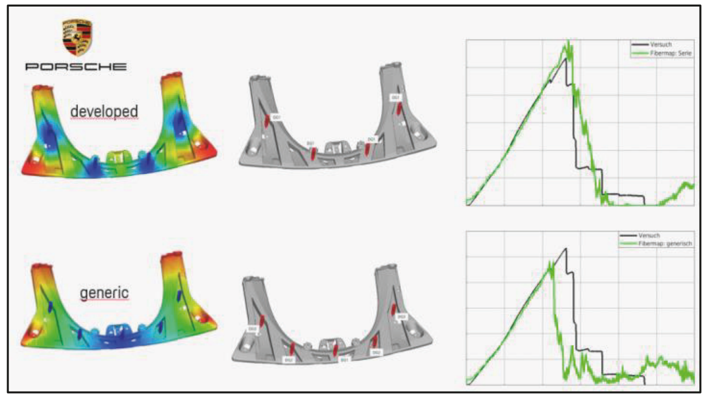


Fig.1: Process comparison: off-tool versus generic

Fig.2: Different failure occurs considering different mapping tool

Recent Enhancement for Modelling Adhesives in the Closed Manufacturing-Crashworthiness Process Chain

Thomas Klöppel, Tobias Aubel

DYNAmore GmbH an Ansys Company, Stuttgart, Germany

1 Abstract

Relatively recently, a new constitutive model ***MAT_GENERALIZED_ADHESIVE_CURING** (***MAT_307**) in LS-DYNA has been presented [1]. It has been developed in the public funded research project “DigiBody”, based on the results examined in [2], to model the behavior of adhesive materials throughout the whole manufacturing-crashworthiness process chain. One of the challenging points is to correctly capture the transition the material undergoes during the manufacturing process from the liquid to the solid phase.

This contribution briefly summarizes the main features of the constitutive model. Beside the above-mentioned material transition this also includes damage and failure modelling based on the TAPO model [2,3]. Furthermore, the key concepts of the material definition and implementation are recapitulated. This summary will provide the necessary basis for the discussion of recent extensions made in ***MAT_307** that are the main focus of this contribution.

The enhancements discussed here aim on widen the range of potential applications. In the “DigiBody” project, the material could successfully be calibrated and validated with experimental results for a BETAMATE™ material provided by DuPont [1]. Obtaining the requested accuracy when modelling further adhesives used in the automotive industries, however, required the implementation of additional curing concepts and curing kinetics models. Moreover, some improvements and new options have been added to the description of viscoelastic properties of the model. These changes will be discussed in some detail in this paper.

Material model ***MAT_307** provides the possibility to address damage and failure of adhesives depending on the current state of cure and the temperature. In most crashworthiness load cases in the automotive industries it can be assumed that the curing process has stopped (not necessary in a fully cured state) and the temperature is constant. Therefore, it seems beneficial to run the simulation of those load cases with a tailored and numerically more efficient material model such as ***MAT_TOUGHENED_ADHESIVE_POLYMER** (***MAT_252**)and locally manipulate the material properties based on the results of a coupled, thermal-mechanical process simulation. Naturally, this requires, on the one hand, a mapping procedure, which is beyond the scope of this work. On the other hand, a modification in ***MAT_252** to process the additional information becomes necessary. Recent implementations will be presented and discussed in this work.

2 Funding

Part of this work was supported by the Federal Ministry for Economic Affairs and Climate Action of Germany for project “DigiBody” (Digital process chain for mapping and optimizing joining technologies in body-in-white constructions).

3 Literature

[1] Klöppel, T., Haufe, A.: “New material MAT_307: A viscoelastic-viscoplastic constitutive formulation to model adhesives during the complete manufacturing-crashworthiness process chain”, 16th LS-DYNA User Forum, Bamberg, Germany, 2022

[2] Matzenmiller, A., Kühlmeyer, P., Meschut, G., Aubel, T.: “ Methodenentwicklung zur Simulation des thermo-mechanischen Verhaltens von Klebschichten in hybriden Fügeverbindungen während des Aushärteprozesses”, Technischer Bericht zum Projekt P1087, FOSTA, 2018

[3] Burbulla, F., Matzenmiller,A.: “ Materialmodell für Klebstoffe aus duktil modifizierten Epoxidharzen (TAPO-Modell)”, Technischer Bericht zum Projekt P828, FOSTA, 2013

Experimental procedure and hardening model for simulation considering forming and baking effects

JiHo Lim[1], Haea Lee[1], Jisik Choi[1]

[1] POSCO

1 Abstract

This paper discusses the experimental methods and a constitutive equation for hardening to consider work hardening and bake hardening of automotive steel sheets in crash simulation. In general BH tensile test, the pre-strained specimen is baked and then re-tensioned without any treatment for the specimen. However, for giga-steels, the fracture occurs frequently in the curvature region due to differences in bake hardening due to non-uniform pre-strain in the parallel section and curvature regions. In this study, in order to prevent this curvature fracture, tensile specimens were re-machined in the uniform deformed region of large specimens subjected to pre-strain, and the re-machined specimens with uniform strength in all regions were re-tensioned. As the pre-strain increases, the strength of automotive steel tends to increase after baking. In giga-steel, shear band rupture often occurs as Lüders band does not propagate and concentrates in a tensile test with pre-strain of several percent or more. This makes it impossible to obtain a true stress-strain curve because the stress-strain curve shows the maximum load beyond the elastic region. To overcome this problem, a DIC Inverse Method was developed to calculate the true stress-strain curve by comparing the strain obtained from DIC images and load curves. This method can be applied not only where shear band fracture occurs, but also in necking fracture, and can be obtained true stress-strain curves for strains up to 2-3 times the uniform elongation. Using the DIC inverse method, true stress-strain curves after pre-strain and baking were obtained for several giga-steels. In order to apply the true stress-strain curve to the simulation, an appropriate hardening model is required. In this study, a new constitutive equation was developed considering the shift in tensile curve by baking and the increase in initial stress due to the yield point elongation. For newly developed model, the user material subroutine of LS-Dyna was configured, and the simulation was performed on the HAT specimens with pre-strain. When both work hardening and bake hardening were considered, there was a significant increase in absorbed energy compared to when only work hardening was considered.

Inductive and RadioFrequency (RF) heating in LS-DYNA for medical and other industrial applications

Iñaki Çaldichoury, Pierre L'Eplattenier, Trang Nguyen

Ansys Inc.

1 Abstract

Inductive and radiofrequency heating both rely on an electromagnetic power source to generate heat. However, they are based on different frequency scales that trigger different electromagnetic behavior and make some terms predominant over others. Inductive heating can be viewed as a “contactless” form of heating where a current source (typically a copper coil) with a frequency in the range of KHz or MHz approaches another conductor thus triggering induced currents (Eddy currents) in nearby conductors which can generate heat, depending on the material’s properties (resistivity, permeability). Radiofrequency heating, on the other hand, can be viewed as an extension of traditional Resistive heating where an electrode is plugged between two ends of a specific material. Contrary to resistive heating, the material’s electrical conductivity is usually very low, or the material can be an insulator, but the input source is in a high frequency range (GHz or higher) which triggers molecular displacements that generate heat via friction. This dielectric heat source term becomes the dominant factor rather than the Ohmic losses term. Inductive and Radiofrequency heating are both present in medical applications and other industrial applications. For example, RF heating of body tissues is used for muscle therapy and at higher temperature to kill tumors and cancer cells. On the other hand inductive heating is present in various domains, from sterilization procedures (Needle and surgical instruments heat treating) to induction casting of dentures and medical implants. This talk will give an overview of LS-DYNA’s capabilities, describe existing features as well as introduce the recent advancements that extend the scope of applications. It will also discuss which pitfalls to avoid and focus on specific items to keep an eye on.

Some New Features of the Dual CESE solver in LS-DYNA and its applications

Grant Cook Jr[1], Zeng-Chan Zhang[2]

[1]LST, an ANSYS Company, Livermore CA 94551, USA
[2]LST, an ANSYS Company, Livermore CA 94551, USA

Abstract

In this paper, we will briefly review the Dual CESE solver, that is an improved version of the regular CESE solver. For instance, compared to the regular CESE solver, it is more accurate and stable, and particularly more stable for triangle (2D) /tetrahedral (3D) meshes, all while maintaining the core features of the CESE method. Some of the new features of the Dual CESE solver in the R15 release will then be explained. Probably the most significant new capability is the addition of the multiphase phase-change solver to the suite of dual CESE solvers. An implementation of a “point-source” method of injecting a gas flow inside the volume of the fluid mesh is also new in R15. Several examples will be given showing how to use the new capabilities of this Dual CESE solver. Also, some significant multiphase capabilities that were new with the R14 release of LS-DYNA are also demonstrated. Other features include several different kinds of time-history outputs from the dual CESE solver that are accessed through the binout mechanism in LSPP. The most recent such feature in R15 is the binout time history interface for the plotting of drag, lift, and related variables; this is now supported in LSPP4.11. LSPP4.10 also supports the time history binout output from all the Dual CESE multiphase solvers, including the new (R15) phase-change solver. An important upgrade for the R15 version of LS-DYNA is a more robust material erosion FSI capability for all dual CESE solvers. Finally, new in R15 are boundary conditions for the single-phase Dual CESE solvers that have been introduced to help with aerospace-type applications.

FSI Simulations with LS-DYNA ICFD Solver: Capabilities, and Best Practices

Satish Kumar Meenakshisundaram[1], Facundo Del Pin[1]

[1]Ansys

Abstract

This document presents an overview of the LS-DYNA ICFD (Incompressible Computational Fluid Dynamics) solver and best practices for its efficient use in solving complex Fluid-Structure Interaction (FSI) problems from a user's perspective. LS-DYNA ICFD solver is a powerful tool for simulating the interaction between fluids and solid structures, allowing for accurate predictions of real-world phenomena. This paper will cover the underlying principles of the ICFD solver, its unique features, and practical applications, along with tips for efficient use. The content presented is crafted to ensure that engineers, regardless of their level of experience with FSI simulations, can easily understand and benefit from it. This document aims to provide a clear and concise guide for researchers and engineers who seek to utilize LS-DYNA ICFD solver for their FSI simulations, enabling them to achieve the expected throughput and save time in the process.

Key Findings of Surveys on Cloud Computing for Engineering Simulation

Wim Slagter[1]

[1]ANSYS, Inc.

Summary

In this presentation, we will unveil the key insights obtained from recent research studies on cloud-enabled simulations. These studies were conducted through surveys involving over 800 engineers, engineering and IT managers, as well as C-level executives. Beyond the overarching findings, you will gain a deeper understanding of the challenges, driving factors (including financial ROI), methods for accessing cloud resources, and best practices for implementing and expanding a cloud computing initiative.

Optimally Solving Ansys LS-DYNA Models: The Value of Ansys Gateway Powered by AWS

Ramin Torabi[1], Dr John Baker[2],

[1]Amazon Web Services+ AWS Privacy (amazon.com)
[2]Ansys + Privacy Notice: Information Usage and Safety | Ansys

1 Abstract

In the realm of engineering simulations, accuracy, speed, and scalability are paramount. Recognizing these requirements, Ansys, AMD with their EPYC™ processors, and Amazon Web Services (AWS) have collaborated to deliver an extraordinary performance boost for engineering simulations using Ansys LS-DYNA on Ansys Gateway powered by AWS.

The collaboration between Ansys, AMD, and AWS delivers enhanced scalability by harnessing the power of AMD EPYC™ processors on Amazon EC2 instances. With the ability to scale up to thousands of cores, engineers can tackle simulations that were previously challenging or time-consuming. This scalability empowers engineers to push the boundaries of their simulations, enabling the exploration of complex phenomena and capturing accurate real-world behavior.

Join this session to learn more about the cloud solution provided and benchmarks associated from two recent studies conducted with Ansys, AWS and AMD.

2 References

- Optimally Solving Ansys LS-DYNA Models: The Value of Ansys Gateway Powered by AWS | Ansys White Paper
- LS-DYNA with AMD EPYC on Ansys Gateway powered by AWS
- Ansys LS-Dyna On Amazon EC2 HPC6A Instances Featuring AMD EPYC 7003 Series Processors

- 3rd Gen AMD EPYC™ processors deliver outstanding scale-out performance running Ansys® LS-DYNA® on one to four Amazon Web Services Hpc6a instances.

Exploring the Potential of ARM Processors: Evaluating LS-DYNA Performance for Cloud-Based High-Performance Computing

Eric Day

Ansys LST

1 Abstract

In the realm of high-performance computing (HPC), x86_64 architecture has traditionally dominated, driven by its robust performance and extensive software support. However, recent benchmarks indicate the emerging viability of ARM processors for compute-intensive workloads, particularly when running LS-DYNA software. This study explores the performance of LS-DYNA on ARM-based chips, specifically evaluating its effectiveness on Amazon Graviton in the HPC cloud environment and Apple M, Cavium ThunderX2, Ampere Altra, Fujitsu A64FX and Amazon Graviton in standalone computing. Power efficiency, high throughput, cost-effectiveness, and scalability position ARM processors as compelling options for cloud-based LS-DYNA computations.

Unlocking the Potential of Cloud Computing for High-Performance LS-DYNA Simulations: Benefits of Scalability, Flexibility, and Cost-Effectiveness with Gompute

Iago Fernandez, Gompute

1 Abstract

The computational demands of LS-DYNA simulations have increased significantly in recent years, necessitating high-performance computing (HPC) resources that may not be readily available to all users. Cloud computing platforms offer an attractive alternative by providing access to flexible and scalable computing resources, allowing LS-DYNA users to optimize their simulations' performance and reduce time-to-solution.
With Gompute, users can scale their LS-DYNA simulations up or down as needed, without needing to invest in additional hardware or infrastructure. This allows users to tackle larger, more complex simulations that may have been too costly on traditional on-premises computing resources.

Additionally, the cloud provides greater flexibility for LS-DYNA users, as they can easily access and use the software from any location with an internet connection. This means that users can work remotely, collaborate with colleagues across geographic boundaries, and easily access the latest updates and versions of LS-DYNA.

Gompute also offers cost-effectiveness benefits, as users can only pay for the resources they consume, or have dedicated resources when workloads are intensive. This can be especially valuable for smaller organizations or individual researchers who may not have the resources to invest in high-end computing equipment as well as for enterprises that look for an outsourced HPC service for the long term.

An Interprocess Communication based Integration of AI User Materials into LSDYNA

Joachim Sprave[1], Tobias Erhardt[2], André Haufe[2]

[1] Mercedes-Benz AG
[2]DYNAmore GmbH

1 Abstract

Machine Learning driven material models have been investigated for some time now. ML models can be trained outside of solvers by means of strain paths and corresponding results which have either been recorded from simulations, drawn from distributions, or even measured from hardware tests. The trained models can be easily be evaluated regarding their performance based on another set of strain paths with results that have not been presented to the model during training. When a model has reached a promising prediction accuracy on the validation data, the natural next step is to test it in an FEM simulation by integrating the trained model as a user material. Unfortunately, coupling ML models trained by established AI frameworks such as Tensorflow or PyTorch, is not a trivial task. Solvers are typically stand-alone programs written in a compiled language such as Fortran or C/C++. For LSDYNA, there are 3 canonical ways to use trained ML material models as user materials: 1. Reimplementation as a UMAT in Fortran or C/C++. This requires a deep insight in models used in training which can be very large and complex. Reimplementation is time-consuming and error-prone, but it is also hard to beat performance-wise. Actually, this is probably the way to go for models which finally go into production realeases. 2. Coupling of LSDYNA with a framework. At least for Tensorflow and PyTorch, there exist so-called bridges which allow to call trained models from within Fortran routines such as LSDYNA UMATs. It requires some coding and a careful compiler and linker setup, but once this is done, almost every model trained in the framework can be used from within a UMAT with almost native performance. Still, this has to be done for each framework separately, and differerent restriction apply for each framework. 3. Interprocess communication (IPC). A dummy UMAT forks a process that communicates with a process written in a scripting language such as Python. Within the Python script, arbitrary frameworks can be used, including, but not restricted to, PyTorch, Tensorflow, and Scikit-Learn. One can even write traditional material models based on continuum mechanics and test them without recompiling and relinking LSDYNA. Compared to 1. And 2., this approach is hopelessly bad regarding performance. On the other hand, a trained model as well as a hand-written material model can be made available for LSDYNA simulations in a couple of minutes. In this work, we present an IPC UMAT for LSDYNA on Unix systems. A dummy UMAT is provided that handles the communication, as well as boiler-plate code in Python that serves as a wrapper to call a trained model for each integration point or batch of integration points, mimicking the logic of simple and vectorized UMATs in Fortran.

On Accelerating Elastoplastic Material Models with Machine Learning

G. Bokil[1], D. Koch[2], André Haufe[2], Holger Steeb[3]

[2]DYNAmore GmbH
[1,3]University of Stuttgart

1 Abstract

This paper presents a hybrid Machine Learning (ML) approach in accelerating elastoplastic material models used in Finite Element (FE) software. As opposed to a holistic approach which aims to replace the material model entirely with an ML model, the hybrid approach replaces a part of the model. The research focus was set on the commonly used von Mises plasticity with nonlinear isotropic and kinematic hardening models for steels in which a return mapping algorithm is employed for computing the stress state from the strain state and material parameters. The idea was to replace the expensive iterative plastic correction step in the return mapping algorithm with a ML-model. This model would essentially predict the plastic state from the elastic or trial state. Moreover, the model should be robust enough to be employed for a range of materials. For this purpose, a Feedforward Neural Network (FNN), a LASSO regression model, and a sparse nonlinear regression model were investigated. It was found that the ML-models could predict the strain increment only from the principal trial stresses and the unit outward normal to the yield surface. To ensure generalization, the models was trained on loading, unloading and cyclic loading scenarios for numerous hardening parameters. Above all, the sparse nonlinear regression model, inspired by SINDy, displayed better computational performance and was thus utilized in the final algorithm. As a result, a novel hybrid von Mises hardening plasticity algorithm was formulated by replacing the iterative plastic projection with a Sparse Identification of Plastic Strain-increment (SIPS) model. The SIPS-based material model was then programmed for LS-DYNA via the User Defined Material to carry out benchmark FE simulations. Notably, the performance analysis on standard tests and FE simulations concluded that the SIPS-based hybrid model reduced the computational cost by 40% and achieved up to 95% accuracy in reference to the classical model.

Using Data from Physical Experiments to Train Machine Learning Material Models

Daniel Sommer[1], Pauline Böhringer[2], Markus Stoll[3], Peter Middendorf[1]

[1] University of Stuttgart, Institute of Aircraft Design, Germany
[2] Mercedes-Benz AG, Research and Development, Sindelfingen, Germany
[3] Renumics GmbH, Karlsruhe, Germany

1 Abstract

Structural analysis of mechanical components, such as predicting the deformation behaviour of sheet metal or assessing the crash safety of a vehicle, typically relies on finite element analysis (FEA). One critical aspect influencing the quality of these simulations are the material models that describe the relationship between strains and stresses. However, the development and selection of the most appropriate models is a significant challenge that involves costly and time-consuming testing and calibration procedures.

Recently, data-driven material models based on machine learning (ML) methods, such as artificial neural networks (ANNs), have shown the potential to substitute classical analytical models. They promise fast computation, a high level of flexibility and thus the adaptability to new materials [1,2].
However, these ML material models are usually calibrated to a certain material with training data obtained from simulations and still require the classic analytical models. The stress outputs to given strains are needed in this supervised machine learning task because the stress field cannot be measured from materials testing. The present learning approach merely produces an alternative ML-based representation of the existing classical material models.

To circumvent the need for a classic material model, we present a methodology to train artificial neural networks using exclusively data available in physical experiments [3]. Instead of using a classic material model to predict stresses, we use only physical equations based on the global reaction force and the measurable displacement field on the surface of the test specimen to train the ANN. To verify comprehensive coverage of the required strain input space, we employ data-centric methods. This strategy contributes to improving the robustness and accuracy of our machine learning models, which will lead to a more efficient approach to ML material model development.

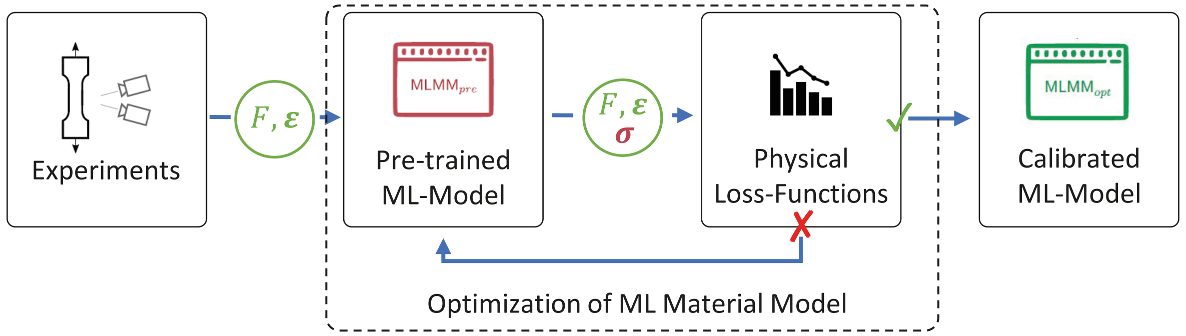


Fig.1: Method of calibrating an ML material model to a new material using data from experiments.

2 Literature

[1] Ghaboussi et al. Knowledge-based modeling of material behavior with neural networks. *Journal of Engineering Mechanics*, **1991**, *117*, pp. 132-153.

[2] C. Bonatti One for all: Universal material model based on minimal state-space neural networks. *Science Advances*, **2021**, *7*.

[3] P.Böhringer, D.Sommer et al. A strategy to train machine learning material models for finite element simulations on data acquirable from physical experiments. *Computer Methods in Applied Mechanics and Engineering*, **2023**, 406, pp. 1-19.

FSI simulations to study eye biomechanics during a Non Contact Tonometry

Elena Redaelli[1], Begoña Calvo[1,2], José Felix Rodríguez Matas[3], Giulia Luraghi[3], Jorge Grasa[1,2]

[1] Aragón Institute of Engineering Research (I3A), Universidad de Zaragoza, Spain
[2] C de Invest Biomecánica en Red en Bioingenieria, Biomateriales y Nanomedicina (CIBER-BBN)
[3] Dept. of Chemistry, Materials and Chemical Engineering "Giulio Natta", Politecnico di Milano, Italy.

Understanding the corneal mechanical properties has great importance in the study of corneal pathologies and the prediction of refractive surgery outcomes [1]. Non-Contact Tonometry (NCT) is a non-invasive diagnostic tool intended to characterize the corneal tissue response in vivo by applying a defined air-pulse. The development of a strong FSI tool amenable to model the NCT, applied to different structural and anatomical configurations, provides the basis to find the biomechanical properties of the corneal tissue in vivo [2]. This paper presents a high-fidelity finite-element model of a patient-specific 3D eye for in-silico NCT. A fluid-structure interaction (FSI) simulation is developed to virtually apply a defined air-pulse to a patient-specific eye model comprising cornea, limbus, sclera, and humors. [3] Three different methodologies are tested to model the humors and the best approach is chosen. Then, a Montecarlo simulation is performed varying both the parameters describing the mechanical behaviour of the corneal tissue and the IOP. The analysis reveals that the mechanical properties of the corneal tissue and the IOP are perfectly coupled. A stiffer material with a low IOP can give the same deformation result on the cornea as a softer material with an higher IOP.

[1] Esporcatte, L. P. G., et al., "Biomechanical diagnostics of the cornea", Eye and vision 7.1 (2020): 1-12.
[2] Ariza-Gracia, MÁ, et al., "Coupled Biomechanical Response of the Cornea Assessed by Non-Contact Tonometry. A Simulation Study", PLoS ONE 10(3): e0121486, 2015.
[3] Redaelli, E., et al., "A detailed methodology to model the Non Contact Tonometry: a Fluid Structure Interaction study", Front. Bioeng. Biotechnol., 10:1826, 2022.

This project has received funding from the European Union's Horizon 2020 research and innovation program under the Marie Skłodowska-Curie grant agreement No 956720.

Ergonomic optimization of rowing seats using personalized Human Body Models

Manuela Boin

Ulm University of Applied Sciences / Technische Hochschule Ulm

1 Abstract

Seat-related discomfort and health problems, which occur especially during long rowing tours or training sessions, can be reduced by rowing seats with a surface geometry that is ergonomically optimized for the particular rower. This seat optimization can be done by analyzing measured pressure distributions and modifying the standard seat surface geometry for a specific person based on these results using CAD tools. The project presented here focuses on the purely virtual development of the optimal geometry for specific rowers. FE simulations were performed using Human Body Models (HBMs) to define seat geometries for specific individuals.

Human body models were personalized using the scaling module of the PIPER toolbox, the ANSUR database and anthropometric measurements of male rowers of the Ulmer Ruderclub Donau e.V.. Metadata for THUMS 5 version 5.0.3 was developed for this virtual process. For the GHBM M50-0 version 4.5, publicly available metadata was used. A scaling procedure was defined and the results of the scaling process were compared for the two different baseline models.

Finally, these different personalized HBMs were used to define user-specific seats for the rowers through simulation. The resulting seat geometries were compared for different rowers.

2 Methods

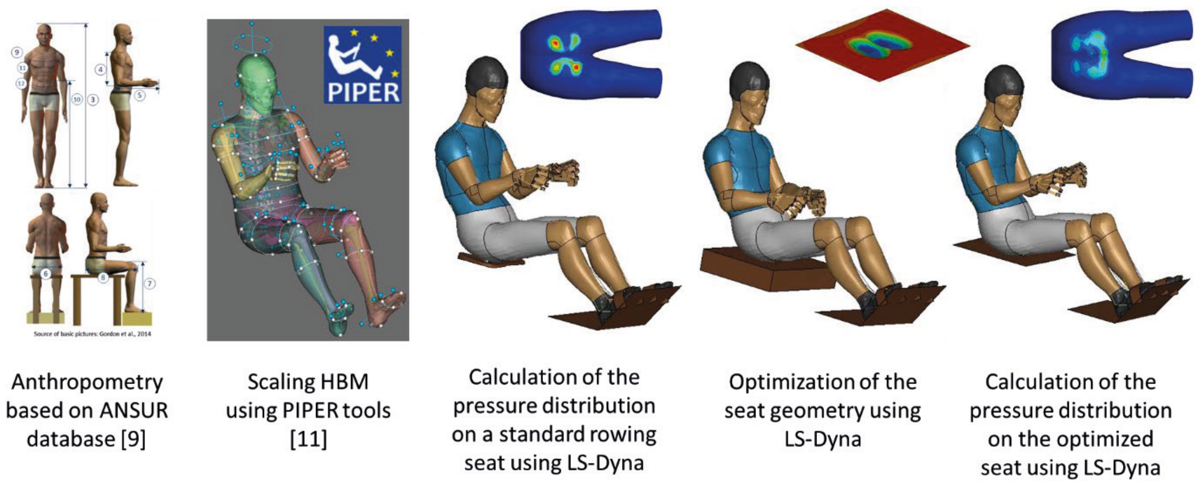


Fig.1: Overview of the five different steps in the seat optimization process

The project required the development of metadata for scaling THUMS Version 5 AM50 with the Scaling Constraints module of PIPER. The anthropometric landmarks, skin and bone entities were defined and the Simplified Scalable Model was set up with the definitions of the segments and sections for scaling with the ANSUR database.

3 Summary

Metadata for the Scaling Constraints Module of PIPER has been set up for THUMS 5.0.3 AM50, including landmarks, segments, sections, skin and bone entities. Personalization of THUMS AM50 and GHBM M50-O was successfully performed on four different individuals using a workflow developed for the PIPER framework. The scaled models were compared. The selected anthropometries could be well reproduced with both baseline HBMs.

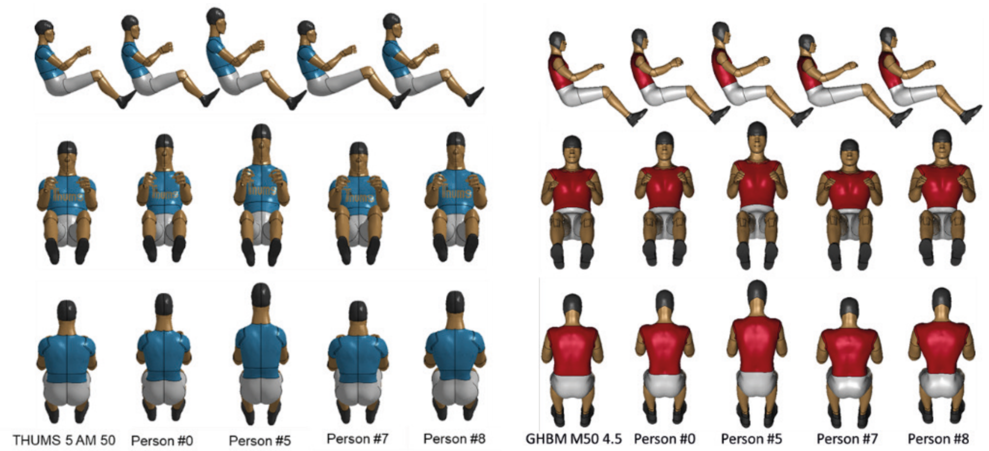


Fig.2: Side, front and rear view of the original HBMs and the scaled models. Left side: THUMS. Right side: GHBM.

All original and scaled HBMs were used to analyze the interface pressure distribution and buttock contact area on a standard rowing seat. Pressure peaks due to the non-optimal seat surface were found.

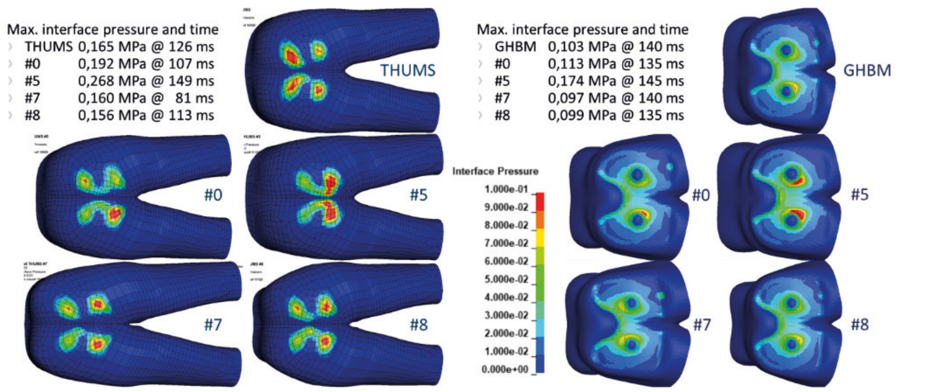


Fig.3: Distribution of the interface pressure at the buttocks for the different models on the standard rowing seat. All plots use the same scale from 0 MPa (blue) to 0,1 MPa (red). The back of the models is on the left. Left side: THUMS. Right side: GHBM.

Personalized seat surfaces were created for each original and scaled HBM. The interface pressure distributions and contact areas were calculated for these optimized seats and compared to the corresponding models with the standard seat. The maximum interface pressure was significantly reduced for all models.

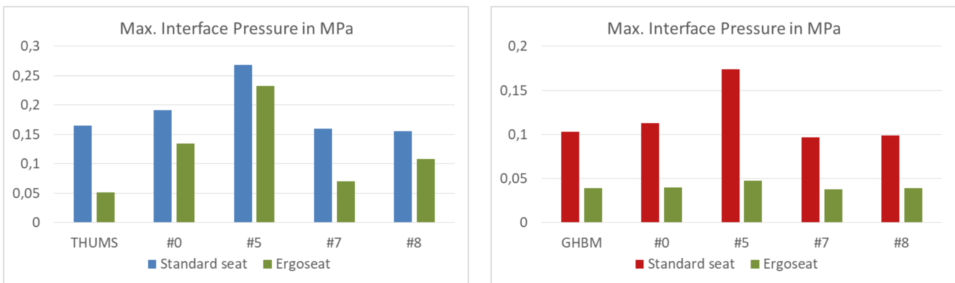


Fig.4: Comparison of maximum interface pressure for all models for the standard seat and the personalized seats. Left side: THUMS. Right side: GHBM.

The contact area between the buttocks and the seat was increased. Differences in the pressure distributions caused by the different anthropometry of the subjects could be reproduced. Systematic differences due to the different refinements and material models of the buttocks of the two different baseline HBMs were found.

Biomechanics II

Anatomically accurate finite element model of a human head for crash applications

Alberto Tacchi, Ivan Colamartino[1], Gabriele Canzi[2], Giorgio Novelli[3], Marco Anghileri[4]

[1] Department of Mechanical Engineering, Politecnico di Milano, Italy
[2] Maxillofacial Surgery Unit, Emergency Department, ASST Grande Ospedale Metropolitano Niguarda
[3] O.U. Maxillofacial Surgery, Department of Medicine and Surgery, School of Medicine,Fondazione IRCCS San Gerardo dei Tintori Hospital, University of Milano-Bicocca
[4] Department of Aerospace Science and Technology, Politecnico di Milano, Italy

As occurrence of fractures during head impacts becomes an increasingly important topic within road safety traumatic events, accurately knowing the real severity of head traumas via numerical simulations would lead to improvements of safety and protection devices and better understanding of any necessary surgical treatments. However, state-of-the-art finite element models of the human body for crash applications, commonly developed targeting simplicity and robustness, are not able to accurately describe fracture patterns nor predict minor or local injuries.
Therefore, the present study focused on the development of a geometrically and anatomically accurate Finite Element Head Model (FEHM), implementing a local strain-based failure model with geometry derived from computerized tomography (CT) scans. Advanced constitutive models have been employed, as well as high-quality solid meshes of the skull for accurate strain description and for injuries and fracture patterns prediction. The model is validated through a multiple-step simulation process based on experimental tests on human cadavers. Good correlation reported in such tests showed the ability of the new FEHM in describing and predicting sustained skull fractures, by means of both global and local injury criteria. Subsequently, the validated model was used in a preliminary accident reconstruction for the analysis of a real-world impact event.

Implementation of Incompressible Smooth Particle Galerkin Method in numerical analysis of adhesive process

Pulkit Rana[1], Benjamin Eble[1], Ulrich Mayer[1], Thomas Schall[1], Wilfrid Polley[1]

[1]Mercedes-Benz AG

Abstract. Adhesive bonds are becoming increasingly important in automotive industry due to their various advantages like ability to join dissimilar materials and weight reduction. Components that previously had to be joined via bolts or welding can now be bonded. In order to be able to meet the high demands placed on the connections, some of which are also safety-relevant, modeling by means of numerical methods is the obvious choice. Various numerical methods are available to model the adhesives for example: Eulerian Finite Difference Method or Finite Volume Method. Since the transport effect can be bypassed in a Lagrangian approach, thus minimizing numerical diffusion, Lagrangian particle methods are becoming increasingly popular. The Incompressible Smoothed Particle Galerkin (ISPG) method is one of these methods and therefore used in this research to model the adhesives. The adhesive is placed in a wavy line on a surface to be covered. With the aid of a stamp, the adhesive is compressed to a certain height and thus distributed over the surface. The aim of the research is to cover the surface as evenly and completely as possible over the remaining gap. The simulations are set up, calculated and evaluated in the simulation software LS-DYNA. In the end, adhesive simulations are compared with the experimental results and a good correlation is found between the simulation and experiments.

Keywords: Finite Element Method, Simulation, Bonding, Joining.

Comparison of Polyurethane and Epoxy Adhesive High Strain Rate Performance Using Cohesive Zone Model

Devon Downes, Manouchehr Nejad Ensan, Lucy Li

National Research Council Canada

1 Abstract

It is known that the ballistic performance of ceramic composite personnel armour is highly dependent on the thickness of ceramic and backing material. Recent studies have begun focusing on the effect of adhesive bonding between the ceramic and the backing plate, because failure of the adhesive layer can cause separation between the ceramic and backing. This debonding between substrates causes the ceramic to underperform by shattering early due to an imperfect transmission of the stress wave to the backing material. Given that the adhesive plays such an important role in armour it is important to better understand the underlying physics.

NRC has an ongoing simulation activity to investigate the behaviour of polyurethane and epoxy adhesives as a bonding material using cohesive zone elements (CZE). The objective is to assess the ability of CZE to replicate the adhesive’s response and compare with experimental data. Polyurethane is known to a have high strain to failure ratio, ranging between 100-350%, however its tensile strength is approximately ~2.0-5.0MPa. Epoxy on the other hand while brittle in nature with a strain to failure ratio of 2%, has a high tensile strength of approximately 50MPa. There is concern that the ability of the adhesive layer to endure large amounts of strain and its stiffness would negatively impact the damage experienced by the ceramic. Another important factor is the acoustic impedance of the adhesive which determines the magnitude of the stress wave transmitted from the ceramic to the backing material. The larger the impedance difference between the ceramic and adhesive, the lower the magnitude of the stress wave transmitted into the backing and greater the stress wave transmitted back into to the ceramic at the bond line interface. Given that ceramics fail in tension, a larger tensile wave is expected to cause more damage in the ceramic. Moreover, a study of the capture of the ceramic failure between a boundary condition of perfect transmission and an imperfect transmission has been presented in this paper. The study showed that the ceramic bonded to aluminum with a perfect transmission boundary experiences a more dispersed damage pattern, when compared to the ceramic bonded to aluminum with an imperfect transmission boundary which experiences a more localized damage pattern.

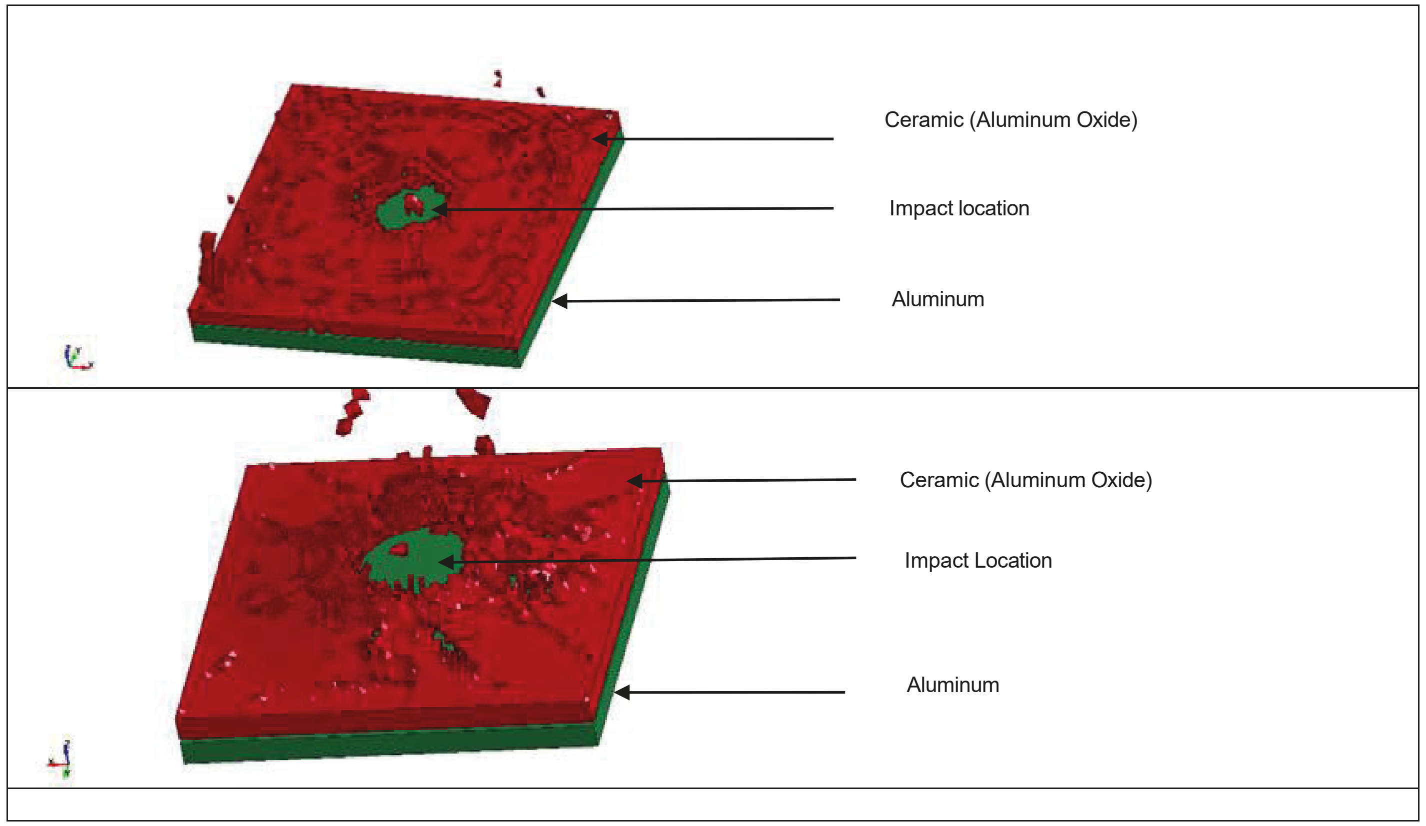


Fig.1: Comparison of Ceramic Damage between a Perfect Transmission Boundary and an Imperfect Transmission Boundary

Adhesives

Characterization of a cohesive zone model for adhesives with *MAT_240 and curve mapping method in LS-OPT

Nicole Betz[1], Martin Holzapfel[1], Tobias Behling[1], Mathieu Vinot[1], Nathalie Toso[1]

[1]German Aerospace Center, Institute for Structures and Design

Keywords: *MAT_240, LS-OPT, cohesive zone model, curve mapping

1 Abstract

The importance of adhesives in automotive structures exposed to high crash loads has increased over the years. To improve the structural sizing, it is necessary to predict the behavior of bonded joints under dynamic impact and crash loads. The present work aims at developing an optimization scheme with LS-OPT for the effective numerical characterization of adhesive layers in Finite-Element simulations as a support to cost-intensive experimental testing. A cohesive zone model approach with the material card **`*MAT_240_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE`** [1] is used to represent the behavior of the adhesive layer. The present work focuses on a curve mapping process with LS-OPT for the automated identification of parameters for **`*MAT_240`**.

First, a parameter optimization scheme based on two fracture mechanical tests for Mode I and Mode III loading is implemented (see Fig.1). As a basis for the optimization, a sensitivity analysis is previously performed to determine the decisive parameters in fitting the numerical to the experimental results and reduce the optimization space. In the next step, an optimization loop for the fracture mechanical tests will be investigated. Therefore, the least square method for the curve mapping process is used for the parameter optimization of with LS-OPT. The determined optimized parameters strength and fracture energy are being used to optimize the material cards on coupon level afterwards. The cohesive zone parameters are being calibrated to the experimental results and the parameter space can be minimized using the Sequential Response Surface Method (SRSM) [2].
The present contribution will show a method for optimized material cards with LS-OPT for adhesive layers.

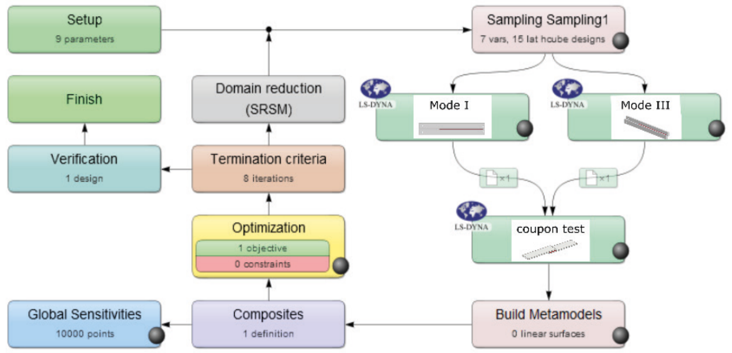


Fig.1: LS-OPT process chain with curve mapping method for adhesive layers

2 Literature

[1] Stephan Marzi, Olaf Hesebeck, Markus Brede, Felix Kleiner. A Rate-Dependent Elasto-Plastic Cohesive Zone Mixed-Mode Model for Crash Analysis of Adhesively Bonded Joints; 2009.

[2] Stander, N.; Basudhar, A.; Liebold, K.; Roux, W.J., Goel, T., Eggleston, T. and Craig, K.J., LS-OPT User's Manual, Version 7.0, Livermore Software Technology Corporation, November 2020

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Fluid added mass modeling in LS-DYNA and its application in structural vibration

Yun Huang, Tom Littlewood, Francois-Henry Rouet, Zhe Cui, Ushnish Basu

Ansys, Inc.

Abstract

For the ship building industry, it is a common requirement to model the added mass (virtual mass) of the water around the hull in vibration analysis. This actually starts from eigensolution of the structures. Instead of a “dry” eigensolution, a “wet” eigensolution is needed in order to include the influence on structural eigenmodes from surrounding water.

To meet this requirement, a fluid added mass computation has been implemented in LS-DYNA and an efficient approach to include the fluid added mass in structure modal analysis and vibration analysis has been developed. The fluid added mass is computed using a boundary integral formulation and is represented by a block low rank approximation to reduce the memory and CPU cost; a matrix-free LOBPCG eigensolver is adopted to speed up the eigensolution for the structure-fluid coupled system. Using this “wet” eigensolution, a variety of vibration analysis can be performed, including FRF (Frequency Response Function), SSD (Steady State Dynamics), random vibration, DDAM (Dynamic Design Analysis Method), and transient modal dynamic analysis.

This paper presents the workflow of this approach and introduces the necessary keywords to drive this solution. Some examples are included for the purpose of illustration and validation.

NVH / Implicit

New Eigensolver Technology for LS-DYNA

Roger Grimes, Francois-Henry Rouet

Ansys LST

Abstract

The Linear Algebra Team of ANSYS LST has added two new eigensolver technologies to LS-DYNA. The first is based on the Locally Optimal Block PCG algorithm. It uses an approximate factorization of a linear combination of the mass and stiffness matrix coupled with an iterative procedure to compute eigenvalues. It is usually more efficient than the standard Block Shift and Invert Lanczos implementation to compute a small number of eigenmodes.
The second, which we are calling Fast Lanczos, is an implementation of the Block Shift and Invert Lanczos algorithm designed to compute thousands of eigenmodes with less computing resources than the standard implementation. It is designed to be competitive with eigensolvers from competitors that are based on the Automatic Multilevel Substructuring algorithm (AMLS). The benefit of this new implementation is to achieve 2 ot 3 digits of accuracy for the computed eigenmodes deep into the eigen spectrum. This is less accuracy than the standard implementation but far better than the accuracy provided by the AMLS algorithm. This implementation is also designed for distributed memory parallel computing for superior scaling on modern computing architectures. Implementations of the AMLS algorithm tend to be only for shared memory parallelism with limited scaling. The target application is the computation of eigenmodes for analyses Noise-Vibration-Harshness by the NVH engineer. The conference proceedings will supply performance results for both new eigensolver technologies using several automotive models provided by customers in the range of 2 to 10 million elements. The results will be contrasted with the standard Lanczos implementation to help demonstrate when to use the various eigensolvers. The computer used for the testing is a “fat” workstation based on a 2 socket installation of Xeon Platinum CPU chips, each with 24 cores, for a total of 48 cores. This machine is not a high-end compute server and costs less than a new automobile.

© 2023 Copyright by DYNAmore GmbH, an Ansys Company

Speeding Up LS-DYNA Implicit with Mixed Precision, Low Rank Approximations, and Accelerators

Jason Cong[1], Florent Lopez[2], Robert Lucas[2], Francois-Henry Rouet[2], Linghao Song[1]

[1]UCLA
[2]Ansys Inc.

Abstract

The run time of LS-DYNA implicit analyses tends to dominated by the default multifrontal linear solver, whose complexity will range from O(N1.5) to O(N2), depending on the model. This talk will give an overview of attempts to reduce the run time of solving large systems of linear equations, both on the host processor as well as with accelerators. On the host, one can now mix single with double precision arithmetic, to reduce storage and increase the computational rate. Low-rank approximation takes this further, replacing off-diagonal blocks of the matrix with the product of two, much smaller matrices. In both cases, the multifrontal direct solver is converted into a high-quality preconditioner for an iterative solver. Meanwhile, Graphical Processing Units (GPUs) are once again of interest, as they offer greater arithmetic processing power and memory bandwidth than their host processors. Like GPUs, today's Field Programmable Gate Arrays (FPGAs) also have High Bandwidth Memory (HBM), and hence are candidates for memory bandwidth constrained algorithms such as Preconditioned Conjugate Gradients.

A standardized and automated method for creating material and failure models for metals

Suri Bala[1], Anantharam Sheshadri[3] , Hamid Keshtkar[3] ,Paul Du Bois[2]

[2]Forming Simulation Technologies, Northville, USA
[1]D3VIEW, Rochester Hills, USA

[3]Stellantis, Auburn Hills, USA

Abstract

In the 2018 16th international LS-DYNA user conference in Detroit a paper was presented by FCA that outlined a standardized way to develop material and failure models for metals. (/1/) In particular the method was based on a well defined and rigid testing program where the shape and size of the samples, testing conditions (such a boundary conditions, strain rates etc...) and format of documentation were fixed in close cooperation with the dynamic mechanics of materials laboratory at the Ohio State University. The purpose was to ensure repeatability, quality, comparability and fast accessibility of the test data. At this point in time however, the calibration of the numerical models remained essentially to be done 'by hand' and this constituted a tedious, rather boring and time consuming task for the engineers in the materials group.

The current presentation will summarize the results of our efforts to automate the calibration of the numerical models with the help of the D3VIEW software. The tedious task of Material calibration, involving several sequential/parallel steps,are captured in zero-code d3VIEW Workflows that provides a standardized method to calibrate a material. With several calibration-focused built-in functions that support data-processing, transformations, job-submission, machine-learning and optimization, the engineer is relieved of these elaborate tasks and can focus more on engineering a solution that provides the best material card for LS-DYNA. Over the years, several industrial materials have been actively calibrated and the d3VIEW platform is actively developed to address newer unforeseen challenges. These developments ultimately lead to a significant gain in lead time and (not least) an improvement of quality of life for the engineers in the materials group.

References

/1/ Experience with material and fracture modeling at Fiat Chrysler Automobiles (FCA), Dr. Anantharam Sheshadri, Hamid Keshtkar, Ashutosh Patil, Paul Du Bois[2 *] ,16th International LS-DYNA Users Conference, June 10-12, 2018, Dearborn, MI

Generic Material Database - From Tensile Test to FLC and a 3d failure model

Ingolf Lepenies
SCALE GmbH

DYNAmore GmbH, an Ansys Company, Industriestr. 2, D-70565 Stuttgart, Germany
Tel. +49 (0)711 - 459600 - 0, info@dynamore.de, www.dynamore.de

ISBN 978-3-9816215-7-0

Images courtesy of Baden-Baden Tourismus