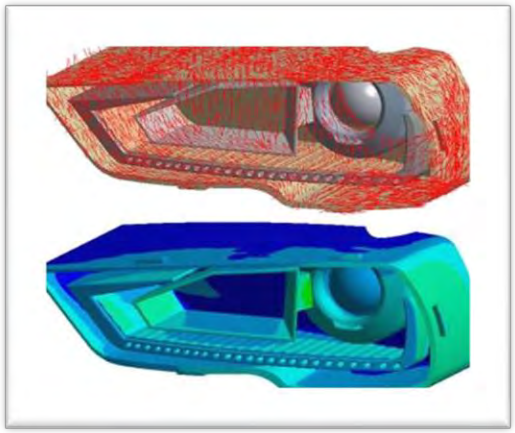
fea INFORMATION Engineering Solutions
www.lstc.com

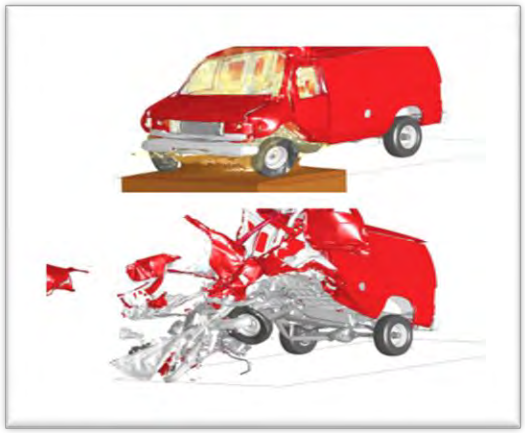
Ansys LST
www.ansys.com

Issue 2, February 2021
www.feapublications.com

ANSYS



LST



ESI Group

DYNAmore

LS-DYNA@ New Feature and Application

- **On Setting up Multi-Materials in S-ALE Models**
- **Explosion with LS-SYNA S-ALE & new features**
- **Overview of the CESE compressible fluid and FSI solvers**

FEA Information Engineering Solutions

www.feapublications.com

The focus is engineering technical solutions/information.

Livermore Software Technology, an ANSYS company

Development of LS-DYNA, LS-PrePost, LS-OPT,
LS-TaSC (Topology), Dummy & Barrier models and
Tire models for use in various industries.

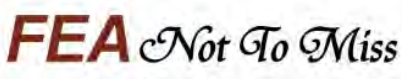
www.lstc.com

To sign up for the FEA News send an email - subject "subscribe" to news@feainformation.com
To be removed from the FEA News send an email - subject "Remove" to news@feainformation.com

If you have any questions, suggestions or recommended changes, please contact us.

Editor and Contact: Yanhua Zhao - news@feainformation.com

Platinum Particpants



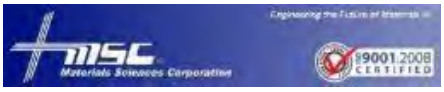
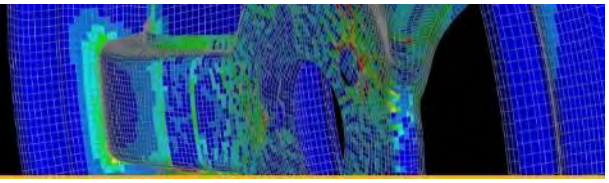


Table of contents

Ansys

Ansys Blog



Published on February 10, 2021 by Marisa Melchiorre
Structural Analysis
Ansys LS-DYNA, Ansys Mechanical, Structural Analysis, Ansys Sherlock

Top 3 Features in Ansys Mechanical 2021 R1

In 2021 R1, Ansys Mechanical continues to deliver features that enable faster simulations, easier workflows and scripting with recording capabilities, as well as product integrations that offer enhanced solver capabilities.What's new in Ansys Mechanical R1 2021 engineering simulation software

Let's look at the top three 2021 R1 features in Ansys Mechanical and how they will benefit users and improve the overall structural design process.

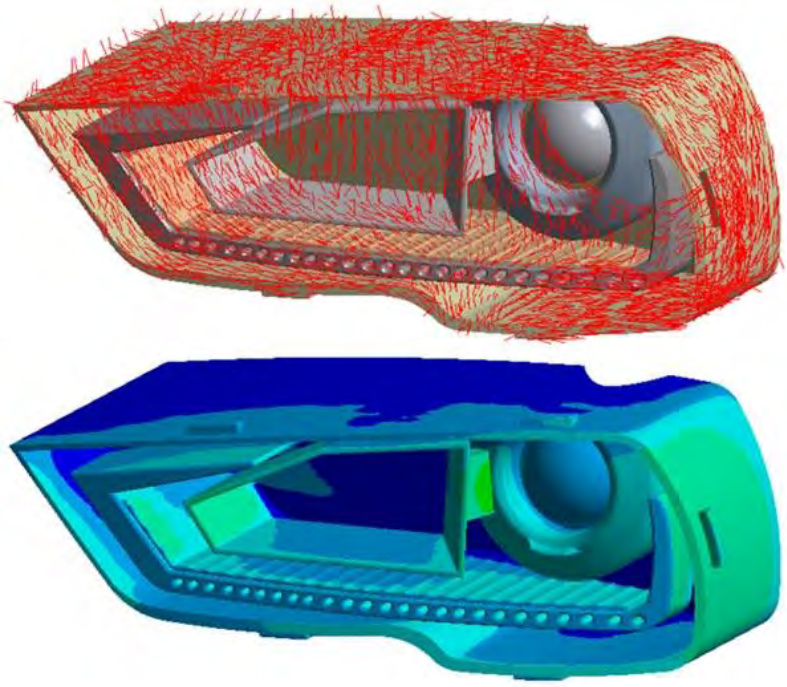
1. Short Fiber Reinforced Composites Workflow

Fiber reinforced plastics are everywhere, from consumer goods, automotive parts and airplanes to portable electronics. In 2021 R1, Mechanical enables engineers to:

- Accurately and easily simulate plastic parts made through injection molding
- Test for deformation of the plastic part
- Predict potential failure points of the part when used in a real-life scenarios

The injection process data that includes fiber orientation and density details are imported and mapped seamlessly onto a finite element mesh in Mechanical and complemented with accurate material behavior at each element. Moldflow, Moldex3D, Sigmasoft and Cadmould injection modeling software vendors are supported. Material data is captured through a multiscale analysis run with Ansys Material Designer to compute the complex plastic behavior. These computed materials can then be applied to whole parts or assemblies to accurately predict part performance ahead of any manufacturing or testing effort.

Ansys Mechanical in Workbench produces realistic analyses of new products designed with fiber-reinforced plastics, while considering real-life conditions. This results in an easier and quicker modeling process with no need for complex workflows.

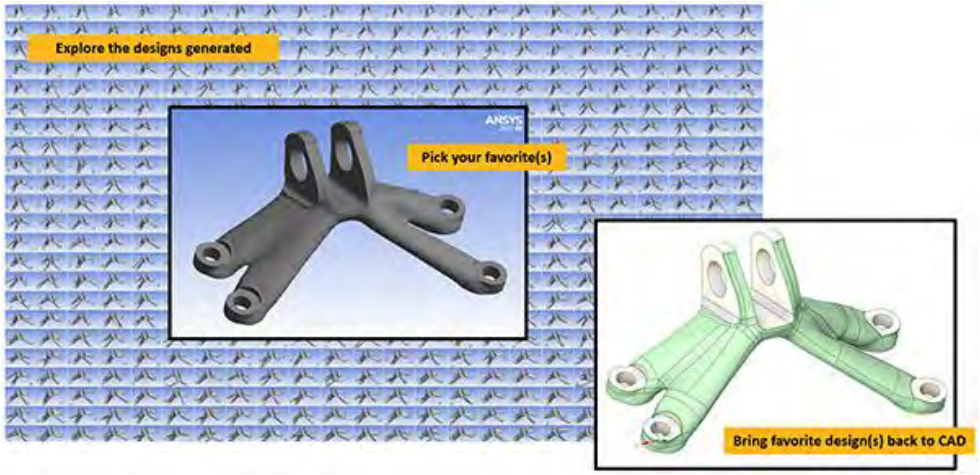


Ansys Mechanical can simulate fiber-reinforced plastic parts made through injection modeling processes such as automotive headlamps.

Register for the webinar: Ansys 2021 R1: Ansys Mechanical Update

2. Back to CAD

2021 R1 also brought the official release of the "Back to CAD" workflow within Ansys SpaceClaim. This new workflow allows users to turn their topology optimization results into computer-aided design (CAD) geometry. The "autoskin" feature saves the user time otherwise spent on recreating geometry. You can use the newly created geometry for validation simulations or continue on to prepare the part for additive manufacturing.

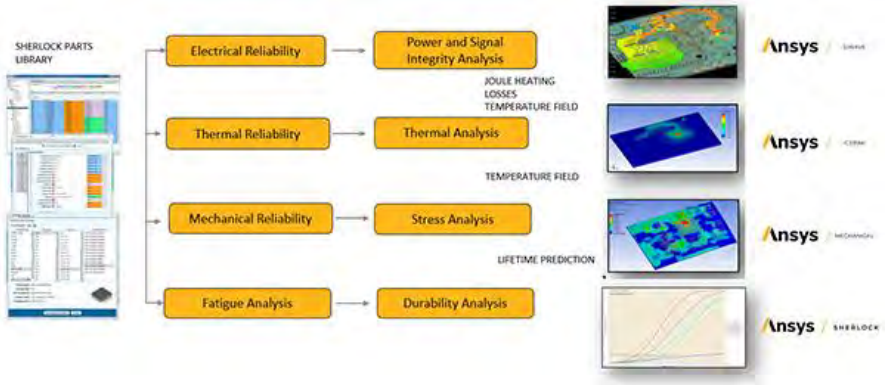


Ansys 2021 R1 enables you to explore thousands of designs, pick your favorite and then bring it back to CAD.

You can perform topology optimization in Mechanical using a range of tools and options there, including the level-set based solver. Ansys optiSlang can then be used to run many scenarios and options parametrically. You can also leverage Ansys Distributed Compute Services (DCS) to run large-scale evaluations and then bring the chosen candidates back to CAD with SpaceClaim. This workflow and toolset are the keys to generative design and unlock new design freedom.

3. Product Integration for Enhanced Electronics Reliability Workflows

Throughout each release, we continue to integrate products into Mechanical to enhance and expand workflows. In 2021 R1, the Ansys Sherlock plug-in in Workbench allows users to access Sherlock automated design analysis software and incorporate Mechanical's random vibration capability for ease of use and expanded capabilities within one interface.



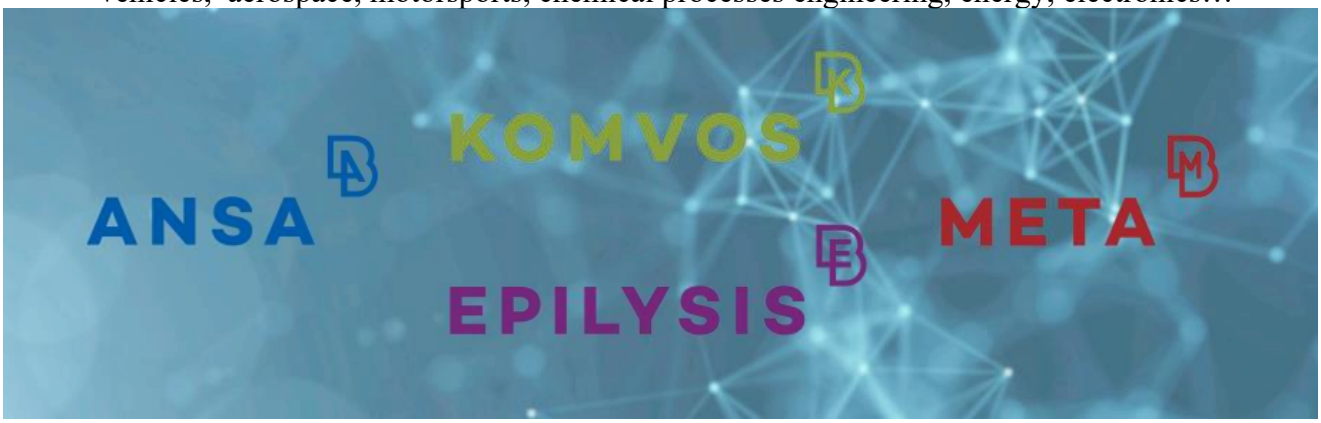
Ansys Electronics Reliability workflow using Ansys SIwave, Icepak, Mechanical and Sherlock.

Ansys LS-DYNA also continues to add solver capabilities to Mechanical software. Users can now perform electronic CAD trace mapping for analysis of printed circuit boards (PCBs) in electronics reliability workflows. Trace mapping is an approach that maps the PCB information from the ECAD file onto the mesh to simplify modeling. It can be applied to solids and shells. Additional capabilities included in Workbench and LS-DYNA feature solver version selector, solution monitoring while solving and result plot trackers.

To learn more about the latest features in Ansys Mechanical 2021 R1, watch the What's New video below.

Read from website

Developing CAE software systems for all simulation disciplines. Products: ANSA pre-processor/ EPILYSIS solver and META post-processor suite, and SPDRM, the simulation-process-data-and-resources manager, for a range of industries, incl. the automotive, railway vehicles, aerospace, motorsports, chemical processes engineering, energy, electronics…



BETA CAE Systems announces the release of the v20.1.5 of its software suite

December 28, 2020

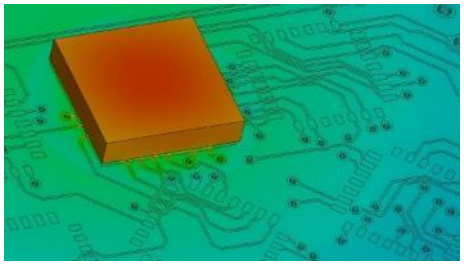
About this release

BETA CAE Systems announces the v20.1.5 release for ANSA/EPILYSIS/META and KOMVOS, hosting numerous fixes in recently detected issues.

Contents

BETA CAE Systems announces the release of the v21.1.0 of its software suite

December 28, 2020



About this release

Consistently trying to minimize simulation turnaround time and accelerate the automatic setup for workflows and processes, BETA CAE Systems proudly presents the release of v21.1.0 of its software suite.

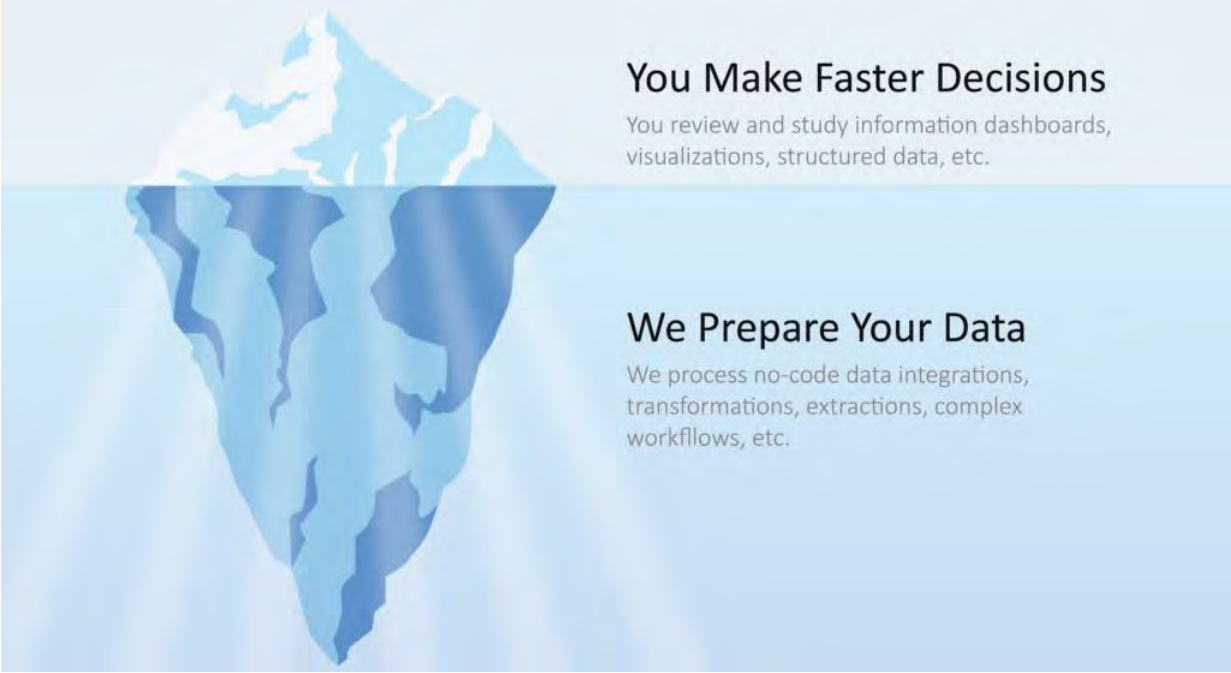
The brand new version offers a plethora of features to unlock new potential for simulation in design and analysis, as well as a range of upgrades and performance improvements for existing workflows.

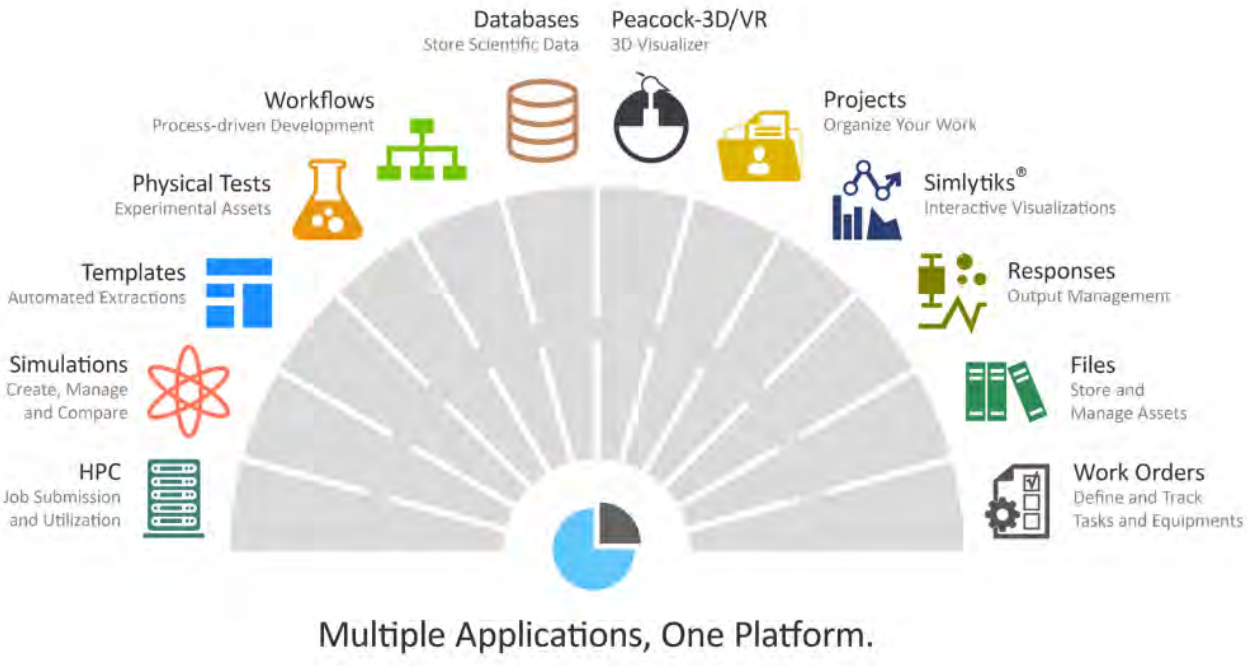
Do not miss:

- The promising entries of Electronic CAD (eCAD) and Electromagnetics, as well as Thermal for structural applications, as simulation fields of analysis.
- The progressing NVH capabilities from pre- to post-processing.
- The impressively accelerated performance in Crash & Safety post-processing processes.
- The augmented pre-processing potential with the advancements in ANSA VR.
- The continuously enriched capabilities of Machine Learning integration in KOMVOS through ANSA.

Contents

d3VIEW is a data to decision platform that provides out-of-the box data extraction, transformation and interactive visualizations. Using d3VIEW, you can visualize, mine and analyze the data quickly to enable faster and better decisions.





www.d3view.com | support@d3view.com

Author: Christian Frech christian.frech@dynamore.de

DYNAmore successfully applies for a research project on AI in the field of material modeling

The Project "AIMM"

For the development of future, efficiency-optimized and low-emission vehicle concepts, the deployment of reliable and precise simulation methods is of fundamental importance. The objective of the Artificial Intelligence for Material Models (AIMM) research project is to complement and/or replace the classical model-based material characterization with an alternative, data-driven material modeling. Particularly in the context of the use of new materials, whose behavior is getting increasingly more complex, this is intended to overcome the limits of conventional material characterization. The focus in AIMM will be on faster deployment of new materials through accelerated description for CAE applications, the development of associated new experimental concepts to generate the necessary training data, and the shortening of the characterization and modeling phase through automation of the processes.

The Project Partners

DYNAmore, with its expertise in the simulation of dynamic tasks in structural mechanics, will be strongly involved in the project administered by the project sponsor TÜV Rheinland, especially with the Material Competence Center (MCC). Other industrial consortium partners are Mercedes-Benz AG, ElringKlinger AG, GOM GmbH and renumics GmbH. On the part of the research institutions, the Ernst-Mach-Institut of the Fraunhofer Gesellschaft, the Institute for Software Engineering and Theoretical Computer Science (IDA) of the Technical University of Berlin, as well as the University of Stuttgart with the Institutes for Aircraft Design and Forming Technology are participating in the project, which is funded with 2.9 million EUR by the Federal Ministry for Economic Affairs and Energy (BMWi) based on a resolution of the German Bundestag.

Professor André Haufe, head of the DYNAmore MCC, comments: "There are numerous possibilities to support, optimize or even replace CAE processes by methods of Artificial Intelligence. As experts in materials characterization, we see new and exciting challenges in this area. The project partners in AIMM are excellently positioned to solve these challenges. A significant project risk is the fact that an industrial application places significantly higher demands on robust implementation, both on the data side for obtaining training data and on the evaluation side. Thus, we are eagerly awaiting the start of the project and look forward to an intensive and productive collaboration."

Contact

DYNAmore GmbH
Prof. Dr.-Ing. André Haufe
Industriestrasse 2
D-70565 Stuttgart
www.dynamore.de
andre.haufe@dynamore.de

Author: Christian Frech christian.frech@dynamore.de

Save the date!

13th European LS-DYNA Conference October 5-6, 2021, Ulm, Germany

Conference Website: www.dynamore.de/en/conf2021

Invitation

We very much hope for a normalization of the situation and that we will be able to welcome the LS-DYNA users personally at a conference again next fall. We kindly invite all users of LS-DYNA, LS-OPT, and LS-TaSC to the 13th European LS-DYNA Conference at October 5-6, 2021 in Ulm, Germany. As usually the conference will be a great opportunity to talk with industry experts, catch up with colleagues and enjoy time exploring new ideas. In addition, attendees can meet with exhibitors to learn about the latest hardware and software trends as well as additional services relating to the finite element solver LS-DYNA, the optimization codes LS-OPT and LS-TaSC, and the pre- and postprocessor LS-PrePost. Training courses and workshops will also take place in the week before, during and after the conference.

Venue

The Congress Centrum Ulm is located directly on the river Danube. The city is best known for its cathedral, the highest church tower in the world and for being the birthplace of Albert Einstein.

Ulm is located directly on the A7 and A8 motorways and can be easily reached from Stuttgart and Munich airports.

Address:
Basteistraße 40
89073 Ulm
Telefon: +49 731 922990
Telefax: +49 731 9229930
www.ulm-messe.de

Abstract submission

Please submit your abstract (maximum length 2,500 characters) by E-Mail to conf@dynamore.de or online at:
www.dynamore.de/en/2021-abstract

Important Dates

Abstract submission: May 28, 2021
Author notification: July 9, 2021
Paper submission: September 3, 2021
Conference date: October 5-6, 2021

Participant fees

Industry speaker:	420 Euro
Academic speaker:	360 Euro
Industry:	640 Euro[1)] / 690 Euro
Academic:	490 Euro[1)] / 540 Euro

[1)] Registration before 30 June 2021. All plus VAT.

Exhibiting and sponsoring

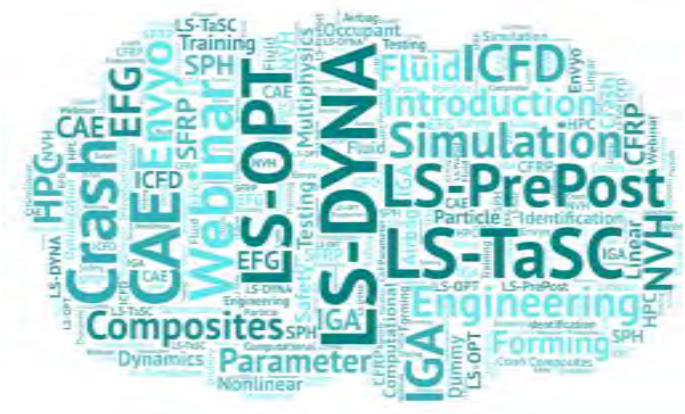
Please request further information.

Contact

DYNAmore GmbH
Industriestr. 2, D-70565 Stuttgart, Germany
Tel. +49 (0) 7 11 - 45 96 00 - 0
E-Mail:conference@dynamore.de
www.dynamore.de/en/conf2021

Author: Christian Frech christian.frech@dynamore.de

Webinars and Video-Seminars 2021



Online trainings in March and April

Webinars LS-DYNA Compact	
Introduction to LS-PrePost	1-2 March
Introduction to Isogeometric Analysis with LS-DYNA	1 March
Introduction to LS-DYNA	2-4, 3-5, 10-13 March
Presentation of the new product generation SCALE.sdm	5 March
Advanced Damage Modeling - Orthotropic Materials	8-9 March
Introduction to ICFD solver	9 March
Smoothed Particle Hydro-Dynamics (SPH)	12 March
Introduction to SPG Method for Manufacturing and Material Failure Analysis	15-16 March
La méthode S-ALE dans LS-DYNA	18 March
Electromagnetism in LS-DYNA	22-23 March
Resistive Heating and Battery Modeling	24 March
Implicit Analysis using LS-DYNA	25-26 March
Introduction to Simulation Data and Process Management in LoCo	29-30 March
User Materials	31 March
LS-OPT Optimization	8 April
LS-OPT Robustness	9 April

Video Seminars	
Introduction to LS-DYNA online	anytime
Crashworthiness Simulation with LS-DYNA	anytime
Modeling Metallic Materials	anytime
LS-OPT - Optimization	anytime
LS-OPT - Robustness	anytime

Visit our website for complete overview and registration www.dynamore.de/en/seminars

A leading innovator in Virtual Prototyping software and services. Specialist in material physics, ESI has developed a unique proficiency in helping industrial manufacturers replace physical prototypes by virtual prototypes, allowing them to virtually manufacture, assemble, test and pre-certify their future products.

ESI Group to collaborate on R&D French project for sustainable automotive aluminum solutions

Paris, January 14th, 2021

ESI Group, global player in virtual prototyping for industries, is to develop lightweight, recyclable and cost-efficient aluminum solutions for the automotive market along with Constellium, Groupe Renault, Institut de Soudure (Welding Institute) and the University of Lorraine within the ISA3 project.

"ESI Group's solutions will help to develop and validate the design and manufacturing of the aluminum doors without the need for physical prototypes. The technology used will enable faster and more accurate predictions of forming, assembly and crash performance of the ISA3 door project. ESI is the French leader offering the most complete range of virtual manufacturing and pre-certification solutions for the industry. The virtualization of all these tests enables our clients to make significant time and cost savings and greatly reduce their environmental footprint." declares Pierre Culière, Pre-Certification & Validation Outcome Director, ESI Group.

Project ISA3 focuses on the design and development of aluminum automotive doors that would be 15% lighter than current average aluminum solutions, more efficient to produce, and recyclable at all stages of its life cycle. This €7 million project is to further lightweight vehicles by accelerating automakers' transition from steel to aluminum. Scheduled to run through 2023, it will focus on alloys and solutions that enable closed loop recycling.

Contacts:

ESI - Shareholder Relations
Florence Barré
investors@esi-group.com
+33 1 49 78 28 28

Verbatee – Press & Investors Relations
Jérôme Goaer, j.goaer@verbatee.com, +33 6 61 61 79 34
Aline Besselièvre, a.besselievre@verbatee.com, +33 6 61 85 10 05

About ESI Group

Founded in 1973, ESI Group is a leading innovator in Virtual Prototyping solutions and a global enabler of industrial transformation. Thanks to the company's unique know-how in the physics of materials, it has developed and refined, over the last 45 years, advanced simulation capabilities. Having identified gaps in the traditional approach to Product Lifecycle Management (PLM), ESI has introduced a holistic methodology centered on industrial productivity and product performance throughout its entire lifecycle, i.e. Product Performance Lifecycle™, from engineering to manufacturing and in operation. Present in more than 20 countries, and in major industrial sectors, ESI employs 1200 high level specialists. In 2019, its turnover was 146M€. ESI is headquartered in France and is listed on compartment B of Euronext Paris. For further information, go to www.esi-group.com.

Read from website

ETA has impacted the design and development of numerous products - autos, trains, aircraft, household appliances, and consumer electronics. By enabling engineers to simulate the behavior of these products during manufacture or during their use, ETA has been involved in making these products safer, more durable, lighter weight, and less expensive to develop.

ETA Inc., Engineering Technology Associates Announces DYNAmore as Master Distributor in Europe

TROY, Michigan (USA) /STUTTGART, Germany – September 8, 2020 –
ETA Inc. (Engineering Technology Associates), an engineering and software innovator with over 37 years in the automotive engineering community, has signed a master distribution agreement with DYNAmore GmbH. DYNAmore is one of the largest distributors of LS-DYNA simulation software worldwide.

"I highly appreciate to further strengthen our long-standing and very good cooperation with ETA and to coordinate the distribution of Dynaform throughout Europe. Together we are well positioned to meet the increasing demands on deep drawing, hydroforming and tube bending simulations."
Ulrich Franz, Managing Director, DYNAmore GmbH

'It is my pleasure to welcome DYNAmore, our long time Dynaform partner and German distributor as our new Master Distributor for Dynaform in the European Union. I am pleased by DYNAmore's business expansion, as they increase their presence in new growth markets across Europe.'
– Dr. Akbar Farahani, CEO & President, ETA Inc.

ETA and DYNAmore have been the most prominent LS-DYNA distributors for over 25 years. This new partnership will bring both companies closer, strengthen the software sales and support to the end-customer and showcase a unified market expansion to European OEMs' and suppliers.

ETA and DYNAmore are committed to creating a powerful virtual presence with webinars, online support and training for customers during the current pandemic and beyond.

DYNAmore will lead the following efforts:

- Supporting customers with the 6th generation of Dynaform
- Providing assistance to European sub-distributors
- Delivering consistent, streamlined communication for software sales and support throughout Europe

For further information on ETA, please visit eta.com
For further information on DYNAmore, please visit www.dynamore.de/en.

Highlights from our FEA Not To Miss Software & Engineering Solutions ISSN 2694-4707
and FEA Not To Miss Website - Sign up for our Monthly Magazine via email

FEA Not To Miss www.feantm.com
Chosen For FEA Information News

Brought to our attention by Christoph Muller

Christoph Müller
Simulation Software and Services worldwide

Who were the brains behind simulation? Besides famous people like Swanson, Hallquist, McNeal Scwendler, and many others, some well "hidden" masterminds saw the potential of simulation in automotive design when the hardware and software were still limited. At that time, not too many people believed in it. One of those persons that believed in it was Dr. Alfred Zimmer that started simulation at Mercedes in the late '60s

Memories of Dr. Alfred Zimmer, who would have been 100 in 2020
FEM pioneer and trailblazer in the automobile industry

CADFEM Users' meeting in Dresden (from left to right): Guenter Mueller (CADFEM), Werner Dirschmid (former AUDI), John Swanson (former SASI/ANSYS), Alfred Zimmer (former Daimler-Benz) and John Hallquist (LSTC/ANSYS)

Shanghai Hengstar & Enhu Technology sells and supports LST's suite of products and other software solutions. These provide the Chinese automotive industry a simulation environment designed and ready multidisciplinary engineering needs, and provide a CAD/CAE/CAM service platform to enhance and optimize the product design and therefore the product quality and manufacture.

恒士达科技 Hengstar Tech.

ENHU

Shanghai Hengstar Technology Co., Ltd

Shanghai Enhu Technology Co., Ltd

Contact us for our LS-DYNA training courses and CAD/CAE/CAM consulting service, such as

- Crashworthiness Simulation with LS-DYNA
- Restraint System Design with Using LS-DYNA
- LS-DYNA MPP
- Airbag Simulation with CPM
- LS-OPT with LS-DYNA

Our classes are given by experts from LSTC USA, domestic OEMs, Germany, Japan, etc. These courses help CAE engineers to effectively use CAE tools such as LS-DYNA to improve car safety and quality, and therefore to enhance the capability of product design and innovation.

Consulting - Besides solver specific software sales, distribution and support activities, we offer associated CAD/CAE/CAM consulting services to the Chinese automotive market.

Solutions - Our software solutions provide the Chinese automotive industry, educational institutions, and other companies a mature suite of tools - powerful and expandable simulation environment designed and ready for future multidisciplinary CAE engineering needs.

Shanghai Hengstar provides engineering CAD/CAE/CAM services, consulting and training that combine analysis and simulation using Finite Element Methods such as LS-DYNA.

Shanghai Hengstar Technology Co., Ltd

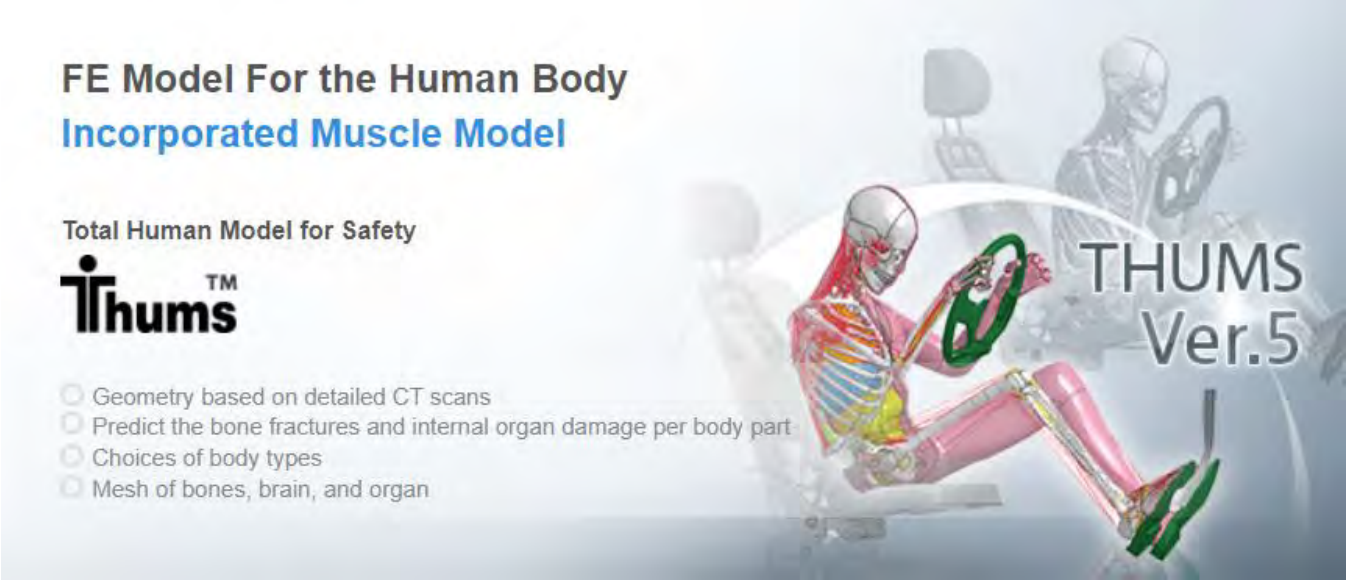
hongsheng@hengstar.com

http://www.hengstar.com

Shanghai Enhu Technology Co., Ltd

http://www.enhu.com

JSOL supports industries with the simulation technology of state-of-the-art. Supporting customers with providing a variety of solutions from software development to technical support, consulting, in CAE (Computer Aided Engineering) field. Sales, Support, Training.



THUMS Features

Use the FE model representing the complex structure of a human body to calculate the behavior and damage due to an impact

Total Human Model for Safety ("THUMS ") is a human body model for injury analysis. It is a Finite Element model (FEM) jointly developed by Toyota Motor Corporation and Toyota Central R&D Labs., Inc.The model aims to simulate "human body kinematics" and "injury on human body" in response to a large impact in a car crash and so on. The geometries of the structurally complex human body parts including the head, torso, joints and organs are represented by FE meshes.

Their material properties refer the list in papers or documents and are compared with component tests listed in papers or documents for validation.THUMS is used by engineers the world over and contribute to the safety improvement of human body.

KAIZENAT Technologies Pvt Ltd is the leading solution provider for complex engineering applications and is founded on Feb 2012 by Dr. Ramesh Venkatesan, who carries 19 years of LS-DYNA expertise. KAIZENAT sells, supports, trains LS-DYNA customers in India. We currently have office in Bangalore, Chennai, Pune and Coimbatore.

KAIZENAT Keep Improving

Kaizen-DYNA App

- "Kaizen-DYNA" is a mobile and web based application which is built by Kaizenat Technologies Private Limited (KTPL) to help LS-DYNA users across the world.
- This powerful application helps LS-DYNA users across the world to stay connected and also help each other by sharing their knowledge.
- The key feature of this application is QUERY and RESPONSE. Where a user can post and respond to queries. The best response for each query will be rewarded with a Kaizen score.
- This application also gives an opportunity for the employers to float their LS-DYNA job openings and alert its user's base with a notification.
- "Kaizen-DYNA" quiz program can help LS-DYNA users to update their knowledge score and trend top in the job seekers list.
- It also gives an opportunity for new users to learn LS-DYNA with training materials FAQ modules.
- This application also brings latest news about LS-DYNA and some useful general information.

Android App

iPhone App

Web App

Contact
Email : support@kaizenat.com Phone: +91 80 41500008

Contact: Email : support@kaizenat.com Phone: +91 80 41500008

On Setting up Multi-Materials in S-ALE Models

Hao Chen, Ansys

LS-DYNA ALE has been widely used to simulating moving fluids interacting with structures. Unlike CFD, the focus is rather on the structure response under dynamic loading from fluids, than the fluids' motion. Fluids are agitated by a high pressure gradient; and then hit the structure, carrying a large momentum. The key in successfully capturing the physics lies in the fluid-structure interaction algorithm. It needs to accurately predict the peak of pressure loading during the impact, which is characterized as a momentum transfer process. This request could only be fulfilled by a transient analysis with a penalty-based coupling between fluids and structure.

In 2015, LSTC introduced a new structured ALE (S-ALE) solver option dedicated to solve the subset of ALE problems where a structured mesh is appropriate. As expected, recognizing the logical regularity of the mesh brought a reduced simulation time for the case of identical structured and unstructured mesh definitions. It also comes with a cleaner, conceptually simpler way of model setup. This article gives a brief description on setting up the S-ALE multi-materials.

Multi-materials.

ALE Multi-materials, or ALE multi-material groups (AMMG), has been used throughout LS-DYNA user manuals and other documentations. This name, by itself, does not mean too much. It was just to follow the naming convention several decades old. The first generation ALE solver only supported one single material so it was called "single material" element formulation (ELEFORM=5 in *SECTION_SOLID). Then the concept of "volume fraction" was introduced and then a "single material with void" element formulation was implemented (ELEFORM=12). Based on the element volume fraction, we could use an interface reconstruction technique to regenerate material interface between that material and void.

Around the same time, the ALE developer found the same technique could be extended to deal with multiple materials flowing inside the ALE mesh. The idea is simple. In single material with void formulation, in each element we have a "volume fraction" which represents the ratio between the material volume and the total element volume. And later after advection, we use that information to reconstruct the material void interface in that element. Now we could simply allow for multiple materials; each has its own element volume fraction value; and each goes through the interface reconstruction process – to find the interface between this material and all other materials. We will cover that process a little bit more later as it is quite important to understand the process to set up the keyword right.

Then the multi-material element formulation (ELEFORM=11) was implemented. And these materials occupying the ALE mesh are referred as "ALE multi-materials". To the author, a more intuitive and shorted name "ALE fluids" might be more appropriate. In the following discussion, "ALE fluids" and "ALE multi-materials" are used interchangeably.

Multi-Material Setup in the Three step setup

S-ALE follows a straight-forward three step setup. First, mesh; secondly, material properties of fluids; thirdly, filling the mesh with fluids. For a detailed description of this 3-step setup, please refer to "On setting up a S-ALE model" published on FEA information Jan 2021 issue.

We set up the ALE fluids through two types of keywords: *MAT_+*EOS to provide material properties; and *ALE_STRUCTURED_MULTI_MATERIAL_GROUP to provide the list of ALE fluids and their order of interface reconstruction.

We will cover the material definitions in the next section and concentrate here on the card *ALE_STRUCTURED_MULTI_MATERIAL_GROUP. The card is of the following format.

*ALE_STRUCTURED_MULTI-MATERIAL_GROUP							
AMMGNM1	MID1	EOSID1					PREF1
…	...	…					…
AMMGNMn	MIDn	EOSIDn					PREFn

"MID" and "EOSID" are the material ID and EOS ID, respectively. *MAT_+*EOS_ provides the complete material properties of 1 ALE fluid. Almost ALE fluids need *EOS properties.

"AMMGNM" is a name one gives to a AMMG (ALE Multi-Material Group), aka ALE fluid. It is a character string and case insensitive. The leading and trailing spaces are trimmed so alignment is not an issue. We suggest short descriptive names such as "soil", "water", "air", "HE"; or sometimes "inAir", "outAir", "airBelow", "airAbove", etc. It will be cross-referenced in other keywords where AMMG needs to be specified. For example, *SET_MULTI-MATERIAL_GROUP_LIST.

"PREF" is to describe the reference pressure or "base pressure" of that fluid. This might be somewhat new to our typical users from solids background. Pressure of a solid material, if not preloaded, always starts from zero. In such case, its reference pressure or base pressure, is zero. But most fluids have non-zero reference pressure. For example, air has a base pressure of 101325 Pa (1 bar atmospheric pressure). Traditionally this reference pressure is prescribed using the field "PREF" in *CONTROL_ALE card. The new *ALE_STRUCTURED_MULTI-MATERIAL_GROUP has a design to allow each AMMG to have its own reference pressure. The idea of reference pressure will be covered more in the next section.

A key point, not well known among even the most experienced ALE users, is that the order of AMMG definitions matters. It plays an important role in the interface reconstruction. And a wrongfully ordered *ALE_STRUCTURED_MULTI_MATERIAL_GROUP card could lead to unphysical behaviors.

The interface reconstruction algorithm is designed to construct one interface between two fluids. It generates a plane cut inside an ALE element; puts one fluid on one side of that plane, another on the other side. When the volumes of these two polygons match the volumes of two fluids, we are all settled there. This algorithm needs to be extended to deal with more than two fluids. For two fluids, we generate one interface. Say three fluids, two interfaces. How should we proceed?

"Onion skin" was adapted. We do one interface at a time. First, we generate the interface between fluid #1 and other fluids. Next, between fluid #1+ fluid #2 and other fluids. Then fluid #1+fluid#2+fluid#3 and others, so on

and so forth. It is like peeling off the onion skin, first we peel #1 off, then #2, then #3. Only now we are constructing #1 first and then stack #2 up and then #3.

Apparently, in order for this process to make sense, an implied assumption needs to be made. That is, all these fluids listed next to each other must also be physically next to each other. Otherwise, the interface reconstruction will put fluids out of its real location.

For example, we have "air", "water", "soil" from up to bottom and we listed them in that order in *ALE_STRUCTURED_MULTI-MATERIAL_GROUP card. In the mixed cell contains all three fluids, we generate the "air" interface first. "Air" is now at the top. And then "air"+"water", so "water" now is below "air". And then "soil" is put at the bottom. Perfect, right? But how about if we list "air" on the first line, but switch the order between "water" and "soil"? Do you see the problem?

OK. First we do "air". "Air" is at the top. Next "air"+"soil" as "soil" is on the second line. Without knowing better, our interface construction algorithm puts "soil" next to "air" at the center. Then "water" at the bottom. We would suddenly find that after advection, there are some "water" mysteriously moved to the bottom of the element. WHY? The order matters.

Not surprisingly the combination of lack of information and limitation in the interface reconstruction algorithm led to lots of strange ALE results. I apologize for the trouble it has brought to our users. As we did not do well on passing this important piece of information effectively to our users. To all readers of this article, I would greatly appreciate if you could forward it to our fellow ALE users.

Material Properties

Most our users are from solid Mechanics background. Same for me. What we care is to apply the fluid loadings onto our structure. The structural response is what we are after. ALE is never intended to solve "CFD" kind of problems. Rather it is to convey a pressure wave through certain fluid(s) and then load it onto a structure.

But in order to do that well, we still need to understand a little bit about fluids. Majorly, how it is different to solids. The first thing here is Equation-of-State. Why almost always we have EOS definition in ALE fluids?

Let us start with internal energy. What is internal energy? Ask a structural engineer, he would say it is the energy deposited into the material by external load. A block free-resting on the ground has zero internal energy. And internal energy starts to increase when load is applied. But is this also true for fluids? Uncompressed fluid has zero internal energy? Take air, can we say it has zero energy when not compressed?

What us structural engineers referred to as "internal energy" is, in fact, "internal energy difference" or "change in internal energy". We pay no attention to the real internal energy of a block not loaded. What we have here is the external work = -internal energy change. The initial internal energy does not matter.

But in certain fluids, like air, we do have a "real" internal energy. And this internal energy contributes to the pressure value. And "EOS_" provides us the way to define the pressure dependence on the internal energy.

Typically in Solid Mechanics, *MAT_ itself is sufficient to describe the whole material behavior. A stress field is divided into dilatational and deviatoric. Dilatational linked to bulk modulus and deviatoric to shear modulus.

And in case of plasticity, dilatational we have no pressure change as it is incompressible. There is no internal energy dependency as it does not play any role in solids behavior.

Fluids are different. First, compression always leads to pressure change. (Please note as ALE is a hydrocode we do not allow incompressibility.) Secondly, pressure depends on internal energy. So for fluids, pressure is a function of compressibility and internal energy. And here comes the *EOS_ keywords to let users prescribe that function.

Take water as an example. Water, when not heavily loaded, could be assumed to be of a linear material with no internal energy dependence. *EOS_LINEAER_POLYNOMIAL (*EOS_01) with bulk modulus defined is sufficient.

```
*MAT_NULL
$#    mid       ro        pc        mu     terod     cerod       ym       pr
         1    998.0       0.0       0.0       0.0       0.0      0.0      0.0
*EOS_LINEAR_POLYNOMIAL
$#  eosid        c0        c1        c2        c3        c4        c5        c6
         1       0.0    2.2e9       0.0       0.0       0.0       0.0       0.0
$#     e0        v0
       0.0       1.0
```

Deviatoric stress only comes from viscous shear for water. This viscosity coefficient (mu) could be defined in *MAT_NULL. For pressure field, we have $p = c_1(\rho/\rho_0 - 1)$. All other polynomial coefficients c_i are zero. e_0 is the internal energy density (per unit volume); and $v_0 = \rho_0/\rho(t), t = 0$ is the compression ratio, at the initial state, respectively. Please note here ρ_0 is the uncompressed material density, not the initial material density. The above setup did two things. First, it defined material properties. Secondly, it also prescribed initial pressure to be 0. The later could be easily overlooked.

So what if we want to prescribe a non-zero initial pressure? For this water, it is simple. We back solve for the initial compression ratio. Say in an ALE model we have two fluids—air and water. To have a pressure equilibrium in the initial state, water needs to have a pressure of 1 bar=101,325 Pa, same as air. We let $v_0 = \rho_0/\rho(t) = \frac{1}{1+p/c_1} = \frac{1}{1+101325/2.2e9} = 0.9999539453$.

Now let us do air. Air is with idea gas law. We could either define it with *EOS_LINEAR_POLYNOMIAL (*EOS_01) or *EOS_IDEAL_GAS (*EOS_12). Let us use *EOS_01.

```
*MAT_NULL
$#    mid       ro        pc        mu     terod     cerod       ym       pr
         2     1.23       0.0       0.0       0.0       0.0      0.0      0.0
*EOS_LINEAR_POLYNOMIAL
$#  eosid        c0        c1        c2        c3        c4        c5        c6
         2       0.0       0.0       0.0       0.0       0.4       0.4       0.0
$#     e0        v0
  253312.5       1.0
```

Same we have no deviatoric stress and for pressure $p = [c_4 + c_5(\rho/\rho_0 - 1)] * E = 0.4 * \rho/\rho_0 * E$. Ring a bell? Idea gas law has a $\gamma = 1.4$ and $p = (\gamma - 1) * \rho/\rho_0 * E$. Its initial pressure of 101,325 Pa is prescribed by assigning internal energy density to $e_0 = \frac{p}{0.4} = 253312.5$ as $\rho/\rho_0 = 1$.

Take another example, high explosive (HE). It is defined by *MAT_HE + *EOS_JWL (*EOS_02).

```
*MAT_HIGH_EXPLOSIVE_BURN
$#    mid       ro        d      pcj     beta        k        g     sigy
         2   1630.0   6930.02.10000E10      0.0      0.0      0.0      0.0
*EOS_JWL
$#  eosid        a        b       r1       r2     omeg       e0       vo
        23.71200E113.231000E9     4.15     0.95     0.37.000000E9      0.0
```

Again, we assume no deviatoric stress and pressure is a function of both compression ratio and internal energy. $p = A\left(1-\frac{\omega}{R_1 V}\right)e^{-R_1 V} + B\left(1-\frac{\omega}{R_2 V}\right)e^{-R_2 V} + \frac{\omega E}{V}$, where $V = {\rho_0}/{\rho}$. It contains two exponential decay terms and an idea gas term to model gaseous explosion process. *MAT_HE controls burn fraction. Basically the ratio of burning at current state. And the pressure is burn fraction times the *EOS_JWL pressure. If a piece of HE is fully burnt, the pressure is simply *EOS_JWL pressure. If not burnt at all, the pressure is zero. At the initial state, everywhere HE is not detonated so everywhere HE has a pressure of zero. From the design of *MAT_HE and *EOS_JWL, we could see that the pressure here is rather gauge pressure than the "real" pressure and they are differed by 1 bar.

Wait. Then how about if we have both air and HE inside an ALE model? Air has an initial pressure of 1 bar (it has to. otherwise how does it expand?). And HE has an initial pressure of … 0! There is no way to reach pressure equilibrium at the first state.

This pressure incompatibility is there for real and we have been living with it for quite some long time. To our defense, HE pressure is of magnitudes higher than air so this discrepancy is considered small, if not tiny. But now we have a way to resolve it. We will see that later in this section.

Reference Pressure

Here is another new to us structural engineers. Reference pressure. We go back to that free-sitting block on the ground. What is its boundary condition? It has a z-direction constraints on nodal motion at the bottom face, right? Is that all?

Right, and wrong. Indeed we only need to constrain z motion at the bottom face to make the simulation right. But one thing most people would overlook is that at all other 5 faces we have pressure boundary condition. Pressure at those 5 faces is … zero. This boundary condition is naturally satisfied. We are getting used to it so much that we never remember there is a boundary condition at those faces.

Now let us continue working on the air, water, HE model. In the previous section, we set up the material definitions and prescribed the initial pressure of each material. Take air, it has an initial pressure of 1 bar and occupies the upper portion of the box-shaped S-ALE mesh. Do we need to apply boundary conditions, say, at the top face?

We must. The air pressure is 1 bar. Without applying a 1 bar pressure boundary condition, the inside air would expand and fly out of the S-ALE mesh. After some time, all air flows out and the run crashes.

One way to apply this pressure boundary condition is to pick all surface ALE segments and apply pressure loadings on those segments through *LOAD_SEGMENT. This is straightforward and easy to understand. But it is a little bit costly. So we took another approach. Instead we subtract this 1 bar pressure from all elements'

pressure when evaluating the internal force. As internal force is only affected by pressure difference, not the real pressure, the nodal force of internal nodes remains the same. To surfaces nodes, subtracting 1 bar off from element pressure is equivalent to applying a 1 bar pressure loading. This is a much faster and more efficient way computational cost wise.

So what we do is, instead of applying pressure boundary conditions, prescribing a reference pressure for each ALE fluid. This brings us a well preserved pressure equilibrium, between ALE fluids and outside.

In addition to that, the newly added *ALE_STRUCTURED_MULTI-MATERIAL_GROUP card had a design to reconcile pressures between different ALE fluids. In last section's example, water and air pressure both being 1 bar at the initial stage; but HE has a base pressure of 0 bar. Previously, the ALE setup only allows for one global reference pressure (PREF in *CONTROL_ALE). So in this case, in elements occupying by HE, pressure was wrongfully subtracted by 1 bar pressure in internal force calculation. *ALE_STRUCTURED_MULTI-MATERIAL_GROUP, to address this problem, allows for each ALE fluid to have its own reference pressure. We can easily reach pressure equilibrium by assigning reference pressure 1 bar to air and water, 0 to HE. Please refer to the example below to better understand this idea.

A Simple Example

We have three ALE fluids, water, air and HE. And a box-shaped S-ALE mesh. Air occupies the upper half; water the lower half; and HE is a small cylinder submerged in the water. Below is the ALE multi-material keywords setup.

Two points to note here.

1. Order matters. HE in the first line, water next, air the third. Of course, we could go the other way around, like air first, then water, HE last.
2. Each ALE fluid has its own reference pressure. Which is the pressure difference between itself and the outside world. For air and water, it is 1 bar. And for HE, it is 0.

```
*ALE_STRUCTURED_MULTI-MATERIAL_GROUP
$# mmgname      mid    eosid                                pref
  HE            3       3                                  0.0
water           1       1                                101325.0
  air           2       2                                101325.0
*MAT_NULL
$#     mid       ro       pc       mu    terod    cerod       ym       pr
       1    998.0      0.0      0.0      0.0      0.0      0.0      0.0
*EOS_LINEAR_POLYNOMIAL
$#   eosid       c0       c1       c2       c3       c4       c5       c6
       1      0.0    2.2e9      0.0      0.0      0.0      0.0      0.0
$#      e0       v0
     0.0      1.0
*MAT_NULL
$#     mid       ro       pc       mu    terod    cerod       ym       pr
       2     1.23      0.0      0.0      0.0      0.0      0.0      0.0
*EOS_LINEAR_POLYNOMIAL
$#   eosid       c0       c1       c2       c3       c4       c5       c6
       2      0.0      0.0      0.0      0.0      0.4      0.4      0.0
$#      e0       v0
```

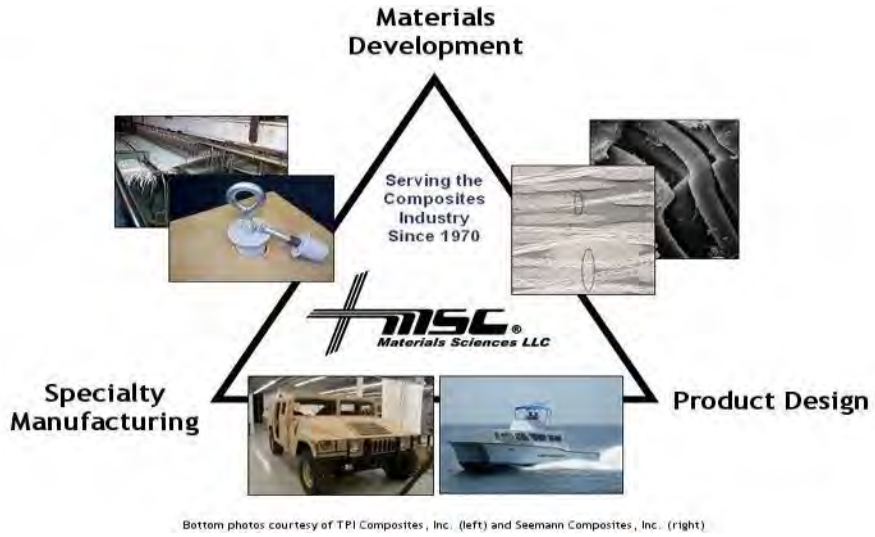
```
 253312.5       1.0
*MAT_HIGH_EXPLOSIVE_BURN
$#     mid        ro         d       pcj      beta         k         g      sigy
         2    1630.0    6930.02.10000E10       0.0       0.0       0.0       0.0
*EOS_JWL
$#   eosid         a         b        r1        r2      omeg        e0        vo
        23.71200E113.231000E9      4.15      0.95      0.37.000000E9       0.0
```

Ending Remarks

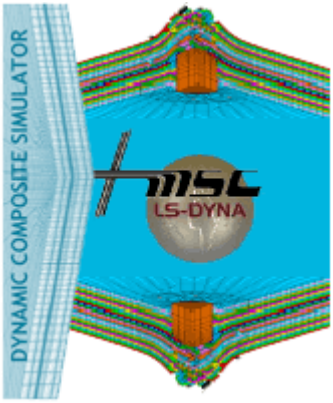
LS-DYNA ALE module has been known for its steep learning curve. Partially it was because setting up Eulerian models are intrinsically different from Lagrange models. But the design of ALE keyword cards, for sure, has caused quite a lot of confusions among our users, new and experienced.

To prompt LS-DYNA ALE usages, Structured ALE solver introduced a new, user-friendly, streamlined three-step setup. We hope this effort could help users, new or old, to perform their work more efficiently and smoothly.

Providing engineering services to the composites industry since 1970. During this time, we have participated in numerous programs that demonstrate our ability to perform advanced composite design, analysis and testing; provide overall program management; work in a team environment; and transition new product development to the military and commercial sectors.

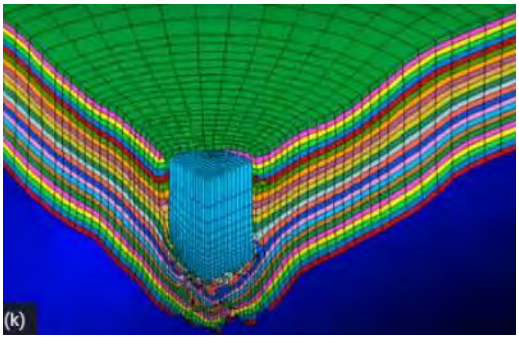


MSC/LS-DYNA Composite Software and Database



Materials Sciences Corporation (MSC) and Livermore Software Technology Corporation (LSTC) announce the Dynamic Composite Simulator module of LS-DYNA.

This enhancement to LS-DYNA, known as MAT161/162, enables the most effective and accurate dynamic progressive failure modeling of composite structures.



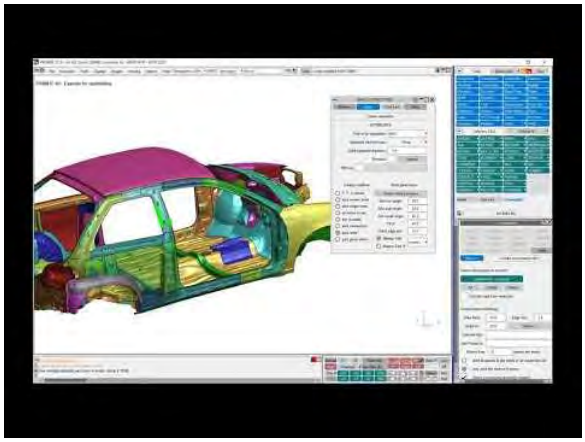
Dyna Fact Sheet (PDF)

Pricing and Contact:

- Types of licenses include: Educational, Commercial, and 30-Day Trial (US only).
- MAT161/162 annual licenses start at $1725 for commmercial use and $175 for educational. (New pricing effective 2017. Contact us for details.)
- Licenses include User's Manual and Technical Support (maintenance, support and updates for time duration of license).
- Please call 215-542-8400 or email dyna_161@materials-sciences.com for more information.

This helps our clients avoid pitfalls, and make exceptionally rapid technological progress. The same broad reach allows us the opportunity to interact with, and evaluate a wide range of suppliers.

Oasys Ltd is the software house of Arup and distributor of the LS-DYNA software in the UK, India and China. We develop the Oasys Suite of pre- and post-processing software for use with LS-DYNA.



Webinar to watch again: Oasys PRIMER - Spotwelding and Connections

Oasys PRIMER has extensive tools and functionality to create and manage connections in LS-DYNA models. This webinar provides an overview and demonstration of the process to create, modify and check spotweld connections.

To view the webinar click on the image on the left.

Top Tip video: Did you know?

Did you know that Oasys PRIMER has a clipboard tool that helps you save selections and copy data between models and include files? To view the video, click on the image on the left.

Oasys and LS-DYNA training courses 2021

The Oasys LS-DYNA training courses are now scheduled and available to view on our website.

Please register your interest by completing the form on the relevant course page.
If you have any additional queries, please reach out to dyna.support@arup.com

Oasys LS-DYNA Social Media Channels

We would like to invite you to join our Oasys LS-DYNA Environment Software LinkedIn Group and YouTube channels. On LinkedIn we share content with other Oasys LS-DYNA software users, from interesting simulations to information about our webinars and training courses.

Please join us by clicking on YouTube and LinkeIn images.

Predictive Engineering

www.predictiveengineering.com

Predictive Engineering provides FEA and CFD consulting services, software, training and support to a broad range of companies.

FINITE ELEMENT ANALYSIS
Predictive Engineering

Predictive Engineering – Western States ANSYS LS-DYNA Distributor – Your Free Coffee Cup is On Its Way!

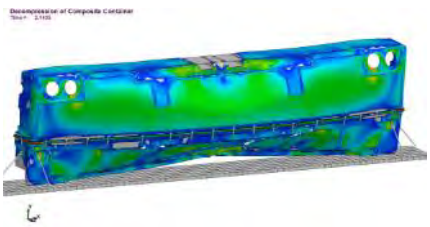
LS-DYNA has been one of Predictive's core analysis tools pretty much since we got started in 1995. It is an amazing numerical workhorse from the basic linear mechanics (think ANSYS or Nastran) to simulating well nigh the impossible. At least that is the way I feel at times when the model is not solving and spitting out arcane error messages and I'm basically questioning my sanity for accepting this project from hell that has a deadline at the end of the week. Which brings me to my favorite project management image – "trough of despair followed by wiggles of false hope then crash of ineptitude and finally the promised land" but I'll leave that for another blog.

For now, let's talk about those free coffee cups. Predictive is now the western states distributor of ANSYS LS-DYNA and provides complete sales, training and services for ANSYS LS-DYNA clients in this region. It is a continuation of our prior setup with LSTC (now ANSYS LST) with the addition of Predictive's ability to offer ANSYS Workbench with LS-DYNA and other ANSYS software tools. So where's my free coffee cup? If you are a current Predictive ANSYS LS-DYNA client, we'll be shipping'em out to you at the end of February and for our new client's – just send us an email or give us a call.

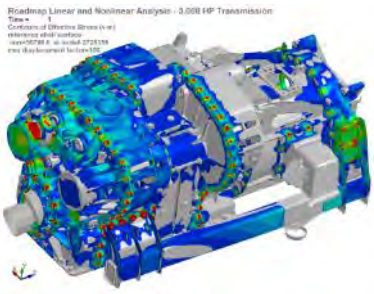
View our portfolio

FEA, CFD and LS-DYNA consulting projects

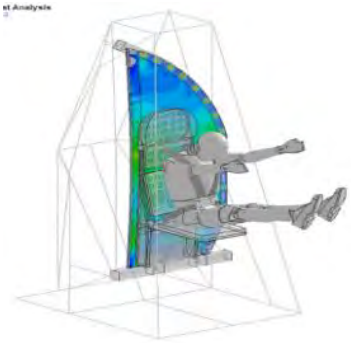
Composite Engineering



Nonlinear Dynamics



Aerospace



Contact:

Address:
2512 SE 25th Ave
Suite 205
Portland, Oregon 97202
USA

Phone:
503-206-5571
Fax:
866-215-1220
E-mail:
sales@predictiveengineering.com

Offering industry-leading software platforms and hardware infrastructure for companies to perform scientific and engineering simulations. Providing simulation platforms that empower engineers, scientists, developers, and CIO and IT professionals to design innovative products, develop robust applications, and transform IT into unified, agile environments.

Rescale Raises $50M Series C to Power Intelligent Computing for Digital R&D

February 2, 2021
English, Thought Leadership, Joris Poort

Today we are excited to announce a new $50M round of funding from both existing and new partners who have joined Rescale's mission to accelerate the digital transformation of R&D-driven organizations through intelligent computing.

From Computing Bits to Atoms

When we founded Rescale a decade ago, we saw the opportunity computing offered to empower the minds of the world's best scientists and engineers driving innovation. In 2010, as cloud and big data were starting to hit their stride to power commodity computing, we envisioned a "big compute" future without computational barriers to accelerate new chip designs, enable computational drug discovery, power the design of new rockets and supersonic jets, and build a sustainable energy future.

Today, Rescale is delivering on this vision — building a better future through intelligent computing for digital R&D.

The Computing Challenge of Our Time

Over the past decade, the algorithms involved in R&D have been underserved by the focus on scale-out hyperscale and big data architectures that leverage commodity computing, and then further compounded by the slowdown of Moore's law. Computational advancement for complex algorithms in R&D, from the physics of rocket science to the latest machine learning and artificial intelligence capabilities, has come from specialized hardware architecture optimizations. The result has been an explosion of specialized purpose-built hardware architectures to drive the continued computing advancements upon which the future of software is built.

Organizations dependent on R&D are thus dealing with a daunting challenge of migrating complex software built on legacy on-premises fixed datacenters to the new paradigm of dynamic cloud environments with new cloud-based specialized architectures.

The Rise of Intelligent Computing

The only way to help R&D-driven companies realize the benefit of specialized computing capabilities in the cloud is through intelligent automation. Specifically, through software-defined infrastructure accompanied by workload-specific hardware performance intelligence, along with the orchestration needed to make the infrastructure invisible to end users in R&D. Any intelligent system has a way of interacting with the world, and uses data to inform its decisions. Rescale builds on the advancement of virtualization and public cloud by establishing a control plane that can work with any cloud or datacenter to run anything from decades-old physics computations to the latest machine learning or containerized applications. Rescale's intelligence layer includes embedded software and hardware performance benchmark metrics combined with pricing and budgeting intelligence. These capabilities ensure R&D organizations are allocating optimal resources to projects that matter most, and that each project is utilizing resources most effectively.

But we haven't stopped there.

The power of connectedness, collaboration, and continuous integration that we've seen in the software development space extends to R&D operations as well. This is the promise of digital R&D – where the R&D model, such as a digital twin, becomes the product. While digital tools for R&D have existed for decades (e.g., computer-aided design, product lifecycle management), the overall R&D cycle has not been fully digitized. For many of our leading customers, Rescale is enabling their digital transformation journey to a digital R&D operating model. It starts with bringing key R&D processes to the cloud, and continues with cloud-based collaboration, data sharing, and completely automated computational pipelines. We've only just seen the beginning of what will become possible with the digitization of R&D.

Open Ecosystem Platform Strategy

Four key elements have been critical for our success as an open ecosystem platform thus far.

Firstly, intelligent computing requires innovation in processor architectures and complementing technologies. NVIDIA, whose venture arm participated in the Series C, is one of the leading players in specialized computing, not only through their innovations in GPU architectures and deep learning, but also with their recent portfolio additions of leading networking solutions from Mellanox and chip design technologies with their pending acquisition of ARM. Many of today's bleeding edge digital R&D capabilities on Rescale are enabled by embedded machine learning and artificial intelligence capabilities powered by NVIDIA's technologies.

Second is the foresight of cloud providers to invest in specialized hardware architectures to broaden the scope of workloads that can be addressed in the cloud. For instance, Microsoft, whose venture arm, M12, participated in the Series C, has built an impressive set of low latency and specialized architectures pushing the entire cloud ecosystem forward in price, performance, and capacity for specialized computing.

Third is the collaboration of software vendors and developers who have embraced cloud deployment and licensing models. Rescale's partnership ecosystem has grown to over 80 commercial partners serving over 680 turnkey applications in our software catalog. Platform capabilities built into Rescale have also enabled open source partners and developers to publish their own software, opening up the ecosystem to all.

Lastly, and most importantly, are the customers and design partners working with Rescale to chart a path towards digital R&D operations – leveraging cloud not only as a place to run applications, but as a way to change how scientists and engineers collaborate and how they accelerate commercializing new product innovations. Samsung, whose venture arm, Samsung Catalyst, participated in the Series C, is a great example of bringing digital R&D capabilities to their fabless customers through Samsung Foundry's cloud design platform, revolutionizing the chip design and fabrication industry.

Focusing on What Matters

In 2020, the world witnessed the pivotal role that science and engineering plays in all of our lives. In the blink of an eye, the world's economy was devastated, with lives lost in the millions. While medical practitioners held the front line, we turned to our scientists and engineers to develop our weapons against this deadly disease. To do our part, Rescale was the first to launch a global coalition with our cloud provider partners, enabling all researchers to access millions of dollars of free supercomputing resources to tackle COVID-19. Again, our scientists & engineers delivered.

As we emerge from the challenges of 2020, we must empower our engineers and scientists with intelligent computing and digital R&D capabilities that will help us accelerate innovation and build a better future. We are still in the early days of realizing the impact the next generation of computing will have as it powers R&D to solve some of the most impactful and important problems of our future. In the years ahead, scientists and engineers will continue to help us not only tackle disease, but also to drive the new innovations that power the future of space exploration, autonomous driving, personalized medicine, and sustainable energy. We are in service to their success, and they deserve the best we can provide.

LS-DYNA China, as the master distributor in China authorized by LST, an Ansys company, is fully responsible for the sales, marketing, technical support and engineering consulting services of LS-DYNA in China.

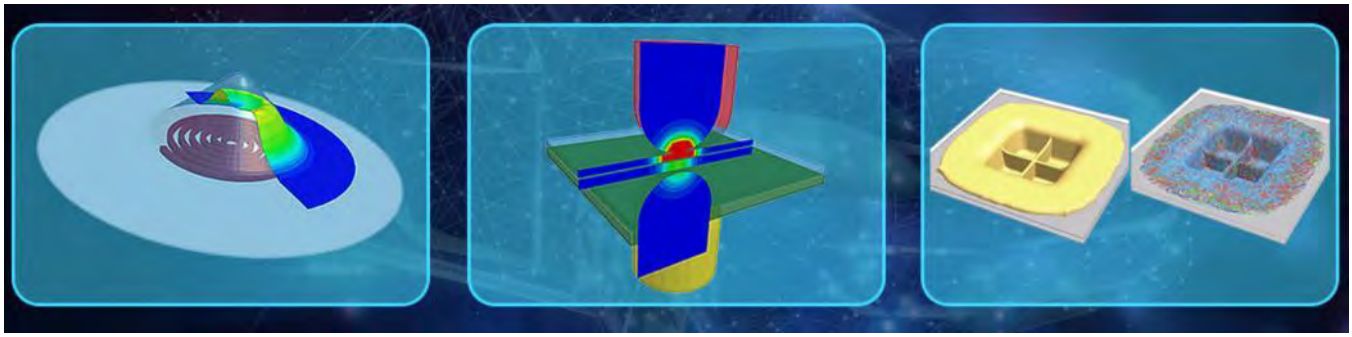
Shanghai Fangkun Software Technology Ltd

Shanghai Fangkun Software Technology Ltd. was authorized by ANSYS Inc as the domestic master distributor of LS-DYNA software. Shanghai Fangkun is fully responsible for domestic sales, marketing, technical support of LS-DYNA. By integrating and managing a wide range of resources such as LS-DYNA agents and partners, Shanghai Fangkun is focus on providing a strong technical support for domestic LS-DYNA users, and help customers to effectively use LS-DYNA software for product design and development.

Based on the strong technical support and developing capability from ANSYS Inc, Shanghai Fangkun attracts a group of top LS-DYNA application engineers and commit to provide LS-DYNA technical support in the automotive industry, electronics industry, rock-soil, aerospace, general machinery and other industries. Shanghai Fangkun devotes to providing all products of LSTC including LS-DYNA, LS-OPT, LS-PREPOST, LS-TASC and LSTC FEA models (dummies model, pedestrian model, etc).

In the meantime, Shanghai Fangkun also relies on strong technical support of ANSYS Inc and will focus on secondary development and process customization of LS-DYNA and its application process. In view of domestic users customization requirement, Shanghai Fangkun will concentrate on customizing custom interface based on LS-PREPOST processing platform, to adjust, standardize and analyzes specific process, improve the efficiency in application, reduce human error, accumulate experience of engineering application, improve customer R&D and competition capabilities.

Shanghai Fangkun will keep mission firmly in mind, devote to improving user satisfaction of LS-DYNA and providing high-quality technical support and engineering consulting services for users.



Contacts

Address: Room 2219, Building No.1, Global Creative Center, Lane 166, Minhong Road, Minhang District, Shanghai
Postcode: 201102 Tel: 4008533856 021-61261195
Sales Email: sales@lsdyna-china.com Technical Support Email: support@lsdyna-china.com

LS-DYNA Training Plan in 2021

Shanghai Fangkun has successfully held several series of LS-DYNA related webinars and training courses in 2020 and received much attention and feedback. Now Shanghai Fangkun release the training plan for 2021 as shown in the following table. Please follow us official Wechat "LSDYNA" to get latest information. All LS-DYNA users and those who interested in are welcome to attend. If you have any questions, please contact email **training@lsdyna-China.com, or dial 021-61261195, 4008533856.**

Date	Topic	Duration
Jan.	LS-DYNA Basic Training	2 days
Feb.	Introduction to LS-PrePost	4-8 hours
Feb.	Introduction to LS-Form & Stamp forming	4-8 hours
Mar	Crash & Safety analysis in LS-DYNA	2 days
Mar	Introduction to LS-Form & Stamp forming	4-8 hours
Apr	GISSMO failure model theory and application of LS-DYNA	4-8 hours
Apr	Simulation of battery crush and nail penetration in multiphysical field with LS-DYNA	4-8 hours
May	Concrete material model in LS-DYNA	2-4 hours
May	Introduction to S-ALE	4-8 hours
Jun	Drop analysis in LS-DYNA	4-8 hours
Jun	Introduction to Contact in LS-DYNA	4-8 hours
Jul	Introduction to EM in LS-DYNA	4-8 hours
Jul	Introduction to LS-OPT	4-8 hours
Aug	ICFD analysis in LS-DYNA	2-4 hours
Aug	LS-DYNA Basic Training	4-8 hours
Sep	Implicit analysis in LS-DYNA	4-8 hours
Sep	CESE analysis in LS-DYNA	2-4 hours
Oct	LS-DYNA application in constranit system	4-8 hours
Oct	Meshfree,SPG and Advanced finite element analysis in LS-DYNA	4-8 hours
Nov	LS-DYNA composite material model training	4-8 hours
Nov	LS-DYNA Thermal-structural-Coupling Analysis	4-8 hours
Dec	LS-DYNA Welding Analysis	4-8 hours
Dec	NVH, Frequency domain and fatigue in LS-DYNA	4-8 hours

Shanghai Fangkun Software Technology Ltd. was authorized by ANSYS Inc as the domestic master distributor of LS-DYNA software and will keep mission firmly in mind, devote to improving user satisfaction of LS-DYNA and providing high-quality technical support and engineering consulting services for users.

CAE software sale & customer support, initial launch-up support, periodic on-site support. Engineering Services. Timely solutions, rapid problem set up, expert analysis, material property test Tension test, compression test, high-speed tension test and viscoelasticitiy test for plastic, rubber or foam materials. We verify the material property by LS-DYNA calculations before delivery.

TERRABYTE Co.,Ltd.
Customer Satisfaction by Computer Solution & Computer Simulation

CAE consulting - Software selection, CAE software sale & customer support , initial launch-up support, periodic on-site support.

Engineering Services - Timely solutions, rapid problem set up, expert analysis - all with our Engineering Services. Terrabyte can provide you with a complete solution to your problem; can provide you all the tools for you to obtain the solution, or offer any intermediate level of support and software.

FE analysis

- LS-DYNA is a general-purpose FE program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing and bioengineering industries.
- ACS SASSI is a state-of-the-art highly specialized finite element computer code for performing 3D nonlinear soil-structure interaction analyses for shallow, embedded, deeply embedded and buried structures under coherent and incoherent earthquake ground motions.

CFD analysis

- AMI CFD software calculates aerodynamics, hydrodynamics, propulsion and aero elasticity which covers from concept design stage of aerocraft to detailed design, test flight and accident analysis.

EM analysis

- JMAG is a comprehensive software suite for electromechanical equipment design and development. Powerful simulation and analysis technologies provide a new standard in performance and quality for product design.

Metal sheet

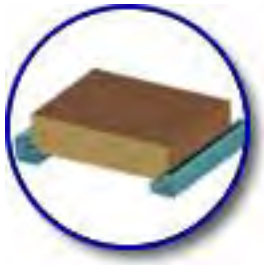
- JSTAMP is an integrated forming simulation system for virtual tool shop based on IT environment. JSTAMP is widely used in many companies, mainly automobile companies and suppliers, electronics, and steel/iron companies in Japan.

Pre/ Post

- **PreSys** is an engineering simulation solution for FE model development. It offers an intuitive user interface with many streamlined functions, allowing fewer operation steps with a minimum amount of data entry.
- **JVISION** - Multipurpose pre/post-processor for FE solver. It has tight interface with LS-DYNA. Users can obtain both load reduction for analysis work and model quality improvements.

Biomechanics

- **The AnyBody Modeling System™** is a software system for simulating the mechanics of the live human body working in concert with its environment.

EXCLUSIVE F-150 ROCKET LEAGUE EDITION SET FOR LAUNCH AS FORD BLASTS FURTHER INTO GAMING WITH PSYONIX COLLABORATION

FEB 10, 2021 | DEARBORN, MICH.

- Ford and Psyonix worked together to create a custom F-150 for Rocket League; the truck will take center stage in the popular multiplayer game where vehicles compete in soccer matches
- Ford is collaborating with Psyonix to be presenting sponsor of the Rocket League Championship Series Winter Majors esports tournament as part of broader gaming strategy
- F-150's appearance in Rocket League continues the company's deep push into gaming in the U.S.; Ford will produce spots narrated by actor Bryan Cranston aimed at gamers

DEARBORN, Mich., Feb. 10, 2021 – The all-new 2021 Ford F-150 is available with a wide array of features, but thanks to a collaboration between Ford and video game maker Psyonix, a new virtual version comes complete with a rocket booster. This animated pickup truck comes as part of the popular Rocket League multiplayer video game where vehicles take center stage to compete in soccer matches.

The Ford F-150 Rocket League Edition will be available as an in-game purchase Feb. 20-28. The F-150 bundle also includes a chairman decal, two sets of F-150 wheels, F-150 Rocket League Edition engine audio, F-150 boost and F-150 player banner.

F-150's appearance in Rocket League continues Ford's wider push into gaming after successful sponsorships with other titles, such as Forza. Ford worked collaboratively with Psyonix, a critically acclaimed video game developer, to create the truck. Ford also will produce commercial shorts narrated by actor Bryan Cranston aimed at gamers.

To ensure the F-150 Rocket League Edition in-game vehicle maintains the real-world styling cues and Built Ford Tough DNA of the iconic pickup, the F-150 design team collaborated with designers at Psyonix, maker of Rocket League. The athletic-looking F-150 that appears on screen is based on the all-new 2021 F-150 and features signature C-clamp headlight design, rounded wheel arches, drop-down windows and "F-150" stamped in the tailgate.

"It had to be an F-150 but one that fits within Psyonix' extreme virtual world pairing soccer and mayhem," said Ehab Kaoud, chief designer, Ford trucks. "F-150 is already the most popular truck in the real world, and this F-150 Rocket League Edition is poised to be the most popular truck in the gaming world, too. It's Built Ford Tough meets gaming."

Along with the in-game vehicle, Ford will serve as presenting sponsor for this month's Rocket League Championship Series Winter Majors, which include an arena full of billboards and a special F-150 that appears between matches to repair and ready the Rocket League pitch for play by repainting boundary lines and towing away damaged vehicles. Ford will sponsor the Ford + Rocket League Freestyle Invitational, a judged competition where select players show off their best in-game moves and tricks – with one lucky contestant winning a real 2021 F-150.

The crossover between the real world F-150 and the virtual reality of the Rocket League follows Ford's collaboration in Europe between Team Fordzilla, the company's e-sports team, and the gaming community there, which led to the creation of the P1 Project virtual race car in 2020.

According to the Entertainment Software Association, a gaming industry trade association, more than 214 million Americans play video games and three-quarters of all U.S. households have at least one gamer in the home.

"As opportunities in gaming continue to grow, we're really looking at this as its own marketing channel," said Scott Denby, strategy manager, Ford brand content and alliances. "It's the same way you would look at social media or TV and film integration. We're identifying authentic ways to be involved in gaming and to do it in a meaningful way. We want to make sure we're adding to the player communities' experience."

The Rocket League Championship Series Winter Majors Driven by Ford continues every weekend in February and can be viewed on both Rocket League's Twitch and YouTube channels. View the remaining schedule below:

- South American Winter Major: Feb. 13-14
- European Winter Major: Feb. 20-21
- Ford + Rocket League Freestyle Invitational: Feb. 24 and Feb. 28
- North American Winter Major: Feb. 27-28

"It's been a great collaborative effort between the Ford and Psyonix teams," said Phil Piliero, vice president, co-studio head, Psyonix. "We learned about the Built Ford Tough DNA, and Ford embraced the customization of rocket-powered cars, resulting in a true one-of-a-kind Rocket League F-150. We're also excited to welcome Ford as presenting sponsor for the Rocket League Championship Series Winter Majors."

The Ford F-150 Rocket League Edition will be available for purchase to new and existing Rocket League players. The game recently expanded and is free to play on all major gaming platforms. Visit rocketleague.com to play or to learn more.

The all-new F-150 – part of the F-Series lineup that's been America's truck sales leader for 44 consecutive years – is purpose-built to be the toughest, most productive F-150 ever. It is built at Ford's Dearborn Truck Plant in Dearborn, Michigan, and Kansas City Assembly Plant in Claycomo, Missouri.

About Ford Motor Company

Ford Motor Company (NYSE: F) is a global company based in Dearborn, Michigan. The company designs, manufactures, markets and services a full line of Ford trucks, utility vehicles, and cars – increasingly including electrified versions – and Lincoln luxury vehicles; provides financial services through Ford Motor Credit Company; and is pursuing leadership positions in electrification; mobility solutions, including self-driving services; and connected vehicle services. Ford employs approximately 186,000 people worldwide. For more information regarding Ford, its products and Ford Motor Credit Company, please visit corporate.ford.com.

Read from website

LS-DYNA - Resource Links

LS-DYNA Multiphysics YouTube

https://www.youtube.com/user/980LsDyna

FAQ LSTC

ftp.lstc.com/outgoing/support/FAQ

LS-DYNA Support Site

www.dynasupport.com

LS-OPT & LS-TaSC

www.lsoptsupport.com

LS-DYNA EXAMPLES

www.dynaexamples.com

LS-DYNA CONFERENCE PUBLICATIONS

www.dynalook.com

ATD –DUMMY MODELS

www.dummymodels.com

LSTC ATD MODELS

www.lstc.com/models www.lstc.com/products/models/mailinglist

AEROSPACE WORKING GROUP

http://awg.lstc.com

Training - Webinars

Participant's Training Classes

Webinars

Info Days

Class Directory

Directory

ANSYS	https://www.ansys.com/services/training-center
BETA CAE Systems	www.beta-cae.com/training.htm
DYNAmore	www.dynamore.de/en/training/seminars
Dynardo	http://www.dynardo.de/en/wost.html
ESI-Group	https://myesi.esi-group.com/trainings/schedules
ETA	http://www.eta.com/training
KOSTECH	www.kostech.co.kr
ANSYS LST	www.lstc.com/training
LS-DYNA OnLine - (Al Tabiei)	www.LSDYNA-ONLINE.COM
OASYS	www.oasys-software.com/training-courses
Predictive Engineering	www.predictiveengineering.com/support-and-training/ls-dyna-training

Contact : 513-331-9139
Email : courses@lsdyna-online.com

LS-DYNA LIVE ONLINE TRAINING & CONSULTING SERVICES

Lsdyna online was created by the LSTC instructor after 25 years of teaching various LS-DYNA courses for LSTC nationally and internationally (more than 20 countries). The online company was established in 2012 and we have been providing many live interactive courses to many companies and organizations. We do consulting work in addition to instructions. Here are some courses, for full list see our webpage.

1. Introduction to LS-DYNA (2 days @ $800)
December 11-12

2. Composites in LS-DYNA (2 days @ $1000)
October 1-2

4. Fracture, Damage, & Failure (2 days @ $1000)
October 5-6

5. Fluid Structure Interaction (2 days @ $1000)
September 29-30

6. Material Models Tests to Simulation (2 days @ $1000)
October 8-9

3. Contact in LS-DYNA (2 days @ $1000)
October 12-13

13. Plasticity, Plastics, & Visco-Plasticity (2 day @ $1000)
November 2-3

14. Penetration Using LS-DYNA (2 days @ $1000)
June 15-16

15. Composite Materials (1 day @ $500)
October 30

16. Blast using LS-DYNA (2 days @ $1000)
November 5-6

17. Introduction to LS-PREPOST (1 day @ $500)
November 4

18. Advance LS-PREPOST (1 day @ 500)
email us for dates

About Tabiei

Dr. Al Tabiei has been a consultant on the use of large scale finite element simulation for more than 25 years to more than 80 large and small companies and government labs in the US and abroad. He was the director of the Center of Excellence in DYNA3D Analysis at the University of Cincinnati (1997-2001). He has more than 150 journal, refereed reports, and conferences papers
He lectured at nearly 20 countries. He also did code development for LSTC. The instructor has developed and implemented many material models in LS-DYNA. Composite Shell element for composite materials and various other development in the code. He was consultant to the US government for several years on the use of simulation for home land security problems. He has served as a Subject Matter Expert (SME) for the government for more than 20 years. He was also on a NASA team for the return to the moon program to investigate different landing scenarios (2006-2010).

Explosion with S-ALE & new features

Eric Piskula ANSYS

In 2015, LSTC introduced a new structured ALE (S-ALE) Solver option dedicated to solver the subset of ALE problems where a structured mesh is appropriate. As you know, LS-DYNA ALE has been widely used to simulating moving fluids interacting with structures. Unlike CFD, the focus is rather on the structure response under dynamic loading from fluids, than the fluids 'motion. Fluids are agitated by a high-pressure gradient; and then hit the structure, carrying a large momentum. It is particularly true for the explosion cases where the Fluid Structure Interaction is the key of success.

S-ALE solver presents multiple of advantages:

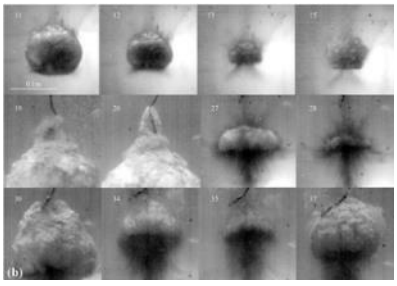
- Structured ALE mesh automatically generated
  - *Smaller input deck; Easier modifications to the mesh; Less I/O time.*
- Shorter calculation time
  - *Sorting, searching faster and more efficient; Also, more accurate.*
- Less memory
  - *A rewritten leaner, cleaner code using less memory to accommodate larger problems.*
  - SMP, MPP, MPP-Hybrid supported
  - *Redesigned algorithm enabled SMP parallelization hence MPP Hybrid.*
  - *Enhancement on MPP efficiency*

Obviously, these main advantages are also for the explosion simulations because the fluid domain is often huge, and it requires a fine mesh.

for introduction, an explosion is a chemical reaction in a substance which converts the original material into a gas at very high temperature and pressure, the process occurring with extreme rapidity and evolving a great deal of heat. The effect of this action is the rise of a pressure wave called a shockwave or detonation wave, which, in water or soil reaches a velocity of up to 8000 m/s, and its pressure at the wave front reaches a value of 100 MPa or higher.

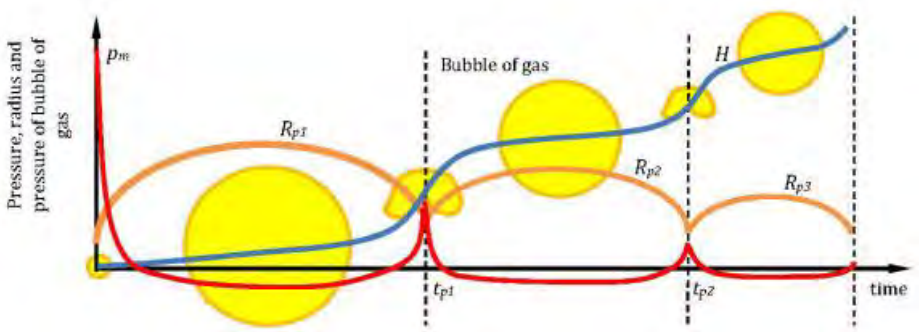
The underwater explosion differs substantially from an explosion in air because it's a bubble of gas formed in water.

The reacted gas sphere interacts with the surrounding fluid in two different phases:



- The first is a transient shock wave, which causes a rapid rise in the fluid velocity, and large inertial loading. The peak pressure of this phase is very high, but its duration is extremely short.

- The second phase in the explosion is a radial pulsation of the gas sphere. During an explosion in water the explosive material having solid state transforms into a gas product having a volume equal to the volume of the explosive material. Under such a high pressure a gas bubble is formed and a spherical shockwave, which at the beginning propagates radially at a high speed. Together with the distance covered, the pressure value on the shockwave front diminishes. The gas bubble is opposed by the hydrostatic pressure of the surrounding water, which causes the so-called pulsation.



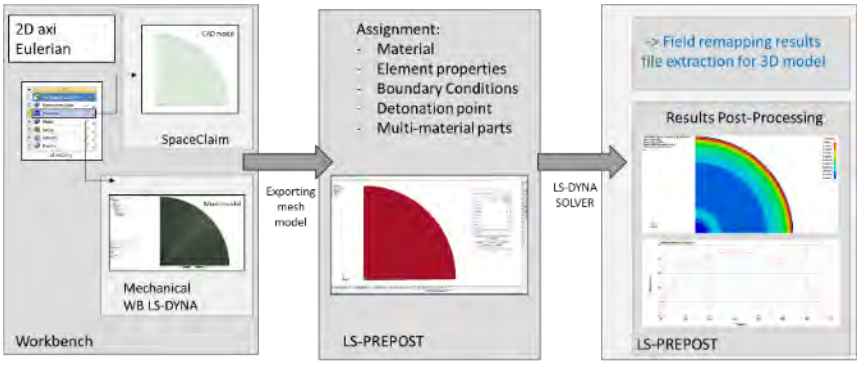
The gas bubble expands and shrinks alternately, moving at the same time towards the surface as the hydrostatic lift force acts on it. When the gas bubble reaches the surface of the water it violently releases the carried gases into the atmosphere throwing them up into the air together with hectoliters of water and vapor.

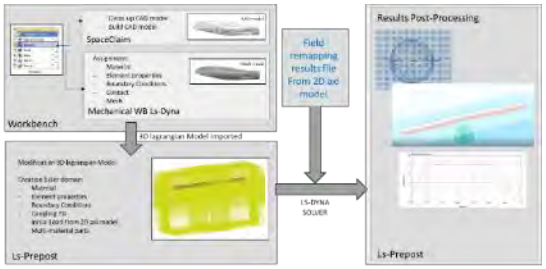
In 2018, S-ALE method had been used with success to simulate close underwater explosion against a vessel on free surface. We had demonstrated the importance to use if possible, the remapping method to keep as longer as possible the correct impulse of explosion.

Indeed, the importance of size mesh is very high to get accurate results. Since 6 to 10 solid elements on the radius of the explosive charge is a minimum requirement for an explosion analysis, the 3D model becomes very quickly very large. More, mesh convergence in a simulation involving explosives requires mesh size of the order of 1-2 mm. So, the model with explosive can fast reach over several billions of solid elements.

So, performing a 2D ALE axisymmetric model allows to get a finer mesh in saving computation time in especially, during the initialization of explosive burning.

A restart is necessary to add 2D previous results into the 3D model trough a remapping.





Once again, the mesh size is always important and can cause to smooth highly results if the mesh size is too coarse.

So globally, coupled blast analysis requires MPP and a sizable cluster: so, it is not accessible to everyone.

In the explosion simulation interacting with structure, the coupling with the Structure is also another point extremely important which requires to satisfy some points. the Lagrangian mesh usually does not share nodes with the ALE mesh. the ALE & Lagrangian meshes interact via a coupling algorithm defined with the command *CONSTRAINED_LAGRANGE_IN_SOLID: This coupling serves to generate forces that resist penetration of the ALE material through the Lagrangian parts. Coupling is a key and sometimes complex aspect of ALE modeling.

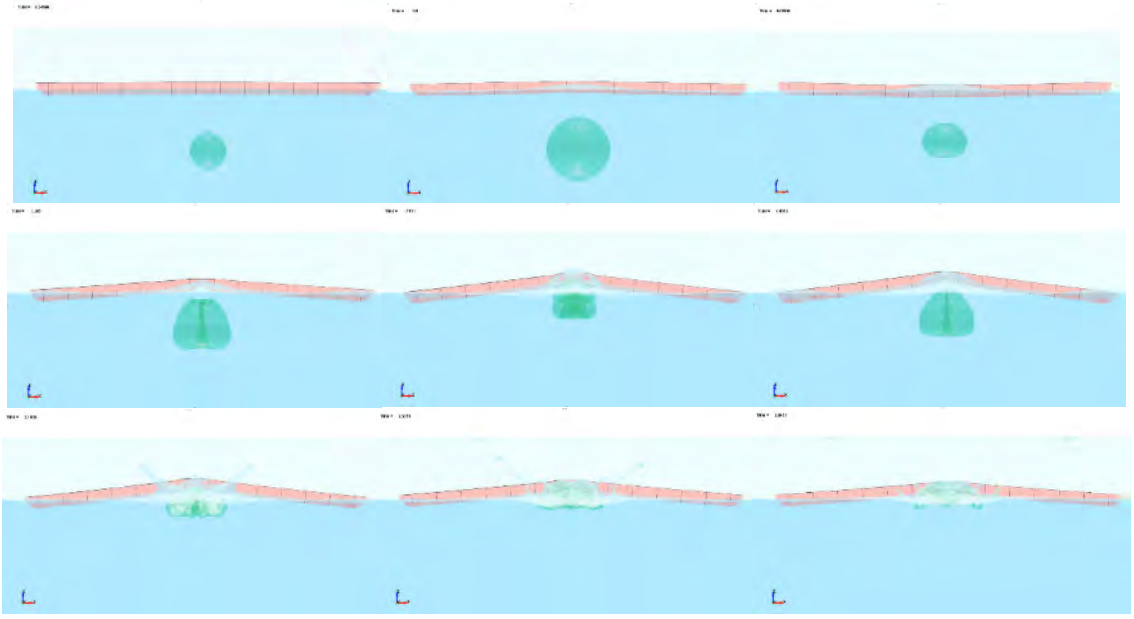
The mesh is one of the success key points about coupling: ideally the good smooth LAGRANGIAN-ALE meshes should be with 1:1 relative resolution to ensure a good coupling. And Obviously, the mesh resolution must be adequate according to flow we want to simulate (example: flow through holes or channels requires more 10 ALE elements…)

Getting a distinct fluid interface coupling to a Lagrangian surface is also recommended with a specific stiffness coupling curve. the default coupling stiffness must be avoided as much as possible because the timestep can affect the coupling stiffness. A load curve that explicitly defines the pressure-penetration(leakage) relationship is the best solution, but this requires to estimate the contact pressure.

In the UNDEX, 3 coupling interfaces needed to be defined:

- Explosive (hot gas) interacting with the ship
- Water in contact with the ship
- Atmospheric Air inside the ship

The combination of these couplings can simulate the collapse of bubble effect on structure as you can see on screenshots below

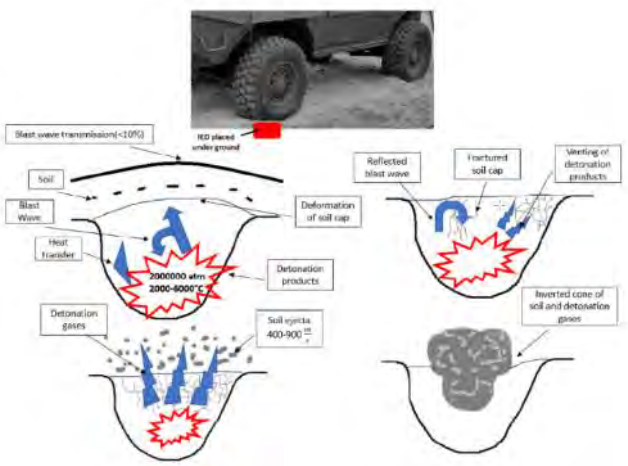


Now talking about our last work: buried landmine explosion effect on light vehicle



To summarize the case of buried landmine explosion, detonation of a landmine causes an explosive, exothermic reaction which results in the formation of a shockwave followed by a rapid expansion of gases. The shockwave is mainly reflected by the soil/air interface and fractures the soil cap over the mine. The detonation products then vent through the voids in the soil, resulting in a hollow inverse cone which consists of the detonation gases surrounded by the soil ejecta. It is the combination of the detonation products and soil ejecta that interact with the target vehicle and cause injury to the vehicle occupants.

As described previously, the two dominant load transfer mechanisms to the target vehicle is the expansion of the detonation products and the physical momentum transfer from soil ejecta.



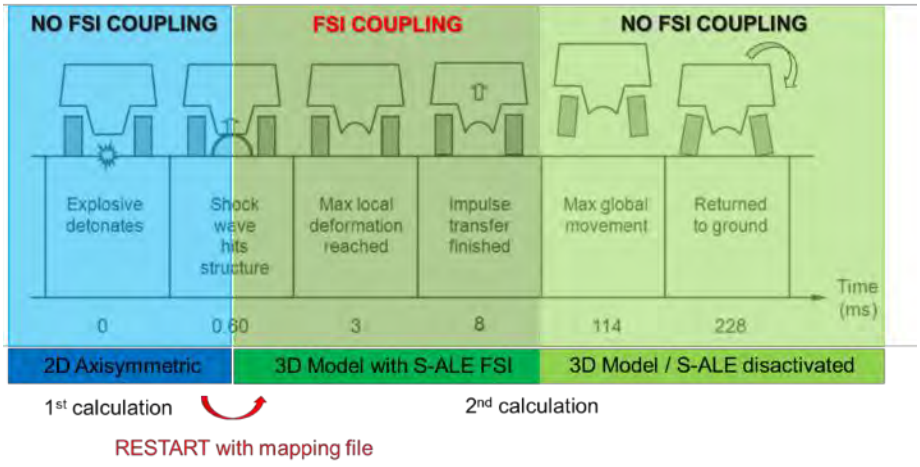
The gas phase provides the first phase of this impulse. During this phase, any portion of the vehicle located in the expansion zone of the detonation products is exposed to a high pressure, transient, supersonic flow field.

The transfer of momentum from the detonation products to the vehicle is governed by its gas dynamics characteristics then the acceleration becomes global and the whole vehicle is lift.

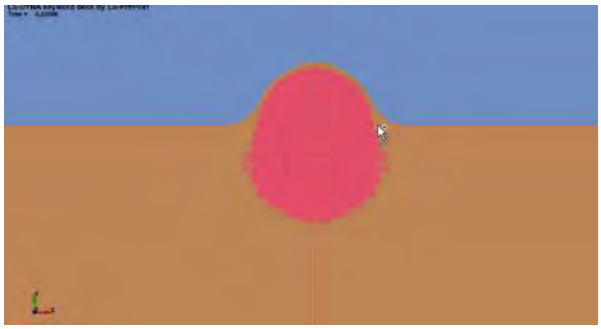
in the case of landmine explosion against a vehicle, the sequence of events is decomposed in 3 steps:

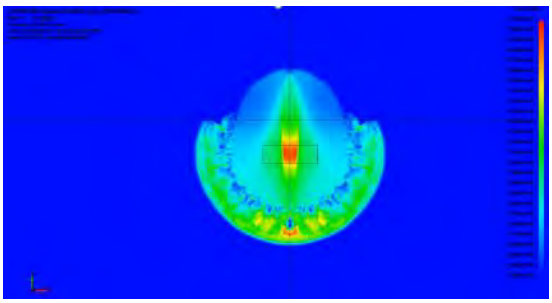
- Explosive detonates / interaction explosive with soil /propagation of shockwave / crater creation / soil debris projection
- The impulse produced from a landmine explosion is mainly the result of detonation products and soil ejecta impacting the target at high velocity. The loading generated from the impulse is characterized by a high pressure during a short response time
- Once, the totally impulse is transferred, the global acceleration involves the lift of the whole vehicle

The approach is similar as UNDEX case: 2D model is required to reduce the model size, especially when the explosion is initiated. But the interaction with structure will start only the blast or soil debris hit the structure. With simulation point of view, the sequence of events can be decomposed like this:

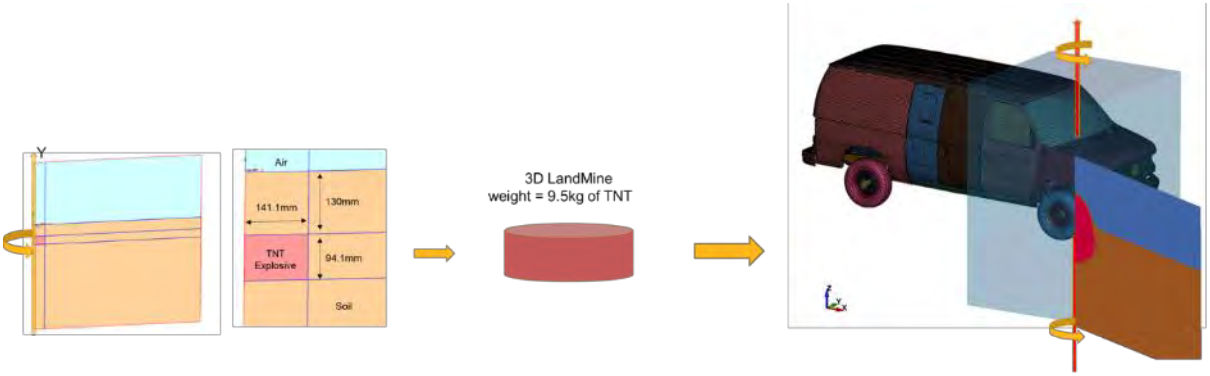


As in the UNDEX case, 2D axisymmetric is performed until the soil ejecta reach the floor of vehicle



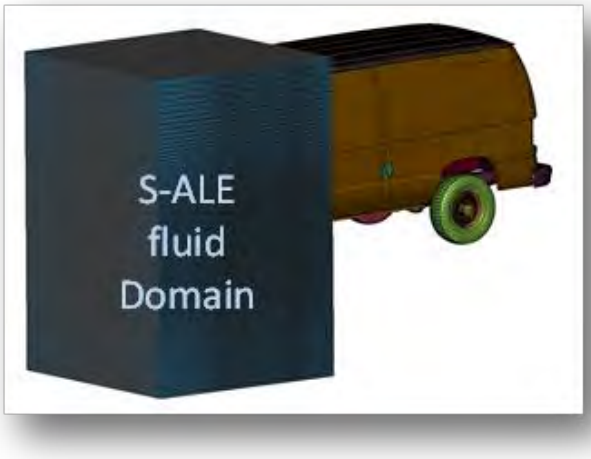


Starting directly from 3D model, this demo case will reach ~2-20 billions solid ALE elements (respectively for 2mm & 1mm mesh size). So, performing a 2D axisymmetric model of the cratering simulation allows to get a finer mesh in saving computation time while being accurate.



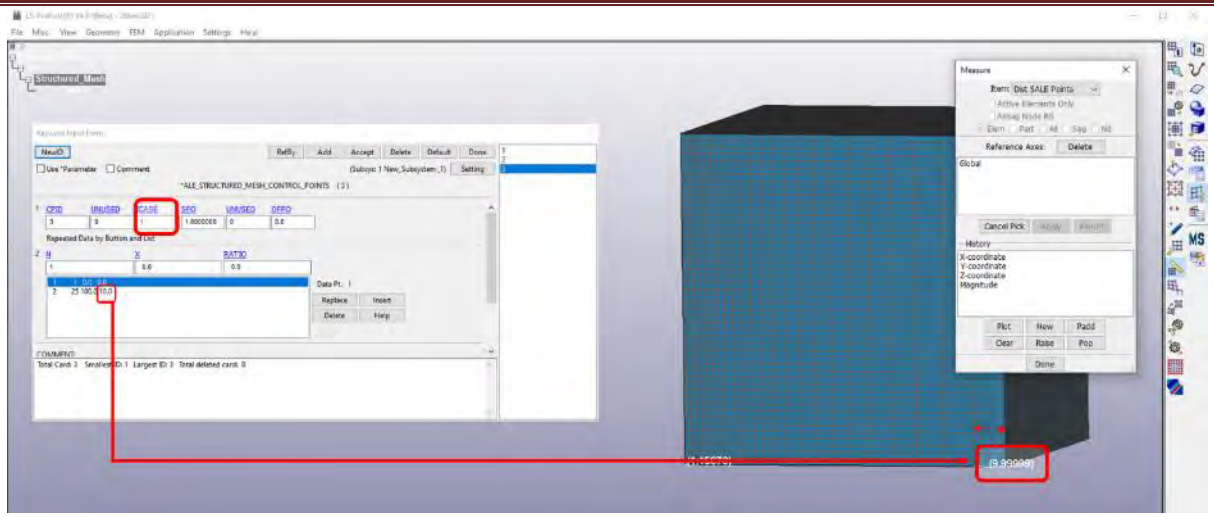
The 2D axisymmetric is done in using ALE solver and the 2D results are mapped on S-ALE 3D model defined through the following keycards *ALE_STRUCTURED_MESH_CONTROLS_POINTS & *ALE_STRUCTURED_MESH.

With the new version R12, a new feature has been added in the keycard *ALE_STRUCTURED_MESH_CONTROLS_POINTS to define more easily a progressive mesh spacing.



The option ICASE=1, allows to generate a progressive mesh in knowing the number of elements and the overall length and the start or the end of mesh size expected.

In this example, 10mm has been chosen for the last S-ALE element. Remark new features available in the next version of LSPP (later than 4.9) allow to help the user to set-up a model including S-ALE. For example, now the space measure between points in S-ALE model is possible; Displaying the S-ALE model in “wire mode” is now also available.



the S-ALE domain requires to be filled with one or several ALE materials.
*ALE_STRUCTURED_MESH_VOLUME_FILLING performs volume filling operations on a structured ALE mesh generated by the*ALE_STRUCTURED_MESH keyword card and so replace old keycard *INITIAL_VOLUME_FRACTION_GEOMETRY which sometimes was not convenient.

In the case of buried landmine, the domain was initially filled with soil & air.

2D axisymmetric results are remapped directly in the 3D S-ALE model and the 3D S-ALE model is initialized with the last results (Pressures, densities, velocities) extracted at the termination time of 2D axisymmetric calculation.

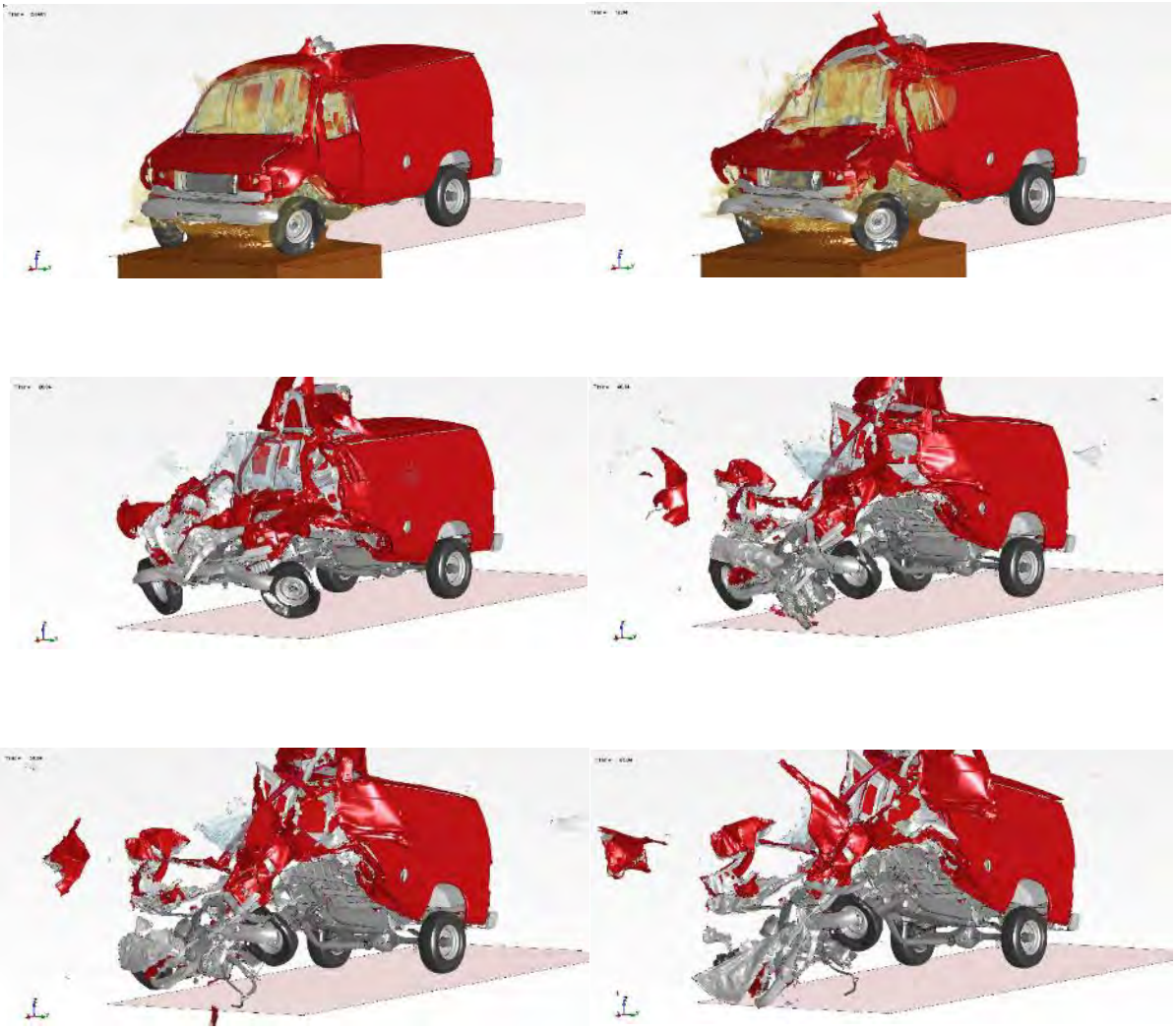
The rule of the coupling has not really changed: each specific fluid material is couple separately with the vehicle and the coupling stiffness is based on pressure versus absolute penetration load curve.

0.0	0.0
~ 0.1* min_ALE_elm_width	P_{max} in ALE domain

But now with R12, the new coupling keycard *ALE_STRUCTURE_FSI has been added and which can use with S-ALE domain. Globally, this keycard has been simplified: less parameters and options are exposed. It mostly follows the format of *CONSTRAINED_LAGRANGE_IN_SOLID cad but with a few exceptions:

- Only penalty formulation coupling method
- Coupling point predefined
- Leakage control automatic
- Normal type selection automatic
- Edge coupling automatic
- Coupling segments updated automatically when erosion is occurred

Once the full impulse transfer is finished, the global acceleration involves the lift of the whole vehicle. From this time, the S-ALE domain can be deactivated.



So in conclusion, the new S-ALE features available in R12 are very promising for complex FSI applications and as seen with the buried landmine demo, they demonstrate a robustness never seen before.

Overview of the CESE compressible fluid and FSI solvers

Zeng-chan Zhang, Grant Cook, Jr. & Kyoung-su Im
Livermore Software Technology, an ANSYS Company, Livermore, CA 94551, USA

Abstract

The Original CESE solver in LS-DYNA is a compressible fluid solver. Over time, more and more capabilities and applications have been added, especially coupled with the LS-DYNA structural FEM solver to solve different fluid/structure interaction (FSI) problems. Among the many problem types suited to using the CESE solver are the following applications: vacuum sucking of flimsy tissue pieces in assembly line processing, airbag deployment, blast wave FSI, cavitation in fuel injection, etc. In this paper, there are three parts: (i) a brief review of the current CESE solver; (ii) introduction of our new dual CESE solver; (iii) our two different FSI solvers and their applications.

1. Current status of the CESE solver

The CESE solver is based on the space-time conservation element and solution element (CESE) method, originally proposed by Chang [1] for solving hyperbolic conservation laws. It has several nontraditional features, including (i) a unified treatment of space and time (thereby ensuring good conservation in both space and time); (ii) unified treatment for arbitrarily shaped polygonal mesh elements; (iii) a simple but efficient discontinuity (shock) treatment. This method is especially useful for high speed compressible flows with complex flows patterns including shock waves and/or detonations. Based on this method, we developed several different CESE solvers, such as:

- **CESE fluid** solver (2D & 3D), general purpose fluid solvers for inviscid and viscous flow calculations
- CESE FSI **solvers, for fluid/structure interaction simulations**
- **CESE non-inertial solver - solving the fluid problem in a non-inertial system (currently only for constant velocity rotating systems)**
- **CESE Chemical reaction flow solver**
- CESE Stochastic particle **solver**

2. Dual-CESE solvers

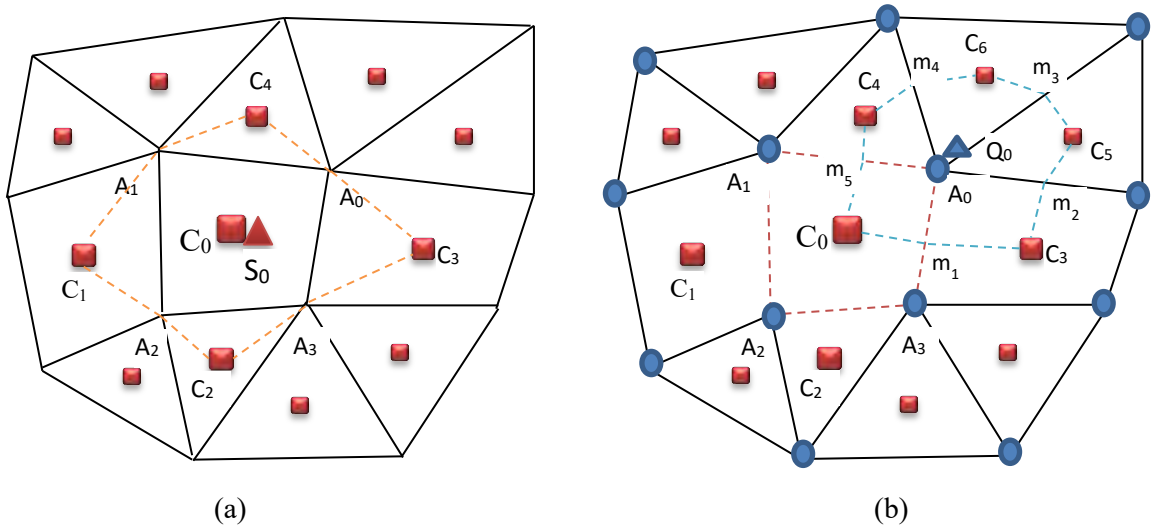
In the CESE method, there are two important concepts, i.e., conservation element (CE) and the solution element (SE), and this is where the CESE method name comes from. In each CE, the flow conservation laws will be enforced, while in the SE, the flow variables will be approximated by some kind of functions, such as first order Taylor series expressions. Another important definition is the solution point (denoted as SP); it is the place where the flow variables are actually solved and stored. For each SP, there is a SE and a CE associated with it.

Fig. 1 shows portion of a typical two-dimensional mesh with mixed triangle and quadrilateral elements. In the standard CESE method, all of the flow variables are solved and stored on the SP of the element center (but not necessarily consistent with it) (see Fig. 1(a)). For example, for the element $A_1A_2A_3A_0$, its CE is $A_1C_1A_2C_2A_3C_3A_0C_4$ (here we only show it in space; if the current calculation time is t^n, then the CE should be extended one time step back

to t^{n-1}), while its SE can be defined as the element itself extended from $t= t^{n-1}$ to $t= t^{n+1}$, plus the middle plane of $A_1C_1A_2C_2A_3C_3A_0C_4$ at $t= t^n$. In most cases, the SP will be not consistent with the element center, for example the element $A_1A_2A_3A_0$, its element center is C_0 , while its SP is S_0 (see Fig. 1 (a)).

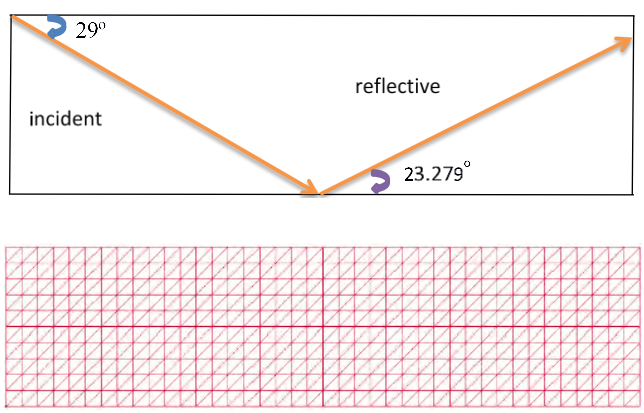
Recently, we have recoded the fluid and FSI solvers mentioned above in the new dual-mesh CESE framework in order to increase their accuracy and stability. In these new solvers, the flow variables are solved and stored on two different sets of SPs in two successive time steps. One set is associated with the element centers (red squares in Fig. 1(b)), the other set is associated with the element vertices (see Fig. 1(b)). Here the solution point of the element center will be consistent with the element center itself, while the solution point of the element vertices is not necessarily consistent with it (e.g., Q_0 is a solution point associated with A_0, see Fig. 1(b)). In the dual-mesh CESE, the CE and SE can be defined as the same, for example, for element center C_0, we can define the CE and SE as $A_1A_2A_3A_0$, similarly for element vertices, we can define its CE and SE as $C_0m_1C_3m_2C_5m_3C_6m_4C_4m_5$ (of course here we only show it in space; it must be extended in the time direction). With this approach, the new CESE solver becomes more accurate than the standard CESE solver using the same mesh resolution. Especially for a two dimensional triangle or a three dimensional tetrahedron mesh, the new dual mesh CESE solver will be more stable (and more accurate too), partially because the number of integration faces increases for each vertex's conservation element.

Fig. 2 and Fig. 3 show comparison results between the standard CESE and the new dual CESE method. The problem is a typical test case [3], i.e., two-dimensional oblique shock reflection problem. In the first case, we use the same number of triangle mesh elements (very coarse mesh, total 800 triangle elements), Fig. 1 shows the mesh (a), pressure contours (c, d) and the pressure distributions at the central line (y≈0.5354). An analytical solution is also included. We can see that the dual mesh CESE solver's results are better than the standard CESE method results. Of course, in this case a very coarse triangle mesh is being used. If the mesh were refined, the difference between these two solvers will also become smaller. Next, we test the coarse quadrilateral element mesh (total 400 quads 40x10) for dual CESE, while for the standard CESE method the mesh is almost doubled (i.e. 57x14=798 quadrilaterals), Fig. 3 shows the results for the two methods (pressure contours and pressure distributions near the middle line). We can see that the dual CESE results are almost the same as the standard CESE results, even if only half the mesh resolution is used.

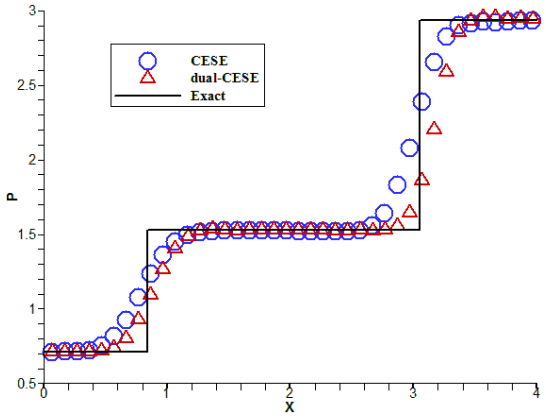


(a) (b)

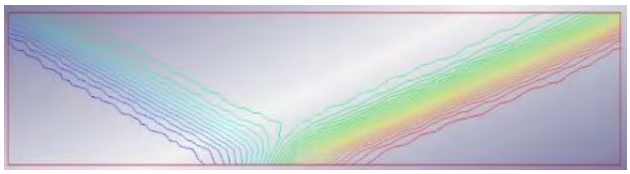
Fig.1 schematic of conservation element (CE) and solution element (SE) under (a) standard CESE and (b) dual CESE solver



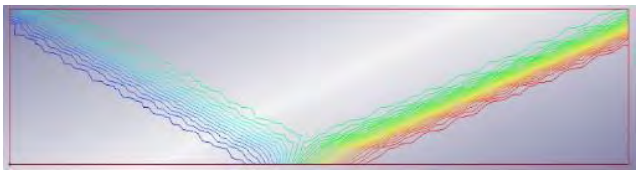
(a) Mesh



(b) pressure distribution at line y=0.5354

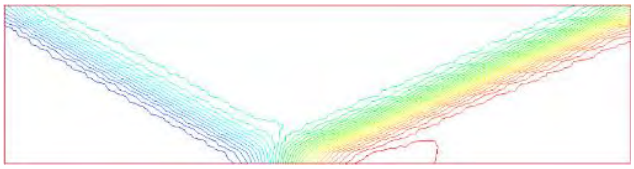


(c) CESE pressure contours

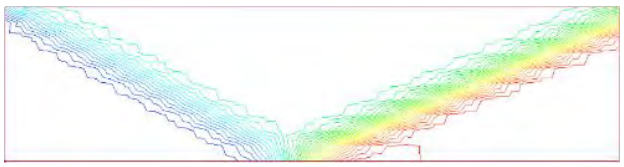


(d) dual CESE pressure contours

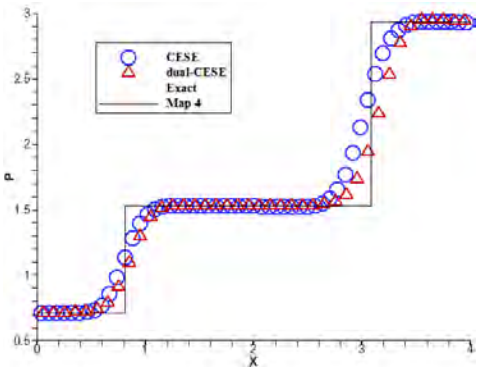
Fig. 2 Comparison between the standard CESE and dual CESE solver with the same mesh



(a) CESE pressure contours



(b) dual CESE pressure contours



(c) pressure distribution near the middle line

Fig. 3 Comparison between the standard CESE and dual CESE solver with different mesh number

3. CESE FSI and Dual-CESE FSI solver

Coupling the CESE fluid solvers with the LS-DYNA structural FEM solver, we have developed the following two fluid/structure interaction FSI solvers based on two different strategies, i.e.,

- **FSI-ibm** solver: It is based on the immersed boundary method (IBM). In this treatment, the fluid mesh is fixed and the structure can move freely inside the fluid domain (fluid mesh and structural meshes are independent of each other). The fluid solver gets the interface displacements and velocities from the structural solver, and then feeds back the forces and/or heat flux to the structural solvers. This solver is very stable and good for large deformation FSI problems.
- **FSI-mmm** solver: In this treatment, the fluid mesh is adjusted to follow the structure's motion in order to match the fluid-structure interface at every time step. The positions of the interior mesh nodes also need to be recalculated every time step or every several time steps. Because of this, it is more expensive than the FSI-ibm solver. But in our new dual CESE implementation with multiple part capability, the FSI-mmm solver's moving portion of the mesh can be limited to small regions near the fluid-structure interfaces. Since the methods used to compute the mesh motion grow at least quadratically in expense with the size of the mesh involved, the possibility to move much smaller pieces of the mesh will greatly alleviate the cost of the mesh motion calculation. This solver is more accurate than the FSI-ibm solver, and it is especially useful for small deformation FSI problems.

Just as the expense of the FSI-mmm solver can be reduced through restricting the mesh motion calculation to just some pieces of the mesh, the expense of the FSI-ibm solver can also be reduced by restricting the pieces of the mesh that are solved with this technique.

Fig. 4 shows accelerating bullet flying through stationary air. Here, an off body bow shock wave is formed in front and a high pressure area is created near its head. For more CESE examples, please go to https://www.dynaexamples.com/cese.

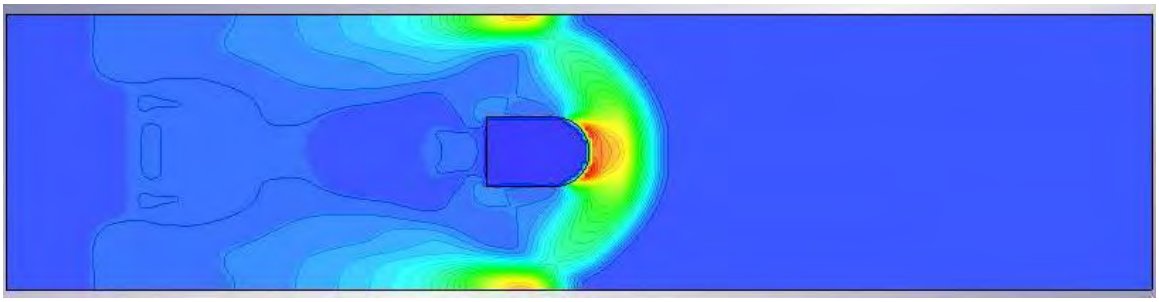


Fig. 4 FSI simulation results (pressure) of bullet flying through a channel

References

[1] S.C. Chang, "The space-time conservation element and solution element – a new approach for solving the Navier-Stokes and Euler equations," J. Comp. Phys. 119 (1995) 295-324

[2] Z.-C. Zhang, S.T.J., Yu and S.-C. Chang, "A space-time conservation element and solution element method for solving the two- and three-dimensional unsteady Euler equations using quadrilateral and hexahedral meshes," J. Comp. Phys. 175 (1) (2002) 168-199

[3] H.C. Yee, R.F. Warming and A. Harten, "Implicit Total Variation Diminishing (TVD) Scheme for Steady-State Calculations," AIAA paper 83-1902 (1983).

BETA CAE Systems. www.beta-cae.com

BETA CAE Systems - ANSA

An advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment. ANSA is a full product modeler for LS-DYNA, with integrated Data Management and Process Automation. ANSA can also be directly coupled with LS-OPT of LST, an ANSYS company to provide an integrated solution in the field of optimization.

BETA CAE Systems μETA

Is a multi-purpose post-processor meeting diverging needs from various CAE disciplines. It owes its success to its impressive performance, innovative features and capabilities of interaction between animations, plots, videos, reports and other objects. It offers extensive support and handling of LS-DYNA 2D and 3D results, including those compressed with SCAI's FEMZIP software.

Solutions for:

Process Automation - Data Management – Meshing – Durability - Crash & Safety NVH - CFD - Thermal analysis - Optimization - Powertrain
Products made of composite materials - Analysis Tools -
Maritime and Offshore Design - Aerospace engineering - Biomechanics

ETA – Engineering Technology Associates www.eta.com
etainfo@eta.com

Inventium Suite™

Inventium Suite™ is an enterprise-level CAE software solution, enabling concept to product. Inventium's first set of tools will be released soon, in the form of an advanced Pre & Post processor, called PreSys.

Inventium's unified and streamlined product architecture will provide users access to all of the suite's software tools. By design, its products will offer a high performance modeling and post-processing system, while providing a robust path for the integration of new tools and third party applications.

PreSys

Inventium's core FE modeling toolset. It is the successor to ETA's VPG/PrePost and FEMB products. PreSys offers an easy to use interface, with drop-down menus and toolbars, increased graphics speed and detailed graphics capabilities. These types of capabilities are combined with powerful, robust and accurate modeling functions.

VPG

Advanced systems analysis package. VPG delivers a unique set of tools which allow engineers to create and visualize, through its modules--structure, safety, drop test, and blast analyses.

DYNAFORM

Complete Die System Simulation Solution. The most accurate die analysis solution available today. Its formability simulation creates a "virtual tryout", predicting forming problems such as cracking, wrinkling, thinning and spring-back before any physical tooling is produced.

ESI Group www.esi-group.com

Visual-Environment is an integrative simulation platform for simulation tools operating either concurrently or standalone for various solver. Comprehensive and integrated solutions for meshing, pre/post processing, process automation and simulation data management are available within same environment enabling seamless execution and automation of tedious workflows. This very open and versatile environment simplifies the work of CAE engineers across the enterprise by facilitating collaboration and data sharing leading to increase of productivity.

Visual-Crash DYNA provides advanced preprocessing functionality for LS-DYNA users, e.g. fast iteration and rapid model revision processes, from data input to visualization for crashworthiness simulation and design. It ensures quick model browsing, advanced mesh editing capabilities and rapid graphical assembly of system models. Visual-Crash DYNA allows graphical creation, modification and deletion of LS-DYNA entities. It comprises tools for checking model quality and simulation parameters prior to launching calculations with the solver. These tools help in correcting errors and fine-tuning the model and simulation before submitting it to the solver, thus saving time and resources.
Several high productivity tools such as advanced dummy positioning, seat morphing, belt fitting and airbag folder are provided in **Visual-Safe,** a dedicated application to safety utilities.

Visual-Mesh is a complete meshing tool supporting CAD import, 1D/2D/3D meshing and editing for linear and quadratic meshes. It supports all meshing capabilities, like shell and solid automesh, batch meshing, topo mesh, layer mesh, etc. A convenient Meshing Process guides you to mesh the given CAD component or full vehicle automatically.

Visual-Viewer built on a multi-page/multi-plot environment, enables data grouping into pages and plots. The application allows creation of any number of pages with up to 16 windows on a single page. These windows can be plot, animation, video, model or drawing block windows. Visual-Viewer performs automated tasks and generates customized reports and thereby increasing engineers' productivity.

Visual-Process provides a whole suite of generic templates based on LS-DYNA solver (et altera). It enables seamless and interactive process automation through customizable LS-DYNA based templates for automated CAE workflows.

All generic process templates are easily accessible within the unique framework of Visual-Environment and can be customized upon request and based on customer's needs.

Visual*DSS* is a framework for Simulation Data and Process Management which connects with Visual-Environment and supports product engineering teams, irrespective of their geographic location, to make correct and realistic decisions throughout the virtual prototyping phase. Visual*DSS* supports seamless connection with various CAD/PLM systems to extract the data required for building virtual tests as well as building and chaining several virtual tests upstream and downstream to achieve an integrated process. It enables the capture, storage and reuse of enterprise knowledge and best practices, as well as the automation of repetitive and cumbersome tasks in a virtual prototyping process, the propagation of engineering changes or design changes from one domain to another.

JSOL Corporation www.jsol.co.jp/english/cae/

HYCRASH

Easy-to-use one step solver, for Stamping-Crash Coupled Analysis. HYCRASH only requires the panels' geometry to calculate manufacturing process effect, geometry of die are not necessary. Additionally, as this is target to usage of crash/strength analysis, even forming analysis data is not needed. If only crash/strength analysis data exists and panel ids is defined. HYCRASH extract panels to calculate it's strain, thickness, and map them to the original data.

JSTAMP/NV

As an integrated press forming simulation system for virtual tool shop the JSTAMP/NV meets the various industrial needs from the areas of automobile, electronics, iron and steel, etc. The JSTAMP/NV gives satisfaction to engineers, reliability to products, and robustness to tool shop via the advanced technology of the JSOL Corporation.

JMAG

JMAG uses the latest techniques to accurately model complex geometries, material properties, and thermal and structural phenomena associated with electromagnetic fields. With its excellent analysis capabilities, JMAG assists your manufacturing process.

LST

Livermore Software Technology, an ANSYS Company
www.lstc.com

LS-DYNA

A general-purpose finite element program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. LS-DYNA is optimized for shared and distributed memory Unix, Linux, and Windows based, platforms, and it is fully QA'd by LST, an ANSYS company. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration.

LS-PrePost

An advanced pre and post-processor that is delivered free with LS-DYNA. The user interface is designed to be both efficient and intuitive. LS-PrePost runs on Windows, Linux, and Macs utilizing OpenGL graphics to achieve fast rendering and XY plotting.

LS-OPT

LS-OPT is a standalone Design Optimization and Probabilistic Analysis package with an interface to LS-DYNA. The graphical preprocessor LS-OPTui facilitates definition of the design input and the creation of a command file while the postprocessor provides output such as approximation accuracy, optimization convergence, tradeoff curves, anthill plots and the relative importance of design variables.

LS-TaSC

A Topology and Shape Computation tool. Developed for engineering analysts who need to optimize structures, LS-TaSC works with both the implicit and explicit solvers of LS-DYNA. LS-TaSC handles topology optimization of large non-linear problems, involving dynamic loads and contact conditions.

LST, AN ANSYS COMPANY Dummy Models

Anthropomorphic Test Devices (ATDs), as known as "crash test dummies", are life-size mannequins equipped with sensors that measure forces, moments, displacements, and accelerations.

LST, AN ANSYS COMPANY Barrier Models

LSTC offers several Offset Deformable Barrier (ODB) and Movable Deformable Barrier (MDB) model.

Material Sciences Corporation www.materials-sciences.com

Materials Sciences Corporation has provided engineering services to the composites industry since 1970. During this time, we have participated in numerous programs that demonstrate our ability to: perform advanced composite design, analysis and testing; provide overall program management; work in a team environment; and transition new product development to the military and commercial sectors. MSC's corporate mission has expanded beyond basic research and development now to include transitioning its proprietary technologies from the research lab into innovative new products. This commitment is demonstrated through increased staffing and a more than 3-fold expansion of facilities to allow in-house manufacturing and testing of advanced composite materials and structures.

Materials Sciences Corporation (MSC) MAT161/162 - enhanced features have been added to the Dynamic Composite Simulator module of LS-DYNA.

This enhancement to LS-DYNA, known as MAT161/162, enables the most effective and accurate dynamic progressive failure modeling of composite structures to enable the most effective and accurate dynamic progressive failure modeling of composite structures currently available.

MSC/LS-DYNA Composite Software and Database -

Fact Sheet: http://www.materials-sciences.com/dyna-factsheet.pdf

- MSC and LSTC have joined forces in developing this powerful composite dynamic analysis code.
- For the first time, users will have the enhanced ability to simulate explicit dynamic engineering problems for composite structures.
- The integration of this module, known as 'MAT 161', into LS-DYNA allows users to account for progressive damage of various fiber, matrix and interply delamination failure modes.
- Implementing this code will result in the ability to optimize the design of composite structures, with significantly improved survivability under various blast and ballistic threats.

MSC's LS-DYNA module can be used to characterize a variety of composite structures in numerous applications—such as this composite hull under blast.

Oasys Ltd. LS-DYNA Environment www.oasys-software.com/dyna

The Oasys Suite of software is exclusively written for LS-DYNA® and is used worldwide by many of the largest LS-DYNA® customers. The suite comprises of:

Oasys PRIMER

Key benefits:

- Pre-Processor created specifically for LS-DYNA®
- Compatible with the latest version of LS-DYNA®
- Maintains the integrity of data
- Over 6000 checks and warnings – many auto-fixable
- Specialist tools for occupant positioning, seatbelt fitting and seat squashing (including setting up pre-simulations)
- Many features for model modification, such as part replace
- Ability to position and depenetrate impactors at multiple locations and produce many input decks automatically (e.g. pedestrian impact, interior head impact)
- Contact penetration checking and fixing
- Connection feature for creation and management of connection entities.
- Support for Volume III keywords and large format/long labels
- Powerful scripting capabilities allowing the user to create custom features and processes

www.oasys-software.com/dyna

Oasys D3PLOT

Key benefits:

- Powerful 3D visualization post-processor created specifically for LS-DYNA®
- Fast, high quality graphics
- Easy, in-depth access to LS-DYNA® results
- Scripting capabilities allowing the user to speed up post-processing, as well as creating user defined data components

www.predictiveengineering.com

Predictive Engineering provides finite element analysis consulting services, software, training and support to a broad range of engineering companies across North America. We strive to exceed client expectations for accuracy, timeliness and knowledge transfer. Our process is both cost-effective and collaborative, ensuring all clients are reference clients.

Our mission is to be honest brokers of information in our consulting services and the software we represent.

Our History

Since 1995, Predictive Engineering has continually expanded its client base. Our clients include many large organizations and industry leaders such as SpaceX, Nike, General Electric, Navistar, FLIR Systems, Sierra Nevada Corp, Georgia-Pacific, Intel, Messier-Dowty and more. Over the years, Predictive Engineering has successfully completed more than 800 projects, and has set itself apart on its strong FEA, CFD and LS-DYNA consulting services.

恒士达科技
Hengstar Tech.

Shanghai Hengstar www.hengstar.com

Center of Excellence: Hengstar Technology is the first LS-DYNA training center of excellence in China. As part of its expanding commitment to helping CAE engineers in China, Hengstar Technology will continue to organize high level training courses, seminars, workshops, forums etc., and will also continue to support CAE events such as: China CAE Annual Conference; China Conference of Automotive Safety Technology; International Forum of Automotive Traffic Safety in China; LS-DYNA China users conference etc.

On Site Training: Hengstar Technology also provides customer customized training programs on-site at the company facility. Training is tailored for customer needs using LS-DYNA such as material test and input keyword preparing; CAE process automation with customized script program; Simulation result correlation with the test result; Special topics with new LS-DYNA features etc..

Distribution & Support: Hengstar distributes and supports LS-DYNA, LS-OPT, LS-Prepost, LS-TaSC, LSTC FEA Models; Hongsheng Lu, previously was directly employed by LSTC before opening his distributorship in China for LSTC software. Hongsheng visits LSTC often to keep update on the latest software features.

Hengstar also distributes and supports d3View; Genesis, Visual DOC, ELSDYNA; Visual-Crash Dyna, Visual-Process, Visual-Environment; EnkiBonnet; and DynaX & MadyX etc.

Consulting

As a consulting company, Hengstar focuses on LS-DYNA applications such as crash and safety, durability, bird strike, stamping, forging, concrete structures, drop analysis, blast response, penetration etc with using LS-DYNA's advanced methods: FEA, ALE, SPH, EFG, DEM, ICFD, EM, CSEC..

Cloud - HPC Services - Subscription JSOL

Contact: JSOL Corporation Engineering Technology Division cae-info@sci.jsol.co.jp

Cloud computing services
for
JSOL Corporation LS-DYNA users in Japan

JSOL Corporation is cooperating with chosen cloud computing services

JSOL Corporation, a Japanese LS-DYNA distributor for Japanese LS-DYNA customers.

LS-DYNA customers in industries / academia / consultancies are facing increased needs for additional LS-DYNA cores

In calculations of optimization, robustness, statistical analysis, we find that an increase in cores of LS-DYNA are needed, for short term extra projects or cores.

JSOL Corporation is cooperating with some cloud computing services for JSOL's LS-DYNA users and willing to provide short term license.

This service is offered to customers using Cloud License fee schedule, the additional fee is less expensive than purchasing yearly license.

The following services are available (only in Japanese). HPC OnLine:

NEC Solution Innovators, Ltd. - http://jpn.nec.com/manufacture/machinery/hpc_online/

Focus - Foundation for Computational Science
http://www.j-focus.or.jp

Platform Computation Cloud - CreDist.Inc.

PLEXUS CAE
Information Services International-Dentsu, Ltd. (ISID) https://portal.plexusplm.com/plexus-cae/

SCSK Corporation - http://www.scsk.jp/product/keyword/keyword07.html

Cloud - HPC Services - Subscription RESCALE

www.rescale.com

Rescale: Cloud Simulation Platform

The Power of Simulation Innovation

We believe in the power of innovation. Engineering and science designs and ideas are limitless. So why should your hardware and software be limited? You shouldn't have to choose between expanding your simulations or saving time and budget.

Using the power of cloud technology combined with LS-DYNA allows you to:

- Accelerate complex simulations and fully explore the design space
- Optimize the analysis process with hourly software and hardware resources
- Leverage agile IT resources to provide flexibility and scalability

True On-Demand, Global Infrastructure

Teams are no longer in one location, country, or even continent. However, company data centers are often in one place, and everyone must connect in, regardless of office. For engineers across different regions, this can cause connection issues, wasted time, and product delays.

Rescale has strategic/technology partnerships with infrastructure and software providers to offer the following:

- Largest global hardware footprint – GPUs, Xeon Phi, InfiniBand
- Customizable configurations to meet every simulation demand
- Worldwide resource access provides industry-leading tools to every team
- Pay-per-use business model means you only pay for the resources you use
- True on-demand resources – no more queues

ScaleX Enterprise: Transform IT, Empower Engineers, Unleash Innovation

The ScaleX Enterprise simulation platform provides scalability and flexibility to companies while offering enterprise IT and management teams the opportunity to expand and empower their organizations.

Rescale Cloud Simulation Platform

www.rescale.com

ScaleX Enterprise allows enterprise companies to stay at the leading edge of computing technology while maximizing product design and accelerating the time to market by providing:

- Collaboration tools
- Administrative control
- API/Scheduler integration
- On-premise HPC integration

Industry-Leading Security

Rescale has built proprietary, industry-leading security solutions into the platform, meeting the needs of customers in the most demanding and competitive industries and markets.

- Manage engineering teams with user authentication and administrative controls
- Data is secure every step of the way with end-to-end data encryption
- Jobs run on isolated, kernel-encrypted, private clusters
- Data centers include biometric entry authentication
- Platforms routinely submit to independent external security audits

Rescale maintains key relationships to provide LS-DYNA on demand on a global scale. If you have a need to accelerate the simulation process and be an innovative leader, contact Rescale or the following partners to begin running LS-DYNA on Rescale's industry-leading cloud simulation platform.

LSTC - DYNAmore GmbH JSOL Corporation

Rescale, Inc. - 1-855-737-2253 (1-855-RESCALE) - info@rescale.com

944 Market St. #300, San Francisco, CA 94102 USA

ESI Cloud Based Virtual Engineering Solutions

www.esi-group.com

ESI Cloud offers designers and engineers cloud-based computer aided engineering (CAE) solutions across physics and engineering disciplines.

ESI Cloud combines ESI's industry tested virtual engineering solutions integrated onto ESI's Cloud Platform with browser based modeling,

With ESI Cloud users can choose from two basic usage models:

- An end-to-end SaaS model: Where modeling, multi-physics solving, results visualization and collaboration are conducted in the cloud through a web browser.
- A Hybrid model: Where modeling is done on desktop with solve, visualization and collaboration done in the cloud through a web browser.

Virtual Performance Solution:

ESI Cloud offers ESI's flagship Virtual Performance Solution (VPS) for multi-domain performance simulation as a hybrid offering on its cloud platform. With this offering, users can harness the power of Virtual Performance Solution, leading multi-domain CAE solution for virtual engineering of crash, safety, comfort, NVH (noise, vibration and harshness), acoustics, stiffness and durability.

In this hybrid model, users utilize VPS on their desktop for modeling including geometry, meshing and simulation set up. ESI Cloud is then used for high performance computing with an integrated visualization and real time collaboration offering through a web browser.

The benefits of VPS hybrid on ESI Cloud include:

- Running large concurrent simulations on demand
- On demand access to scalable and secured cloud HPC resources
- Three tiered security strategy for your data
- Visualization of large simulation data sets
- Real-time browser based visualization and collaboration
- Time and cost reduction for data transfer between cloud and desktop environments
- Support, consulting and training services with ESI's engineering teams

VPS On Demand

ESI Cloud features the Virtual Performance Solution (VPS) enabling engineers to analyze and test products, components, parts or material used in different engineering domains including crash and high velocity impact, occupant safety, NVH and interior acoustics, static and dynamic load cases. The solution enables VPS users to overcome hardware limitations and to drastically reduce their simulation time by running on demand very large concurrent simulations that take advantage of the flexible nature of cloud computing.

Key solution capabilities:

- Access to various physics for multi-domain optimization
- Flexible hybrid model from desktop to cloud computing
- On demand provisioning of hardware resources
- Distributed parallel processing using MPI (Message Passing Interface) protocol
- Distributed parallel computing with 10 Gb/s high speed interconnects

Result visualization

ESI Cloud deploys both client-side and server-side rendering technologies. This enables the full interactivity needed during the simulation workflow along with the ability to handle large data generated for 3D result visualization in the browser, removing the need for time consuming data transfers. Additionally ESI Cloud visualization engine enables the comparisons of different results through a multiple window user interface design.

Key result visualization capabilities:

- CPU or GPU based client and server side rendering
- Mobility with desktop like performance through the browser
- 2D/3D VPS contour plots and animations
- Custom multi-window system for 2D plots and 3D contours
- Zooming, panning, rotating, and sectioning of multiple windows

Collaboration

To enable real time multi-user and multi company collaboration, ESI Cloud offers extensive synchronous and asynchronous collaboration capabilities. Several users can view the same project, interact with the same model results, pass control from one to another. Any markups, discussions or annotations can be archived for future reference or be assigned as tasks to other members of the team.

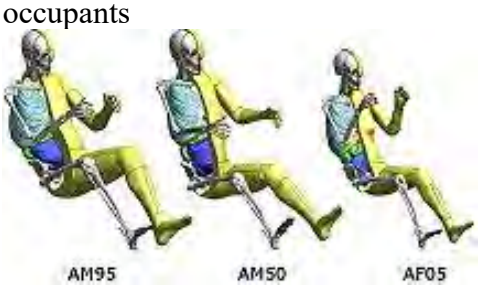
Key collaboration capabilities:

- Data, workflow or project asynchronous collaboration
- Multi-user, browser based collaboration for CAD, geometry, mesh and results models
- Real-time design review with notes, annotations and images archiving and retrieval
- Email invite to non ESI Cloud users for real time collaboration

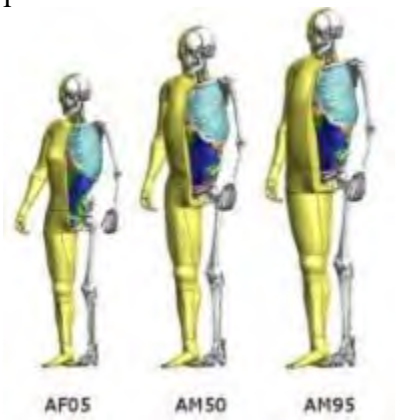
TOYOTA - Total Human Model for Safety – THUMS

The Total Human Model for Safety, or THUMS®, is a joint development of Toyota Motor Corporation and Toyota Central R&D Labs. Unlike dummy models, which are simplified representation of humans, THUMS represents actual humans in detail, including the outer shape, but also bones, muscles, ligaments, tendons, and internal organs. Therefore, THUMS can be used in automotive crash simulations to identify safety problems and find their solutions.

Each of the different sized models is available as sitting model to represent vehicle occupants



and as standing model to represent pedestrians.



The internal organs were modeled based on high resolution CT-scans.

THUMS is limited to civilian use and may under no circumstances be used in military applications.

LSTC is the US distributor for THUMS. Commercial and academic licenses are available.

For information please contact: THUMS@lstc.com

THUMS®, is a registered trademark of Toyota Central R&D Labs.

LST, An ANSYS Company – Dummy Models

Crash Test Dummies (ATD)

Meeting the need of their LS-DYNA users for an affordable crash test dummy (ATD), LSTC offers the LSTC developed dummies at no cost to LS-DYNA users.

LSTC continues development on the LSTC Dummy models with the help and support of their customers. Some of the models are joint developments with their partners.

e-mail to: atds@lstc.com

Models completed and available (in at least an alpha version)

- Hybrid III Rigid-FE Adults
- Hybrid III 50th percentile FAST
- Hybrid III 5th percentile detailed
- Hybrid III 50th percentile detailed
- Hybrid III 50th percentile standing
- EuroSID 2
- EuroSID 2re
- SID-IIs Revision D
- USSID
- Free Motion Headform
- Pedestrian Legform Impactors

Models In Development

- Hybrid III 95th percentile detailed
- Hybrid III 3-year-old
- Hybrid II
- WorldSID 50th percentile
- THOR NT FAST
- Ejection Mitigation Headform

Planned Models

- FAA Hybrid III
- FAST version of THOR NT
- FAST version of EuroSID 2
- FAST version of EuroSID 2re
- Pedestrian Headforms
- Q-Series Child Dummies
- FLEX-PLI

LST, An ANSYS Company – Barrier Models

Meeting the need of their LS-DYNA users for affordable barrier models, LSTC offers the LSTC developed barrier models at no cost to LS-DYNA users.

LSTC offers several Offset Deformable Barrier (ODB) and Movable Deformable Barrier (MDB) models:

- ODB modeled with shell elements
- ODB modeled with solid elements
- ODB modeled with a combination of shell and solid elements
- MDB according to FMVSS 214 modeled with shell elements
- MDB according to FMVSS 214 modeled with solid elements
- MDB according to ECE R-95 modeled with shell elements
- AE-MDB modeled with shell elements
- IIHS MDB modeled with shell elements
- IIHS MDB modeled with solid elements
- RCAR bumper barrier
- RMDB modeled with shell and solid elements

LSTC ODB and MDB models are developed to correlate to several tests provided by our customers. These tests are proprietary data and are not currently available to the public.

All current models can be obtained through our webpage in the LSTC Models download section or through your LS-DYNA distributor.

To submit questions, suggestions, or feedback about LSTC's models, please send an e-mail to: atds@lstc.com. Also, please contact us if you would like to help improve these models by sharing test data.

 **FACEBOOK**

BETA CAE Systems
CADFEM
ESI Group

 **TWITTER**

BETA CAE Systems
ESI Group
ETA
CADFEM

 **LINKEDIN**

BETA CAE Systems
CADFEM
DYNAmore Nordic
ETA
ESI Group

 **YOUTUBE**

YOUTUBE Channel	WebSite URL
BETA CAE Systems	www.beta-cae.com
CADFEM	www.cadfem.de
ESI Group	www.esi-group.com
ETA	www.eta.com
Lancemore	www.lancemore.jp/index_en.html
LS-DYNA OnLine - (Al Tabiei)	https://www.youtube.com/user/LSDYNATV

GOOGLE+

BETA CAE Systems