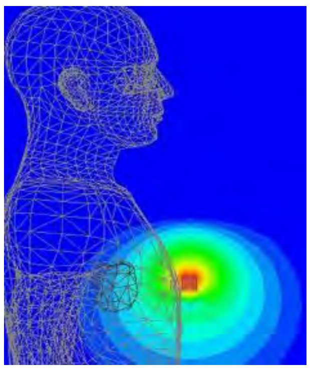
feaINFORMATION Engineering Solutions

Ansys LST

Issue 5, May 2021

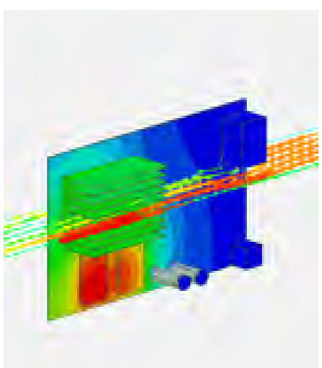
www.lstc.com www.ansys.com www.feapublications.com

ANSYS

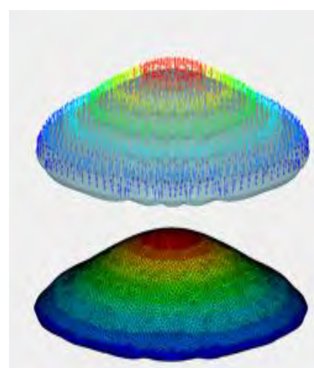


JSOL

KAIZENAT



LST



LS-DYNA@ New Feature and Application

- **On Setting up the Structured ALE Mesh**
- **FSI Based on CESE Compressible Flow Solver with Detailed Finite Rate Chemistry**

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

Articles – Blogs – News

Automotive News, Resource links, LS-DYNA Training

LS-DYNA New Feature and Application

Resources

LS-PrePost® an Advanced Pre- and Post-processor

LS-PrePost® is an advanced pre- and post-processor developed for LS-DYNA®. It is fully multi-platform with support for Windows, Linux and Mac OSX. LS-PrePost is based on the OpenGL rendering engine with a design that is both efficient and intuitive. It is delivered with LS-DYNA without additional cost and may be installed on multiple platforms. License keys are not needed.

Geometry and Meshing Includes

- A geometry engine which allows the creation and modification of curves, surfaces, and solid objects. Also included are tools to heal and simplify the geometry model
- An automatic surface meshing tool
- An automatic 3-Dimension(3D) tetrahedron meshing module
- Various methods to create a mesh by dragging, spinning, offsetting, and sweeping
- The construction of middle surface shells from 3D Solids

Pre- and Post-Processing Capabilities

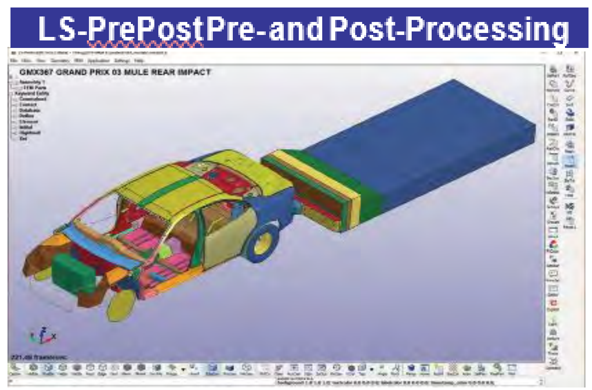
- Complete LS-DYNA Keyword management
- Tools to create and modify LS-DYNA entities
- General model setup for NVH (Noise, Vibration and Harshness), Implicit, and Thermal Analyses
- Tools to measure FEA data like distance, area, angle, volume, mass, etc.
- Section cuts for better visualization in complicated models
- Comprehensive time history plotting for the d3plot, ASCII history, and BINOUT databases
- Time history plotting for user defined data
- Particle elements (SPH, CPM, DES, SPG) visualization
- CFD models and results visualization

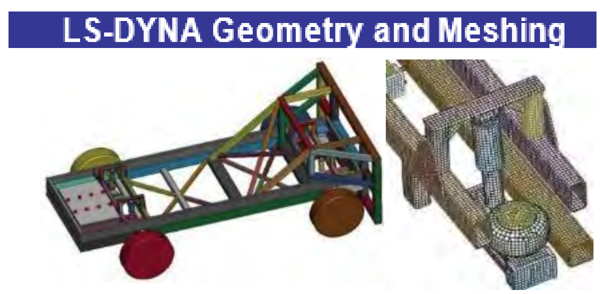
Other General Functions

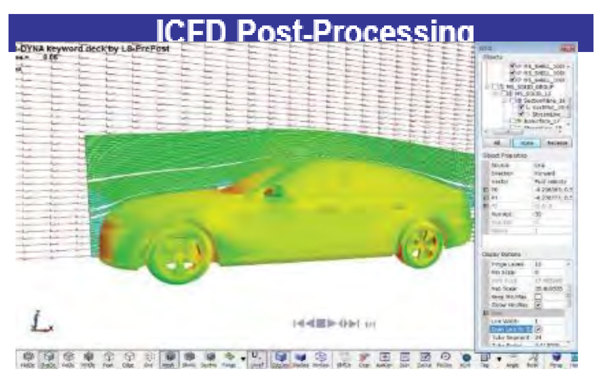
- Tools to display, reverse, and auto reverse the normal vector directions of Shells, Segments, Thick Shells, and Cohesive Elements
- Printing of High Definition pictures in a choice of formats
- Movie creation for animation sequences
- Commands, Macros and a Scripting Command Language (SCL) with C /Python API for automated Pre- and Post-Processing

Applications

- Airbag folding
- Comprehensive model checking including contact initial penetration check
- Dummy positioning
- Metal forming process setup
- Seatbelt fitting







Ansys

Ansys Blog

May 10, 2021

Simulate How Permanent Magnets Impact Medical Implants

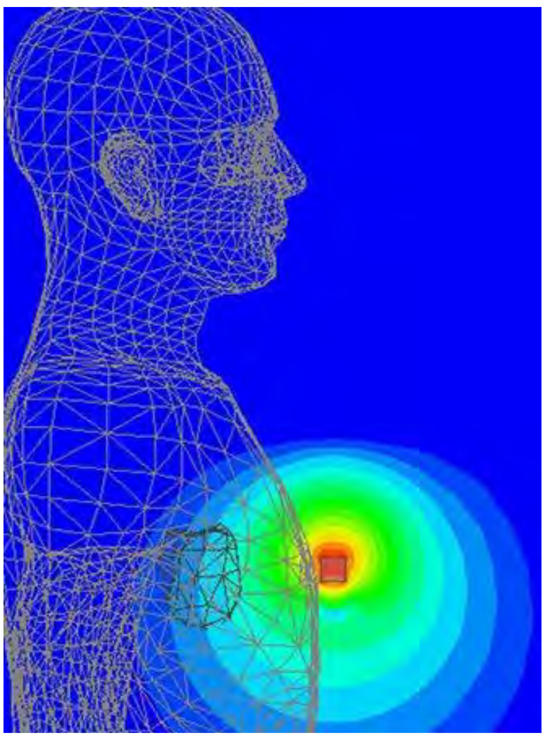


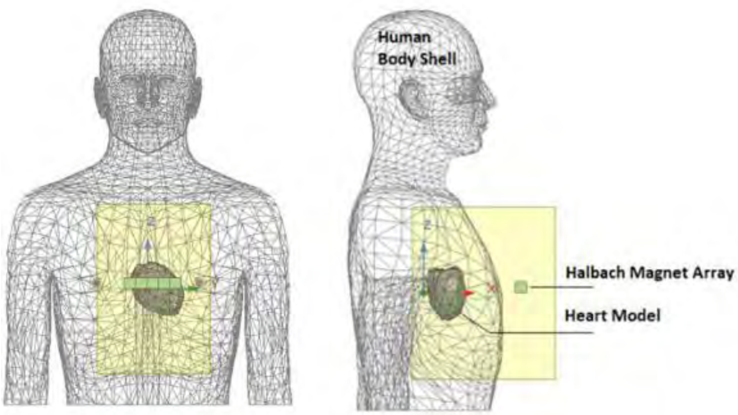
Fig1. Side view of field overlay plot on Human body, heart, and magne

Many people who have medical implants like pacemakers are instructed to maintain a certain distance from magnetic fields that may affect or interfere with these life-saving devices. Leakage fields around permanent magnets in modern consumer electronic devices like phones, laptops and tablets may extend to unwanted areas, if not contained by shielding.
Magnetic field sources may come from many places with various strengths. For example, the Earth's magnetic field ranges near 30 microtesla mT, which is very small. Permanent magnets may have fields at their surface near 1 Tesla – large enough to cause issues with some electronic devices. Some permanent magnets are used in consumer electronics devices for purposes of latching or aligning wireless charging coils. However, they carry certain risks, depending on their proximity to sensitive medical devices.

The fringing pattern around a permanent magnet will depend on its strength, size and the amount of ferrous material in close proximity, which in turn impacts the field strength at different distances and orientations of the magnets. When designing consumer electronics with permanent magnets, it is important to assess the fringing pattern near medical devices.

Evaluate Magnetic Fields for Biomedical Applications

Using Ansys Maxwell, you can define arbitrary magnet assemblies and evaluate magnetic fields in 3D space. You can evaluate the fields at different positions using a human body model, benefitting from automatic adaptive meshing in Maxwell to achieve an accurate, efficient and appropriate mesh for any geometric combination. Creating a design of experiments for position or rotation of the magnet assembly allows you to study resulting field strength near the medical device in question. This allows you to investigate different magnetic architectures, evaluate magnetic shielding options or develop guidelines for acceptable distances the magnets should be kept from the medical device.



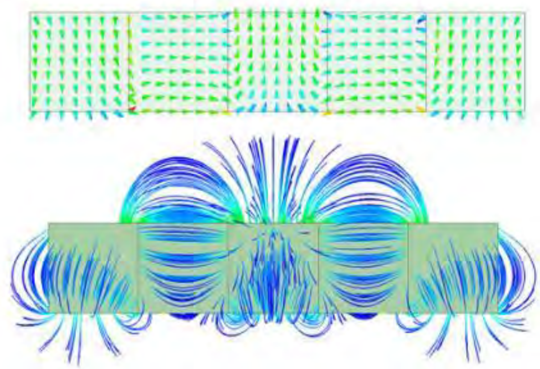


Fig. 2 (left): Front and side views of Human body, heart, and magnet models. Fig. 3 (right): Magnetization direction in magnets

The examples shown here use a Halbach magnet array, a special arrangement of magnetization patterns to both enhance and cancel magnetic fields in different areas of the magnet as shown in Fig.3. First, we set up a parametric sweep for the distance between the heart and the magnet array as well as the grade of magnet material. We define a variable of distance as "distance_x," sweeping from +20 mm to -20 mm, representing a distance far from and close to human heart model.

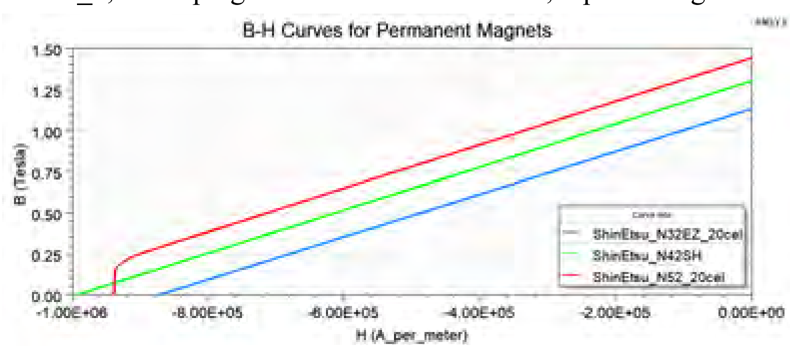


Fig. 4. B-H curves for permanent magnets

Three grades of magnet materials are swept including "ShinEtsu_N32EZ_20cel," "ShinEtsu_N42SH-R_20cel" and "ShinEtsu_N52_20cel."

Plotting the magnitude of the B field versus distance and magnetic grades shows quantitatively how the magnet field around human heart decays for different magnet materials as the permanent magnet is moved further away from the heart.

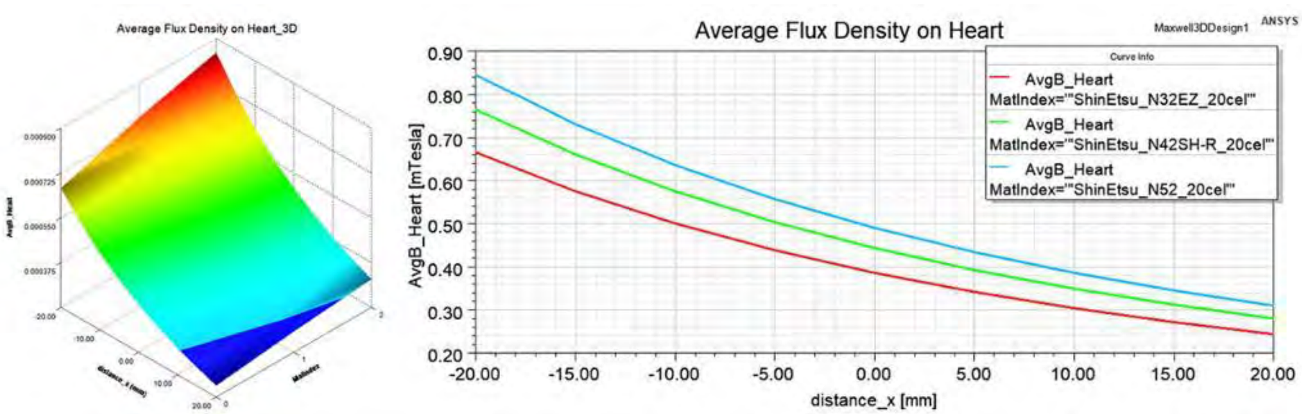


Fig. 5. Average flux density in heart vs. distance between heart and magnet and magnetic grades

Field overlay plots enable visualization of the fields around human body. The amination of field overlay plots intuitively shows the changing of magnetic flux density around heart with respect to the location of the permanent magnet.

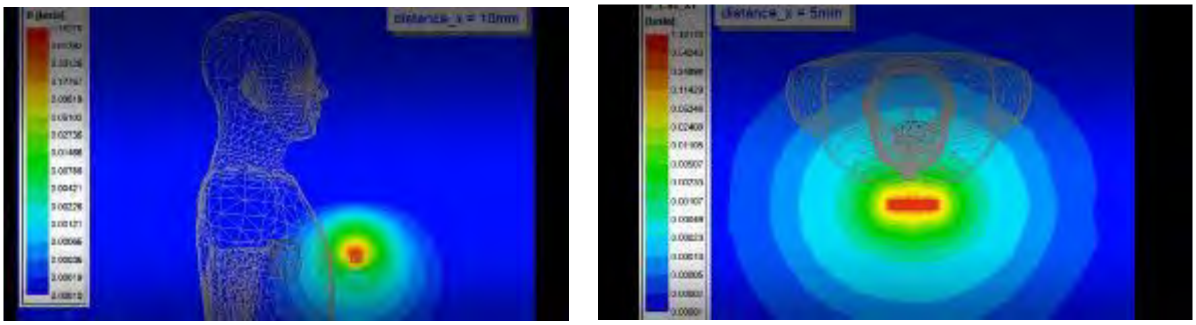


Fig. 6. Side and top views of field overlay plots of flux density in heart vs. distance between heart and magnet

Understanding the interaction of magnetic fields that result from permanent magnets used in consumer electronics and sensitive medical devices can be achieved using Maxwell. This gives you vital information about the strength of magnetic fields from magnet assemblies, as well as the fields medical devices are exposed to.

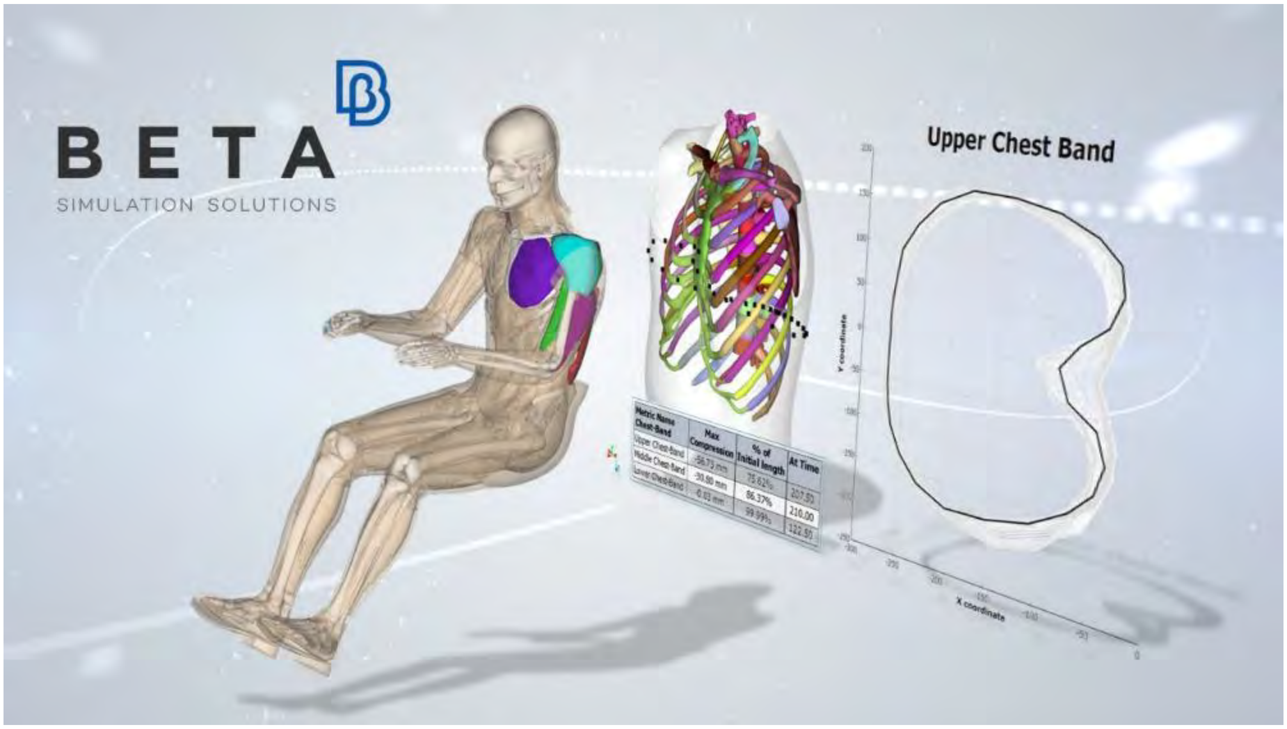
Watch this webinar to learn more about low-frequency electromagnetic solutions that help drive the design of biomedical devices.

Read in Ansys website

BETA CAE Systems

www.beta-cae.com

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…



Pre- and Post- processing for HBM models made easy.

The advent of autonomous driving with its many out of position loadcases, makes the use of HBMs in safety simulations a necessity. Positioning an HBM during pre- processing, however, has been a challenge while post- processing and analyzing the results from numerous simulation runs can be a time-consuming task. Through combining existing and new dedicated tools in ANSA and META pre- and post- processor, BETA CAE Systems push simulation with HBM to the next level and offers advanced methods and automation capabilities.

ANSA PRE PROCESSOR

The ANSA pre- processor, in its latest version offers a novel solution to this complex modeling task through the HBM positioning tool. Cooperating with GHBMC, BETA has made the positioning of the GHBMC model in ANSA as easy as positioning an ATD model. By using the advanced integrated MBD solver in parallel with the morphing algorithms, ANSA provides real-time articulation and positioning of the GHBMC model in an easy user interface. While the user just articulates the human model with the mouse in a most direct way, the biofidelic joint modelling guarantees realistic model movements and produces a ready-to-run model without the need of pre-simulation. In case the user wishes to run a pre-simulation, this can be also automatically set up. Furthermore, all tools and procedures available for

ATDs (restraints, coupled dummy-seat movement, etc.) are also available for the HBM models. Thus, positioning HBMs and ATDs are treated in the same way, within the pre- processor giving the liberty to the engineer to perform the analysis he wishes. The HBM positioning tool for the GHBMC model uses an easy-to-learn user interface

and is available to all ANSA users starting from version 21.1.0. All needed modelling metadata (biofidelic joint definitions etc.) is prepared and validated by BETA CAE Systems. Support for all available HBMs is planned for the current year along with other exciting developments that will give users even more power in handling and using HBMs in their simulations.

Following the same direction, the META post- processor, through its HBM tool, provides all the necessary functionality for post- processing results of all GHBMC Models. Running interactively or in batch mode, META creates PPTX and PDF reports including videos and images of GHBM's kinematics, strain contour plots, elements erosion identification, and injury criteria calculations (Brain CSDM, Abdominal soft tissue organs SED, etc). Moreover, time history results can be extracted from the Occupant Injury Criteria tool and compared to corresponding results of Anthropomorphic Test Devices (ATDs), while it is also easy to make comparisons between multiple simulation runs or results from different solvers.

Find out more and request the needed meta-datafile for the GHBMC M50-0 by contacting ansa@beta-cae.com

Watch a demonstration of the functionality of the HBM positioning tool on:
https://youtu.be/sdR4scGim9s

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.

Test Engineer Process: How d3VIEW Aids in Virtual Product Development

May 6, 2021 | by Elisa

d3VIEW Features   d3VIEW Platform Tips

d3VIEW offers solutions for a variety of sciences and disciplines. Test Engineers are not excluded from this as the platform has a dedicated application for physical tests that works in tandem with other features and applications to enhance product development.

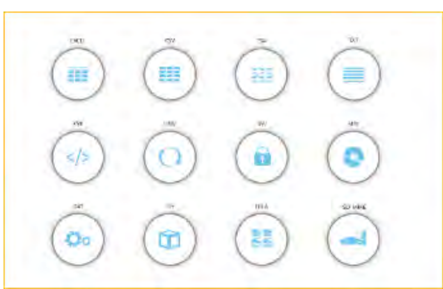
1. Prototype and Testing

The first step of the process involves the physical testing phase. The engineer executes experiments and evaluates the prototype in real-world situations.

2. Data Acquisition

Next, the engineer acquires the data from the experiments and then brings the data onto the platform by uploading it through the physical tests application.

The app supports an array of different data file types.



Supported Data Files

3. Automated Post Processing

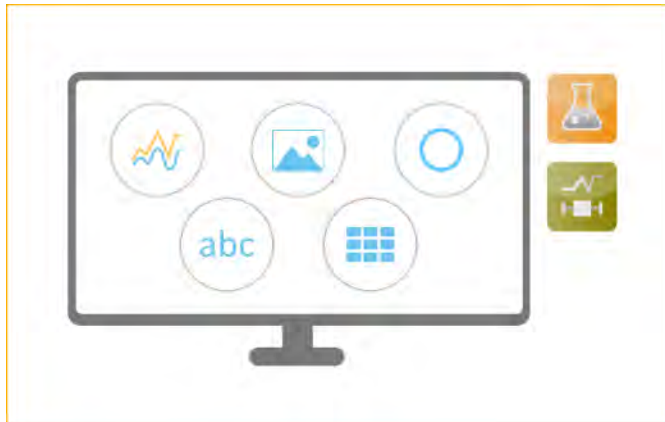
From here, d3VIEW performs automated post processing that involves the following:

1. Test quality scan: make sure data is consistent, comparable and accessible

2. Data sanitization: fix any consistencies or issues found in the scan
3. Event detection: reveal patterns and important aspects
4. Intelligent down-sampling: balance any uneven data
5. Decision-based organization and notifications: classify responses and retrieve important information

4. Test Data Visualization

The data is now ready to be viewed and visualized. The responses, or data-outputs, can be examined directly on the Physical Tests application. The engineer can add templates, transformations or overlays to extract or reveal more information about the data.



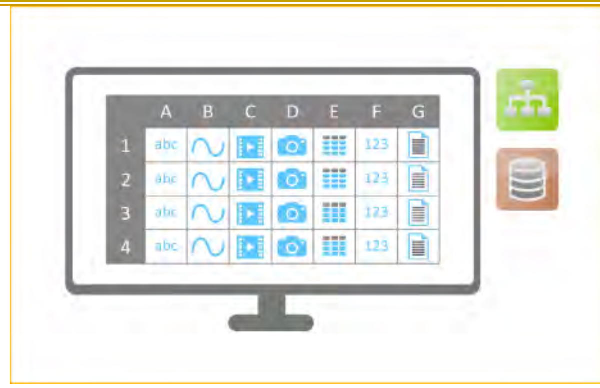
View Responses in Physical Tests

Compare Tests in Simlytiks

Even more, the engineer can compare this test with other tests by importing them into the Simlytiks applications and using its visualizers to better understand and explore the data.

5. Test Database

After uploading multiple tests, engineers deploy workflows to automate data management and create databases to store and track the data. Database records can also be compared through the Simlytiks application for further exploration.



6. Analytics, Reporting and Collaboration

PPT Builder in Simlytiks

With all the new insights and discoveries, the test engineer can effortlessly share the data mining experience with colleagues. Physical Tests, Simlytiks and Databases applications all incorporate ways to export data especially via the PowerPoint Builder which allows for creating slide layouts directly on the platform.

For more information on some of the applications mentioned above, you can visit our Platform page here.

[Read from website]

d3VIEW

www.d3view.com | support@d3view.com

Author: Christian Frech christian.frech@dynamore.de

Online and On-site Event! Submit your Abstract!

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

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

Invitation

We kindly invite all users of LS-DYNA, LS-OPT, LS-PrePost and LS-TaSC as well as our dummy models to the 13th European LS-DYNA Conference at October 5-6, 2021 in Ulm, Germany, and online.

Online and On-site

Whether online or on site - 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.

Venue

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

We will inform you about the online part as soon as possible.

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
Online speaker:	150 Euro
Industry:	640 Euro[1)] / 690 Euro
Academic:	490 Euro[1)] / 540 Euro
Online	200 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

DYNAmore part of the funded research project CO2-HyChain

The project

Traffic on Germany's roads causes around 160 million tons of CO2 every year and is thus responsible for around 20% of the country's total CO2 emissions. One extremely effective way of reducing CO2 emissions from passenger cars is to reduce vehicle weight through functional lightweight construction.

In order to reduce the weight of car bodies economically and ecologically without compromising the safety of the occupants, three different technologies and measures are currently being used. These are firstly, the increased use of ultra-high-strength aluminum alloys, secondly, aluminum-steel composite construction, and thirdly, stress-optimized tailor welded blanks (TWB) made of steel sheets with different strengths and thicknesses.

The objective

The objective of this project is to further develop solutions to produce high-strength aluminum and hybrid aluminum-steel tailor-welded blanks, which have so far been researched on a laboratory scale, through technology transfer from research institutions to industrial manufacturers and users, and to increase the maturity of the entire value chain. The widespread introduction of this technology is expected to reduce CO2 emissions from passenger cars by 15%.

In the project, highly efficient production plants for the manufacture of such hybrid TWBs are being developed, tested experimentally and their further processing safeguarded until they are ready for application. Recycling concepts are already included in the basic development.

By involving the entire value chain - from material manufacturers, small and medium-sized plant manufacturers and suppliers through to well-known automotive manufacturers - the CO2 savings potential can be realized without barriers.

The project partners

Three institutes of the University of Stuttgart, two OEMs, several technology companies and engineering firms are also involved in the project. The project is funded by the German Federal Ministry for Economic Affairs and Energy and is initially scheduled to run for three years with a budget of 5 million euros.

DYNAmore is very happy to be part of this research project and looks forward to the collaboration with all project partners.

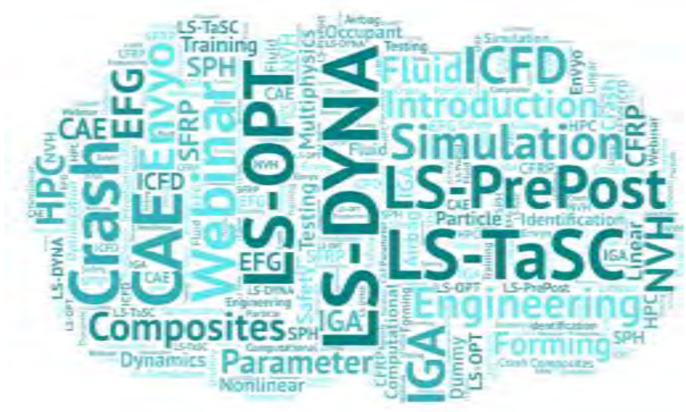
More information and contact

DYNAmore GmbH
Industriestrasse 2
D-70565 Stuttgart
www.dynamore.de
info@dynamore.de

Supported by:
Federal Ministry for Economic Affairs and Energy
on the basis of a decision by the German Bundestag

Author: Christian Frech christian.frech@dynamore.de

Webinars and on-demand Video-Seminars 2021



Online trainings in June and July

Webinars

CPM Airbag Modelling	11 June
La rupture dans LS-DYNA : d'une utilisation simple aux modélisations avancées	17 June
Discrete Element Method	23-24 June
Introduction to Passive Safety	1-2 July
Metal Forming with LS-DYNA	5-7 July
LS-OPT Optimization	12 July
LS-OPT Robustness	13 July
Basics of Structural Optimization	15-16 July
ICFD Incompressible Fluid Solver	19-21 July
Applied Forming Simulation with eta/DYNAFORM	26-27 July

Video Seminars

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

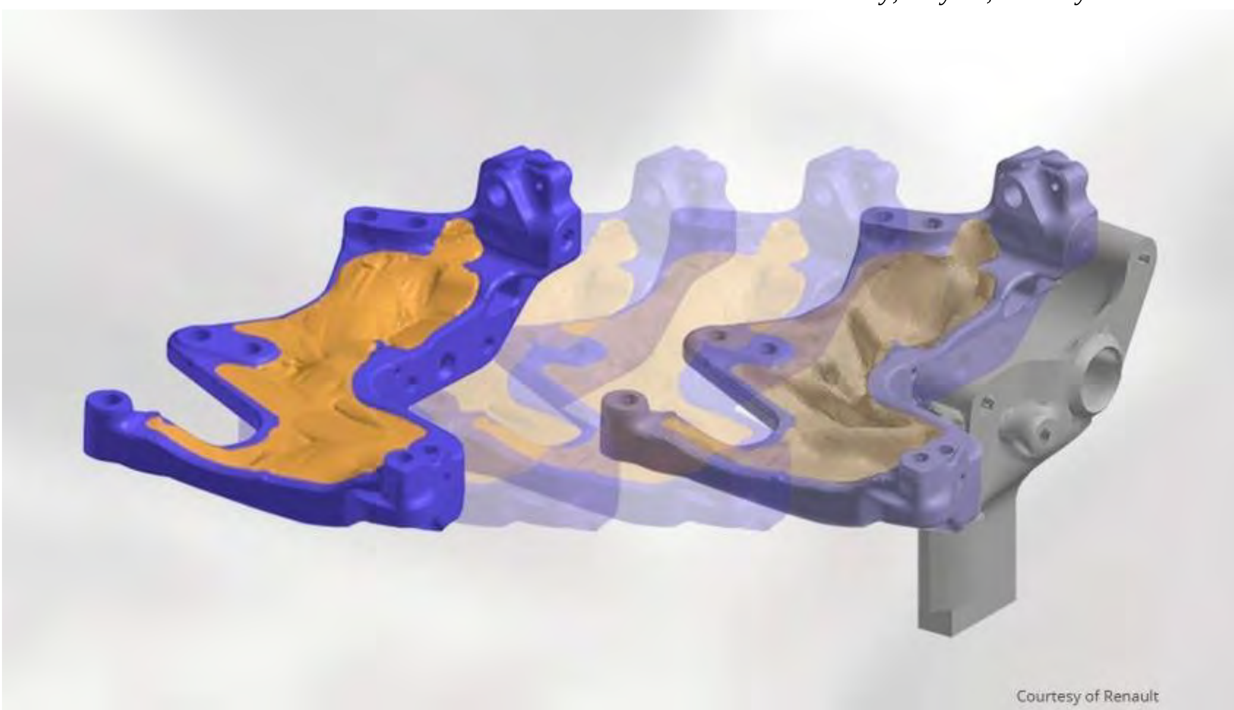
Visit our website for complete overview and registration www.dynamore.de/en/seminars

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.

Dream up Your Most Innovative Lightweight Designs with Topology Optimization

How can you go about making your product more lightweight and reducing manufacturing costs – without sacrificing performance or safety? What if switching from steel to new lightweight material isn't an option? We will address these topics and more in this blog.

Wednesday, May 12, 2021/By Sandrine Dischert



The lighter the products, the cleaner the air

According to the United States Environmental Protection Agency, the largest sources of transportation-related greenhouse gas emissions include passenger cars and light-duty trucks, including sport utility vehicles, pickup trucks, and minivans. The remainder comes from other sources, with aircrafts topping the list. Governments realized the gravity of the situation and answered the call by setting targets to reduce consumption and CO_2 emissions. So, how have auto and aircraft manufacturers learned to mitigate this problem?

Lightweighting, or reducing vehicle weight, is the most adopted method of reducing energy consumption and its by-product, CO_2 emissions. New materials like advanced high-strength steel (AHSS) and composites have been a major step in this area; they are much lighter than steel and can reduce total vehicle weight by up to 10% as opposed to traditional steel. Lightweighting is of particular interest to electric vehicle OEMs in their continuous effort to increase range.

The same rings true for aerospace manufacturers facing the two-fold challenge of further reducing fuel consumption (accounting for up to 30% of ownership cost) and the urgent need to reduce the environmental footprint by lowering emissions. While lightweight alloys have been widely deployed in aircrafts to decrease the weight-to-strength ratio, there is still a need for further reduction. Among the breakthrough technologies, Additive Manufacturing is unleashing a wider design space to redesign lightweight parts that can be manufactured at a reasonable cost and a quick time to market. It is also paving the way for innovative and complex parts, integrating several functions, and creating even more weight and cost savings.

Topology optimization makes way for new, lightweight designs

How, then, does one go about making a product more lightweight and reducing manufacturing costs – without sacrificing performance or safety? And what happens when switching from steel to new lightweight material isn't an option? How do companies using additive manufacturing arrive at an optimal design?

Let's reflect on how products are typically created. You think up a design and then you focus on bringing that design to life through ad-hoc manufacturing capabilities used on industrial production lines. But what if, instead, you were able to create a completely new and lighter design, and would identify constraints early on in the manufacturing stage?

Topology optimization is a technology used by many designers to change the shape of a part by whittling away at the material. The end goal is to find the best shape that fully respects a set of given constraints. One of the constraints can be reducing volume or weight, which makes the part lighter.

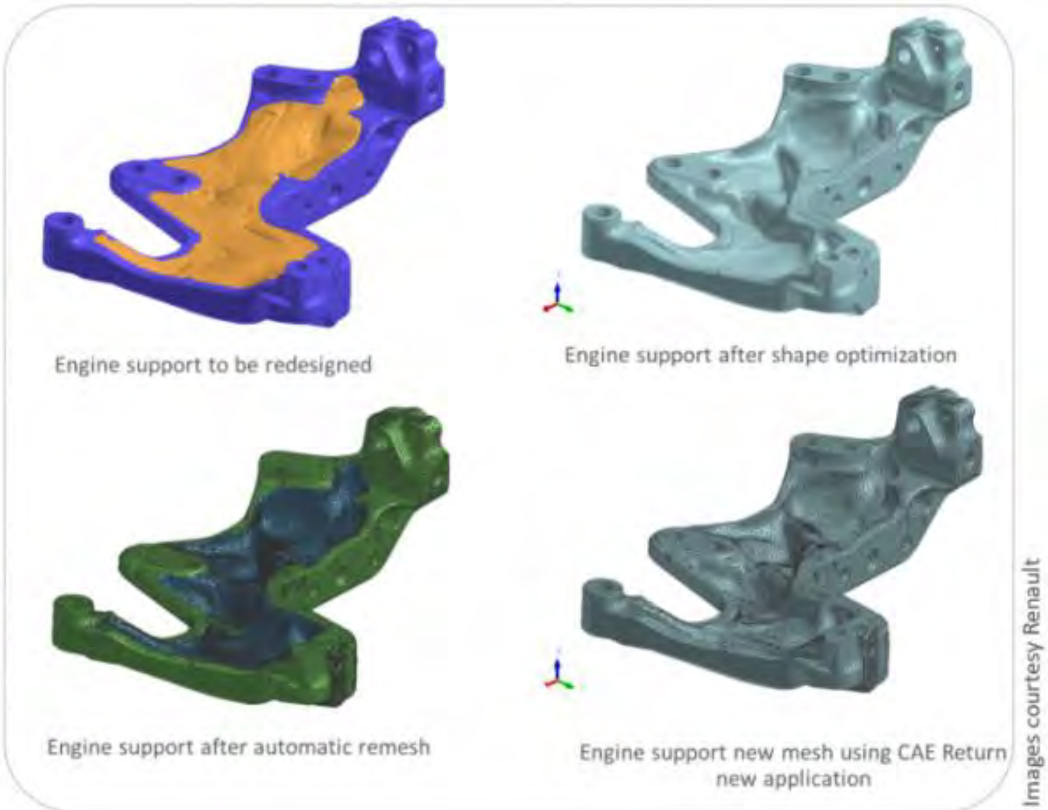
The first generation of optimization solutions uses a material density approach (SIMP method). This technology has allowed the emergence of new designs. But its limitations have encouraged the search for new methods to further push the limits of innovation in the service of design. ESI has developed a new method addressing this gap, to define an optimal shape.

Most of the widespread topology optimization solutions on the market use a material density approach (SIMP method) mainly because it is easier to implement. The problem with this approach is that, at the end of the process, the user isn't left with the optimal shape but, instead, with several solutions depending on the threshold chosen. In addition, the chosen resolution methods do not make it possible to fully comply with the imposed specifications

The topology tool with the winning combination

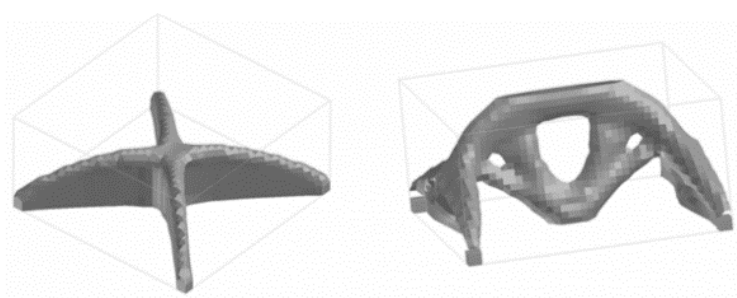
Several years ago, ESI started developing TOPAZE, a new generation solution for Topology Optimization based on the level-set method. The benefit is a unique final shape with a better-defined contour, which respects the constraints set by the designer.

The application also features a new CAE Return tool. With this tool, designers easily verify their optimization results or reintegrate the new design in their dimensioning process. This opens the door for users to perform the complete circle of the optimization process, starting from a mechanical model through to the same model with the final optimum shape.



The topology optimization process. Courtesy of Renault.

Additionally, with TOPAZE, designers account for manufacturing constraints very early on. For example, molding constraints and symmetry of the results can be applied at an early stage so that solution is guaranteed to comply with the casting manufacturing process.

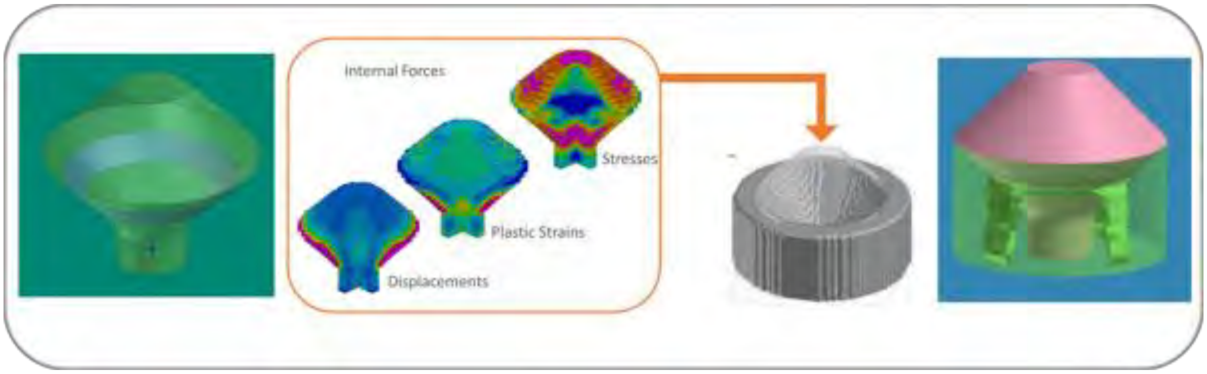


Test case optimization, with molding constraints.

TOPAZE lends a helping hand to the 3D printing world

With the advent of additive manufacturing, typical constraints of the design process are eradicated and all shapes are acceptable. Users optimize their components with TOPAZE and then export the design for later use with 3D printing.

Coupled with ESI's Additive Manufacturing Solution, TOPAZE is used to optimize the shapes of the supports needed during the manufacturing process, which happens on an experimental level.



The process of optimizing supports needed for additive manufacturing

Looking to the future, ESI is working to further define specific additive manufacturing constraints and integrate them into the topology optimization process.

ESI partners with OEMs to continue pushing the bounds of innovation

TOPAZE is an innovative application and its development is supported through participation in collaborative projects such as the TOP project (Topology Optimization Platform) with industrial partners Renault and AIRBUS, and the SOFIA project (Solutions for Industrial Metal Additive Manufacturing) with partner AddUp and many others.

The TOP Project

TOP is a four-year program with three principal objectives:

- Scientific breakthroughs in the field of optimal design
- Develop innovative and robust digital tools to deal with the design requirements of complex structures
- Demonstration of the reliability of these tools on industrial cases provided by end-user partners

Through the TOP Project, Harmonic Response, dynamic criteria, was developed and completely integrated into TOPAZE.

The SOFIA Project

SOFIA is a six-year applied research program aimed at developing a complete metal additive manufacturing value chain (powders, production equipment, processes).

This project will enable the development of new constraints related to the additive manufacturing process and integrate them into TOPAZE. The ultimate will goal will be that the final optimal shape predicted by TOPAZE can be directly manufactured via additive manufacturing solutions.

Keep following these projects and ESI for the most cutting-edge developments related to shape optimization and how it can benefit you.

For more information on optimizing your lightweight designs, download a white paper on multi-material assemblies.

For more information on the projects, visit TOP and SOFIA.

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.

Dynaform Updates

ETA is excited to have announced the release of Dynaform 6.1.0 with major enhancements and key bug corrections. The following new features were newly introduced in this release on top of the existing features from previous versions.

- Blank & Trim Line Development
- Automatic and manual Springback Compensation.
- Assembly BSE to automatically perform Unfolding and Nesting for metal stamping assemblies.
- Multi-Rows Nesting in Blank Size Engineering (BSE).
- Support for different Unit Systems.

UPCOMING WEBINARS!

Stay tuned for our next upcoming webinar: Roll Forming within Dynaform & Profil. Profil is a roll form design software for manufacturers of cold roll-formed profiles or seamed tubes from sheet metal and for designers of roll formers and tube mills.

VPG Updates

VPG Suite 2021 was released earlier this year in Asian markets. Keep an eye out for the upcoming release later this summer in North America and European markets.

Presys Updates

After implementing new requested features based on user input & feedback, Presys 2020 R1 SP1 was released on 12/09/2020.

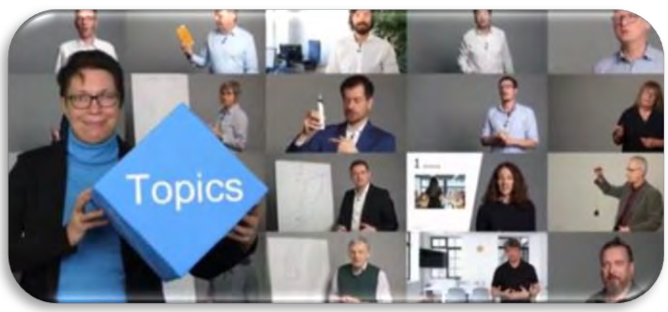
Highlights from our FEA Not To Miss Software & Engineering Solutions ISSN 2694-4707 and FEA Not To Miss Website - Sign up for our Monthly Magazine via email

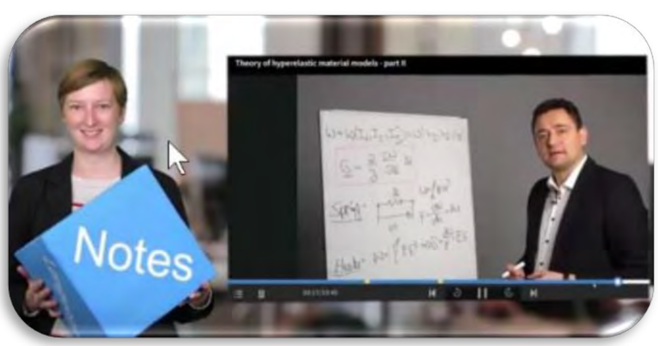
Video - CADFEM Learning

Now in English - CADFEM eLearning goes international with Brian Morris.

Now you can hear the training/seminars in English but you can still change it to hear the original German voice.

your platform for simulation training for engineers by engineers.





Hengstar Technology

www.hengstar.com

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.

Online workshop for basic of LS-DYNA

Shanghai Hengstar Technology will organized a Web Training of the basic in LS-DYNA on Mar 24 2021.

Contents:

1. Introduction of LS-DYNA
2. Control card
3. Instructions of LS-PrePost
4. Output files and data
5. Element Type
6. Time step and CPU time
7. Selection of material model
8. Hourglass
9. Loading and setting initial conditions
10. Connect
11. Boundary condition
12. Damping application
13. Contact settings
14. Finite element model evaluation
15. Exercise

Instructor:

Jun Liu (Senior Engineer)

Jun Liu was graduated from Tongji University in 2008 in vehicle engineering. He has been engaged in automobile R & D industry for more than ten years, and has rich experience in automobile safety performance research and simulation. He has accumulated a lot of pre-processing and post-processing techniques for FRB, ODB, aemdb collision modeling and simulation analysis. And He has a lot of experience in simulation and optimization of five-star vehicle development process . Also can master LS- DYNA and ansa, meta, HyperMesh, hyperview, primer and other pre-processing software.

Duration and Style:(7 hours web training): Mar 24 (9:00AM-17:00PM) **Language:** Mandarin

Contact: Xixi Fei Tell:021-61630122 mobile:13524954631 Email:Training@hengstar.com

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.

J-SEATdesigner Features

LS-DYNA preprocessor for seat design with data management and auto assembling

The recent automotive crash simulation is associated with a direct evaluation of the dummy injury criteria. Appropriate setups for a seat model and/or restraints are required to improve the accuracy of that evaluation.

J-SEATdesigner (JSD) manages various simulation cases with the model files in the database and auto assembles appropriate models based on the determined conditions. A wide range of regulations/assessments stored in the database can be loaded instead of the user input value. Measurement of H-points and pre-simulations including seating simulation are also available.

In the automotive crash simulation, the model becomes complicated and large to achieve a more accurate result. J-SEATdesigner is a powerful integrated system for seat design, supporting design engineers' challenges.

J-SEATdesigner Functions

Auto assembly of the sled analysis model Unify management of the simulation cases in the database

J-SEATdesigner can reduce the hours and efforts of engineers building a sled analysis model. Users can access the database that stores and manages a wide range of simulation cases along with the model files via the GUI. The database also stores the dummy-boundary condition combinations that satisfy the regulation/assessment. Users can use the data instead of inputting corresponding values individually. J-SEATdesigner's auto assembly will build simulation models according to the simulation cases defined in the Assembly information and output the assembled model as an LS-DYNA input file. The capability of pre-simulation, including the H-point measurement simulation and seat simulation, is also implemented.

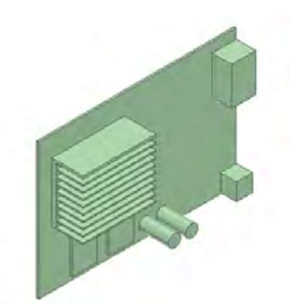
KAIZENAT Technologies Pvt Ltd is the leading solution provider for complex engineering applications and is founded on Feb 2012 by Dr. Ramesh Venkatesan, who carries 19 years of LS-DYNA expertise. KAIZENAT sells, supports, trains LS-DYNA customers in India. We currently have office in Bangalore, Chennai,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

Thermal Analysis of Graphics Card – AEDT Icepak

Objective: To study various factors affecting thermal performance of heat sink. According to the design objective, the graphics card temperature should not exceed 80°C, when air sweeps the fins at an ambient temperature of 20°C.

Modelling:

Graphics card consists of PCB, CPU, Heat sink, Two Capacitors, Two Memory and Serial port.

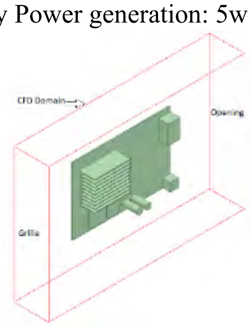


Methodology:

1) Studied various factors to be considered while designing the Heat sink.
2) Factors which affects the performance Heat transfer of Heat sink.
3) Thermal analysis of graphics card using AEDT Icepak software.

ANSYS AEDT Icepak workflow:

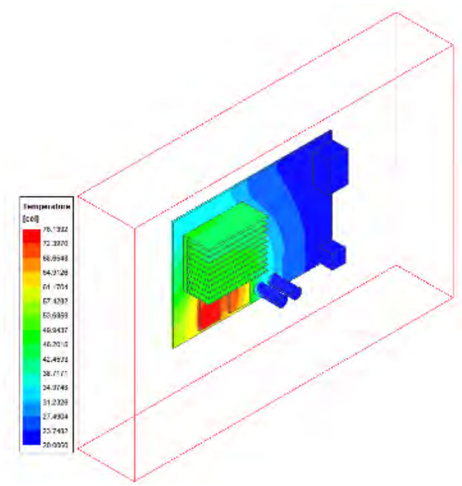
- Steady state analysis(Thermal & Flow)
- Inlet Air Opening Velocity: 2 m/s
- Outlet Grille Free Area Ratio: 0.8
- CPU Power generation: 25w
- Memory Power generation: 5w



Size of CFD domain is selected such that atmospheric condition should not change.

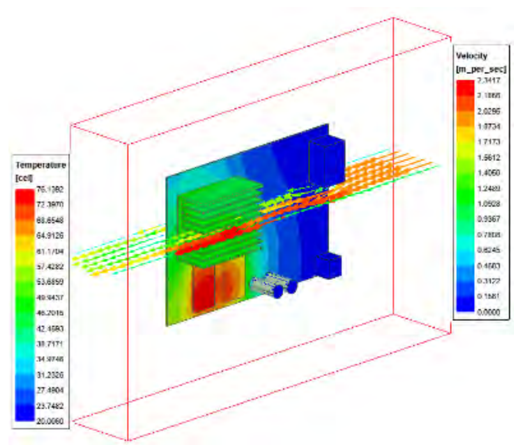
Temperature Plot:

Temperature Plot:



Maximum Temperature of the graphics card is **76.13°C**.

Temperature and Velocity Vectors Plot:



Ending Remarks:

1. Maximum temperature of graphics card is 76.13°C, which is below of designing temperature 80°C.
2. Design and type of Heat sink is important.
3. For better cooling effect maximum amount of air should passed through heat sink.

Contact: support@kaizenat.com **Phone: +91 8041500008**

On Setting up the Structured ALE Mesh

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 how to setup the S-ALE mesh.

Automated mesh generation.

S-ALE solver expects a regular, box-shaped rectilinear mesh. This regularity enables an automated mesh generation by the solver, instead of pre-processor. The whole process is simple and easy. All we need to do is to specify 1. Origin, 2. Local coordinate system and 3. Most importantly, mesh spacing along the three directions (two if 2D), through the keyword named *ALE_STRUCTURED_MESH. S-ALE solver, will read in the keyword, process it, and then generate all the nodes, elements during the initialization phase. This process is fully automated, uninterrupted and without the need of user intervention.

This practice is different from most of other Finite Element solvers and new to most of our LS-DYNA users. WHY would we want to do that? There were several reasons.

1. **Convention Compliance.** The solver utilizes faster, more efficient algorithms based on the regularity of the mesh, through a set of conventions. For example, when numbering the elements, it swipes through x first, y next, and z last. Following this convention, switching between IJK index and element ID becomes a simple calculation without any condition check: eleID=K*Ny*Nx+J*Nx+I. The easiest way to ensure that all meshes feed into the solver follow these convention, is to let the solver take full control during the whole process of mesh generation. Ironically, by putting all burdens on the solver, we avoid all the trouble of generalizing, publicizing, enforcing, error-checking these internal conventions.

2. **Efficiency.** This streamlined approach provides us with savings on several aspects. The "normal" way for a Finite Element solver to get mesh involves three steps. First, geometric data to mesh, through a mesh generator (pre-processor). Next, mesh output by mesh generator is then included in keyword and read in by the solver. Finally, the mesh is processed and translated into the internal mesh database by the solver. Now we only have one step, the geometric data is read in by the solver and immediately translated into the database. All the memory, diskspace used in the three steps are no longer needed, simply because all mid-steps are skipped.

   For a big model, say one that has 100 million elements, not only do these savings become significant, but also crucial to make the model runnable. As S-ALE elements are not generated until each processor has its own subdomain hence its S-ALE elements block only, the huge memory required by processor #0 to build its elements database and perform the domain partition is no longer needed. Because of this, S-ALE could run far much larger model than ALE when using MPP.

3. **Less Error-prone.** This process could ensure a clean mesh with much less user mistakes. During the author's 18 years work at LSTC then Ansys, he has seen so many different user mistakes on mesh. And as ALE mesh and Lagrange mesh are on top of each other, some mistakes are, if not possible, very difficult to find. For example, the ALE mesh and Lagrange mesh are aligned at certain surface. The user thought he has ensured that ALE and Lagrange meshes are not sharing any node. And we checked some, they were all good. We then switched our focus onto other parts of the input deck and tried and tried and tried… And guess what, who would think ahead of time that there is one node skipped our scrutiny? And even if we thought about that, how could we single out that one node from like a thousand nodes, without help of some specifically written scripts? Another time we even had two sets of ALE mesh, identical, on top of each other. Do not even ask how it happened. The run was all OK, except for that the fluid keeps "leaking" out of the container. Not until half a day later, just by coincidence, the mistake revealed itself. I had to use the word "revealed itself" as I did not give even a tiny little thought on the mesh. Now all those mistakes are gone. More importantly, we could focus more on studying the model and the physics.

Offsetting and Rotating the Mesh

The keyword *ALE_STRUCTURED_MESH is used to take the user-provided geometric data. That is 1. Origin; 2. Local coordinate system; 3. Mesh spacing. 1 and 2 are optional. If they are left blank, we are constructing the mesh with neither any offset, nor any rotation. And yes, that is what NID0 (origin) and LCSID (local coordinate system) do. They specify some translation and rotation motion to the mesh. In that sense, it is kind of like *INCLUDE_TRANSFORM. You could construct a S-ALE mesh, and then offset and rotate it to align this mesh with the other Lagrange structure in the model. BUT there is more to that.

This S-ALE mesh construction is not only at the first step, during initialization. Rather it is constructed at EVERY timestep, during the full course of the run. What does that mean? The mesh can move. And NID0 and LCSID control how it moves. Remember that origin and the axes are prescribed using nodes? Origin by 1 node and the local coordinate system by 3 nodes. If these four nodes are stationary, good; the mesh is stationary in space. But if some specific motion is provided to these four nodes, then the S-ALE mesh will move with these four nodes accordingly. This greatly simplifies the S-ALE mesh motion and more importantly, much more efficient than mesh motion algorithm employed by generic ALE solver.

For example, we are modeling fuel tank sloshing. The tank moves horizontally forward and backward for a few cycles. S-ALE mesh needs to follow this translational motion. What we could do it to simply pick a Lagrange node from the fuel tank structure, a node where no big deformation is expected, and then set that node as the origin. Of course, we need to offset the nodal coordinates in the three *ALE_STRUCTURED_MESH_CONTROL_POINTS card as the following.

```
*ALE_STRUCTURED_MESH
$     mshid      dpid      nbid      ebid
          3         4     40001     40001
$     cpidx     cpidy     cpidz      nid0     lcsid
       1001      1002      1003         1       101
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$      CPID      NONE      NONE       SFO      NONE      OFFO
       1001                                              0.5
$                   x1                  x2                  x3                  x4
                     1            &xminale
                &nxale            &xmaxale
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$      CPID      NONE      NONE       SFO      NONE      OFFO
       1002                                              0.5
$                   x1                  x2                  x3                  x4
                     1            &yminale
                &nyale            &ymaxale
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$      CPID      NONE      NONE       SFO      NONE      OFFO
       1003
$                   x1                  x2                  x3                  x4
                     1            &zminale
                &nzale            &zmaxale
*NODE
        1             -0.5             -0.5             0.0        0        0
```

The full model is at https://ftp.lstc.com/anonymous/outgoing/hao/sale/models/tanksloshing2/. By setting NID0 to Node 1 which is a Lagrange node on the tank, we attached the S-ALE mesh to the tank. Now S-ALE mesh follows whatever translational motion of Node 1 takes. Same thing for rotation, if we pick three orthogonal nodes on the tank and assign the resulting local coordinate system to the S-ALE LCSID, the S-ALE mesh rotates in the exact same pattern as the tank.

Mesh Spacing.

For a 3D problem, we need to construct a box shaped rectilinear mesh. So what is the least information we need to generate that box? Divide and conquer. Let us deal with one dimension at a time. And then repeat it three times. In 1D, mesh become a line containing some number of segments. The minimum information we need is 1. Starting point; 2. Ending point; 3. Number of segments. We provide this information through the keyword *ALE_STRUCTURED_MESH_CONTROL_POINTS. Below we specify a 1D mesh containing 10 evenly distributed elements (11 nodes) spanning from -0.5 to 0.5.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
  CPID
   101
          N1                    X1
           1                   -0.5
          N2                    X2
          11                    0.5
```

By the way we have an implicit convention here. Abscissa and ordinates must be ascending. Let us assume this is our intended mesh spacing along x-axis and y-axis. We only need another card to construct the z-axis mesh.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
103
1                    0.0
11                   1.0
```

And now let us construct the box mesh by putting these three _CONTROL_POINTS into the *ALE_STRUCTURED_MESH card.

```
*ALE_STRUCTURED_MESH
MSHID     DPID      NBID      EBID
1         11        10001     20001
CPIDX     CPIDY     CPIDX     NID0      LCSID
101       101       103
```

Job done. We now have a box shaped S-ALE mesh which spans from (-0.5,-0.5,0.0) to (0.5,0.5,1.0) containing 1,000 elements and 1331 nodes, with their IDs starting from 10001 and 20001, respectively. The generated mesh has a mesh ID of 1 and is assigned with a Part ID of 11 (DPID). Simple, right? But is it enough for all applications?

By taking controls away from users, the keyword should be designed to cover all possibilities. This means no matter how complicated the mesh spacing is, the design should be able to provide a mechanism to describe and process it. Now let us go wild. No reason, someone wants the following mesh along z-direction. The sizes of these 10 element are generated randomly between 0.09 and 0.11. By executing ***r=0.02*rand(1,10)+0.09*** in Octave (free version of Matlab), It comes out to be [0.099215 0.108612

0.095970 0.099048 0.103958 0.102598 0.090284 0.096290 0.096238 0.094896]. Let us keep the starting point unchanged at 0.0, the ending point is now at 0.98711 by doing a ***sum(r)***. But that is not helping. The elements in between these two nodes are assumed to be of evenly distributed, which is not true. So how do we do it? A little cumbersome but still doable. Let us get the nodal coordinates of these 11 nodes first. The first node is at 0.0. The rest 10 nodes are [0.099215 0.207827 0.303798 0.402846 0.506803 0.609401 0.699685 0.795975 0.892213 0.987110] by doing a ***cumsum(r)***. Now the ugly part. We enter them one by one into the *ALE_STRUCTURED_MESH_CONTROL_POINTS card.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103
          1              0.0
          2              0.099215
          3              0.207827
          4              0.303798
          5              0.402846
          6              0.506803
          7              0.609401
          8              0.699685
          9              0.795975
         10              0.892213
         11              0.987110
```

Tedious it is, right? To our defense, this is only to accommodate the extreme case. With help from some simple scripts and pre-processors, we believe that the workload is still somewhat manageable on the user's side. Plus we do not expect it from happening too often.

The design of this keyword should be relatively easy to understand now, after covering the two extreme cases. One is 10 elements evenly distributed between two nodes; another is 10 completely random-sized elements. There are lots of possibilities in between these two extremities. Let us study one to introduce the concept of regions.

One might want to divide the line into several regions and choose to have different mesh densities in these regions. Say we want to have three regions: 3 elements in [0.0, 0.3]; 8 elements in [0.3, 0.7]; and another 3 elements in [0.7, 1.0]. The reason behind the finer mesh in the middle, let us say, is to capture the detonation process of the high explosive placed at z=0.5 more accurately.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103
          1              0.0
          4              0.3
         12              0.7
         15              1.0
```

As shown above, the design is based on the concept of "regions". Two points define a region; and the mesh can contain as many regions as one wants. Unless progressive meshing is specified, inside a region the mesh is always evenly distributed. And in case of progressive meshing, the mesh size inside a region is gradually increased (or decreased).

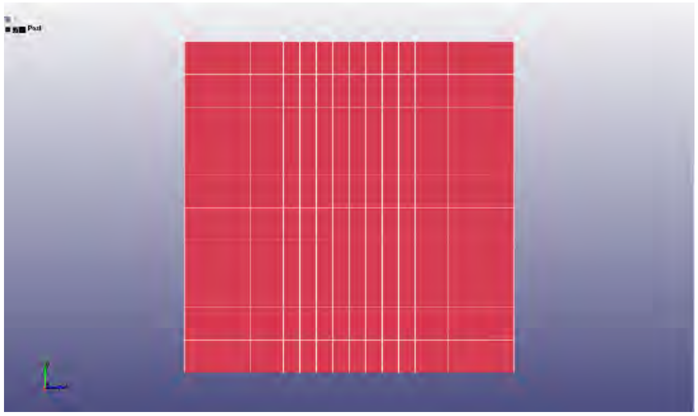


Fig 1. Z-direction mesh has 3 regions with 2 different mesh densities.

We used two mesh densities in the above mesh. An element size of 0.1 in the below and upper; 0.05 in the middle region. Most users will not stop there as we all know a sudden change in mesh density triggers wave reflection. And we DON'T want that. So is it possible to make a smooth transition between regions?

Progressive Mesh Spacing

The answer is yes. And there are several ways to do that. The first way puts all burdens on the user. Along a direction, one could design a line mesh containing several regions, with different mesh densities; and choose to have gradually increasing/decreasing element size in some regions. And then by using some pre-processor, or some scripting language like pyhton or matlab, or even hand calculation, one could derive nodal coordinates of all these nodes. And then list all these coordinates under the *ALE_STRUTURED_MESH_CONTROL_POINTS card.

The second one is to use "Ratio" option in the _CONTROL_POINTS card. Remember "region"? A region is composed of any two consecutive entries under the _CONTROL_POINTS card. Let us revisit the CPID #103 in the example we studied in the last section.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103
          1                    0.0
          4                    0.3
         12                    0.7
         15                    1.0
```

CPID #103 contains 3 regions, defined by 4 entries. First region is from entry #1 (1,0.0) and #2 (4,0.3); second from #2 (4,0.3) and #3 (12,0.7); third from #3 (12,0.7) and #4 (15,1.0). The mesh size in region 2 is one half of those in region 1 and 3. Now let us smoothen the transition from region 2 to region 3 with a ratio of increase of 0.1 from left to right. This means that in region 3, the 2^{nd} element size is

1.1*dl, the 3rd 1.1^2*dl, assuming the 1st element size is dl. That leads us to the following equation: dl*(1.1^0+1.1^1+1.1^2)=0.3. From high school algebra, (1-x^n)/(1-x)=sum(1+x+x^2+…+x^(n-1)), the above equation could be simplified to dl*(1-1.1^3)/(1-1.1)=0.3 → dl=0.090634. The three element sizes are 0.090634, 0.099698 and 0.10967, respectively. Compared with the element size of 0.05 in region 2, it appears the transition is not that smooth. So let us try using a ratio of 0.5. By repeating the above process, we get a new set of three sizes of 0.063158, 0.094737 and 0.18947. Still a little bit off as 0.063158/0.5=1.26 which means a ratio of 0.26. Then we try some numbers between 0.1 and 0.5 and it turns out a ratio of 0.4 yields a set of [0.068807, 0.096330, 0.13486], which we think good enough. Now let us add that into the entry #3 which precedes the region 3.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
    103
            1            0.0
            4            0.3
           12            0.7            0.4
           15            1.0
```

The convention here is, a mesh expansion ratio of 0.4 will be applied at the region beginning at the 12th node and ending at the 15th node. We require the expansion ratio to be put in the entry where the region starts at.

Let us deal with region 1 to region 2 transition now. Simply put, all we need to do is to put a negative expansion ratio of -0.4 at the first entry as follows. And the three sizes are, from left to right, [0.13486, 0.096330, 0.068807].

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
    103
            1            0.0           -0.4
            4            0.3
           12            0.7            0.4
           15            1.0
```

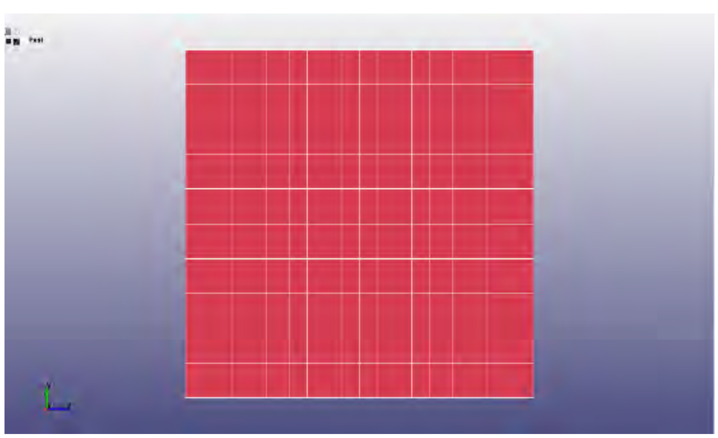


Fig 2. Progressive mesh along z-direction with a ratio of 0.4

The "ratio" parameter used here, is positive in case of expansion and negative in reduction. A special note here. A ratio of 0.4 means each time the element size increased by 40%, i.e. $dl_{n+1} = 1.4 * dl_n$. But a ratio of -0.4 does NOT mean a decrease of 40% in size from left to right. Rather it means a size increase of 40% in the reversed direction, from right to left. We could illustrate this using equation forms. A decrease of 40% in size is, $dl_{n+1} = (1 - 0.4) * dl_n = 0.6 * dl_n$. While what we want is an increase of 40% from the (n+1)th cell to the nth cell, i.e. $dl_n = 1.4 * dl_{n+1} ==> dl_{n+1} = 0.71429 * dl_n$. A common misconception one could have, is that increasing 10% along the positive direction is the same as decreasing 10% along the negative direction. But 1/1.1 is not 0.9; it is 0.90909. This misconception led to some confusions among our users and the author hopes the above explanation could help in resolving it.

The prescription of expansion ratio mentioned above is our second method of making a progressive mesh spacing. It requires some user efforts. A few iterations might be needed to find the optimal expansion ratio to provide us with a smooth transition. It works but lacks user-friendliness the authors longs for.

At the beginning of this year, 2021, one ACE expert from our ANSYS European team, came up with an idea. Instead of asking for ratio, why do not we let the user input the intended element size at certain control points? And then we could do all the tedious calculation and iterations internally to provide a smoothed transitioning mesh. And here comes the option 3. The ICASE option.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
  103              ICASE
        1              0.0
        4              0.3
       12              0.7
       15              1.0
```

Currently ICASE can be either 1 or 2. More options might be added in the future to deal with more special cases per users' request.

ICASE=1: Let us illustrate its usage with CPID #103. What we prescribe now is the element size, dl, instead of ratio. In region 2, from node 4 to 12, we have an element size of 0.05. Try with the following setup:

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
  103                1
        1              0.0
        4              0.3                 0.05
       12              0.7                 0.05
       15              1.0
```

The mesh comes out a little off from what we expected.

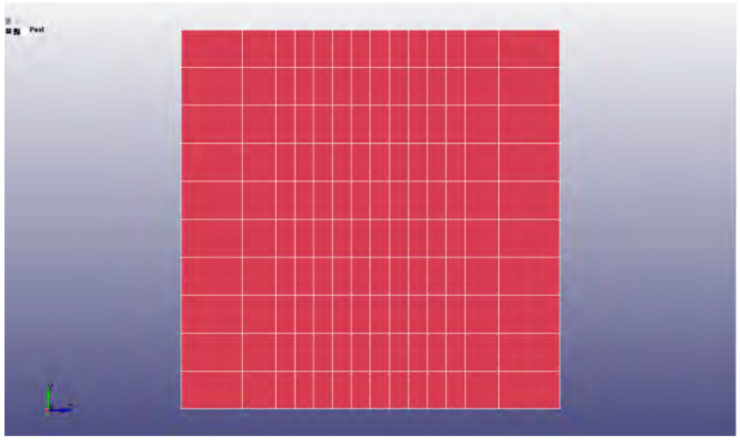


Fig 3. Progressive mesh along z-direction using ICASE=1, "not quite right".

We could see in Fig 3 that the 3rd element is of a size of 0.05, also the 12th element. What we intended to do, is to prescribe the element size from the 4th element to the 11th element. So a little modification:

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103                   1
         1                 0.0
         5                 0.35                0.05
        11                 0.65                0.05
        15                 1.0
```

And now everything is in order. The three cells have a size of 0.134047, 0.0964929 and 0.0694597, respectively. The reason is as follows. We are prescribing the element size at a node. This requires us to assign element sizes at both elements connecting to that node. In this example, the number of nodes along one direction is only 10. In cases of more nodes used, this small discrepancy will go unnoticed.

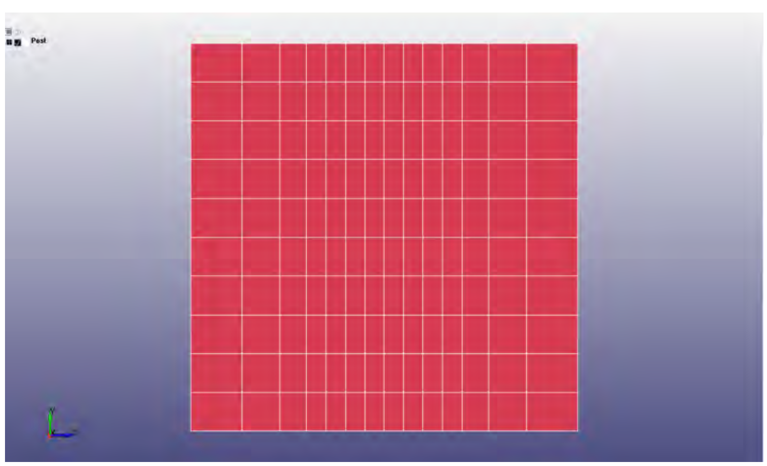


Fig 4. Progressive mesh along z-direction using ICASE=1.

There are also other ways to obtain this progressively spaced mesh. For example, we could also set the element size to be 0.13486 at the first and the last cell.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103            1
        1           0.0        0.134047
        4           0.4
       12           0.7
       15           1.0        0. 134047
```

And we will have the exact same mesh spacing as shown in Fig 4.

ICASE=2: The idea behind ICASE=2 is to build the progressive spaced mesh starting from a point and extend the mesh to the left and the right. To use this option, we need to first pick a "base node" and put it at a location by specifying its coordinate. Let us continue our play with CPID #103. This time we pick a "base node" and then extend the mesh into left and right. We put node #5 at x=0.35 and ask for a cell size of 0.05. We extend the mesh to left with mesh size gradually increased until it reaches the 1st node with a cell size of 0.134047. On the right side, we keep the cell size unchanged until the 11th cell and then starting from the 12th cell, gradually increase the mesh size from 0.05 until it reaches the right end with a cell size of 0.134047.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103            2
        1                      0.134047
        5           0.30         0.05
       11                        0.05
       15                      0.134047
```

We would end up with the exact same mesh as in Fig 4. Real case scenario we would not use such awkward number as "0.134047". Pick node #8 to be the "base node" with a coordinate of 0.5 and change the cell size to 0.12 for the leftmost and rightmost element. We get the following mesh.

```
*ALE_STRUCTURED_MESH_CONTROL_POINTS
   103            2
        1                        0.10
        5                        0.05
        8           0.50         0.05
       11                        0.05
       15                        0.10
```

Along the z direction, mesh spans from 0.05763 to 0.942366. As shown below.

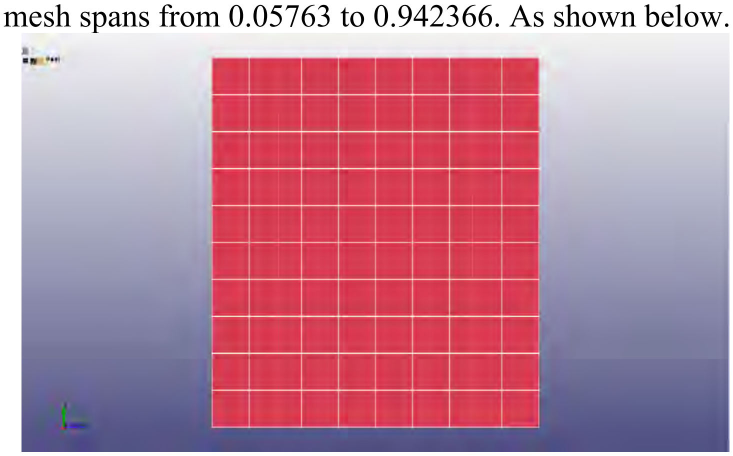


Fig 5. Progressive mesh using ICASE=2. First and last element have a size of 0.1.

For more examples on ICASE usages, please refer to LS-DYNA user's manual.

Ending Remarks

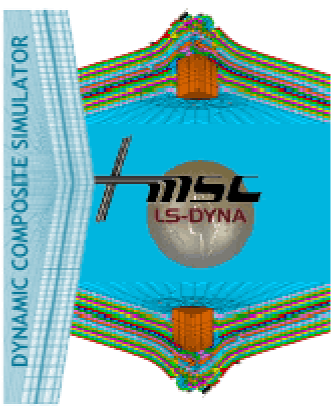
LS-DYNA ALE module has been known for its steep learning curve. Partially it was because setting up Eulerian models are intrinsically different from Lagrange models. But the design of ALE keyword cards, for sure, has caused quite a lot of confusions among our users, new and experienced.

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.

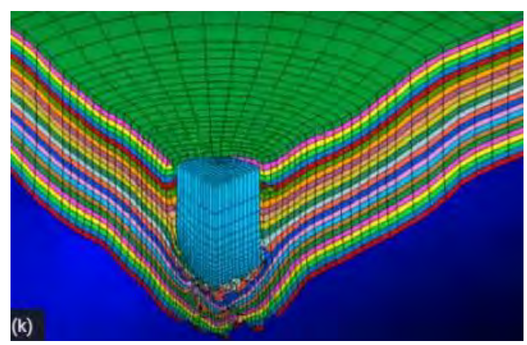
Bottom photos courtesy of TPI Composites, Inc. (left) and Seemann Composites, Inc. (right)

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

www.oasys-software.com/dyna

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.

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

Webinar to watch again: LS-DYNA - Civil/Structural applications

This webinar shows how LS-DYNA is used in the civil domain to inform the design of complex buildings. Through project examples, a variety of applications will be shown including how LS-DYNA enables advanced seismic analysis of structures, complex soil-structure interactions and helps to overcome construction challenges.

To view other past webinars, please go on to our YouTube channel.

Top Tip video:
Did you know?

Oasys REPORTER lets you create variables which can be used to automate reporting analysis results?

Click on the image to watch.

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 LinkedIn 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

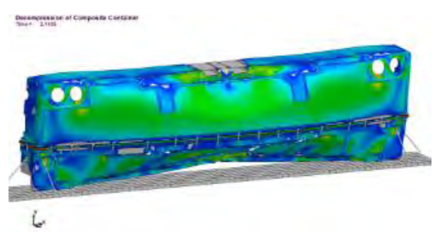
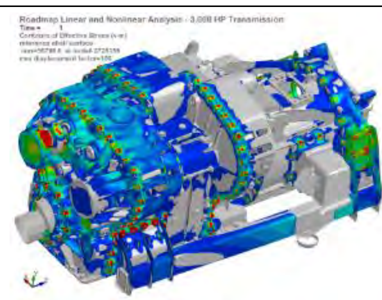
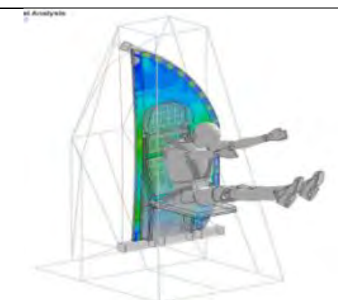
LS-DYNA has been one of Predictive's core analysis tools pretty much since we got started in 1995. It is an amazing numerical workhorse from the basic linear mechanics (think ANSYS or Nastran) to simulating well nigh the impossible. At least that is the way I feel at times when the model is not solving 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.

Predictive Engineering – Western States ANSYS LS-DYNA Distributor

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.

Platform Updates and Software Release Notes – March 2021

March 9, 2021
Aerospace, Automotive, Engineering, English, Life Science, Product Info & Tutorials
Jolie Hales

Here are a few recent platform update highlights:

Siemens Simcenter STAR-CCM+ 2021.3.1 – Simcenter STAR-CCM+ is a multiphysics and computational fluid dynamics (CFD) software for the simulation of products operating under real-world conditions. Learn more.

Altair AcuSolve, Feko and Flux 2021

- Altair AcuSolve provides computational fluid dynamics (CFD) modeling via a full range of flow, heat transfer, turbulence, and non-Newtonian material analysis capabilities. Learn more.
- Altair Feko is a leader in high-frequency electromagnetic simulation. Learn more.
- Altair Flux is simulation software focused on optimizing performance, efficiency, dimensions, cost, and weight of electromechanical equipment. Learn more.

FLOW-3D HYDRO v 1.0 – FLOW-3D HYDRO is a CFD modeling solution for the civil and environmental engineering industry, featuring a streamlined, water-focused user interface. Learn more.

AVL Simulation Suite 2020 R2 – AVL Simulation Technologies offers high-definition insights into the behavior and interactions of components, systems, and entire vehicles. Learn more.

NUMECA INCL FINE / TURBO v13.2 – A complete end-to-end solution for turbomachinery design. Learn more.

Our full software catalog is available here.

Shanghai Fangkun

http://www.lsdyna-china.com

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.

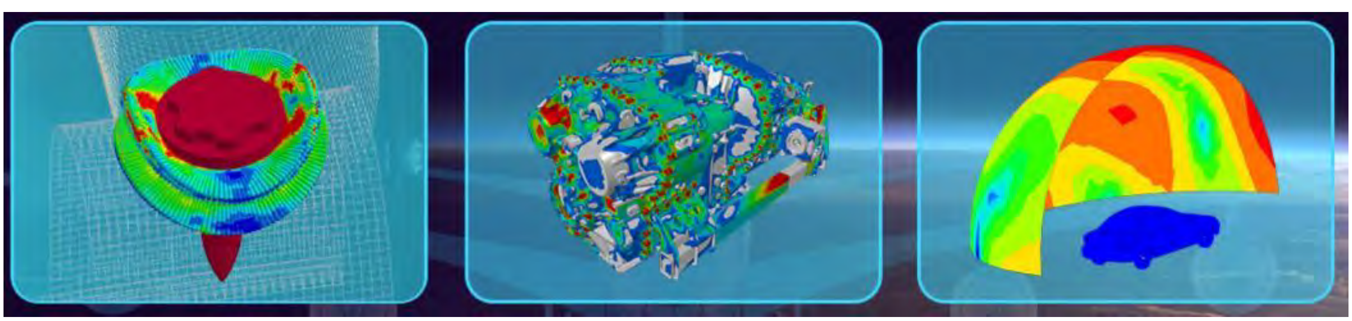
Shanghai Fangkun Software Technology Ltd

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.

Terrabyte

www.terrabyte.co.jp/english

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.

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

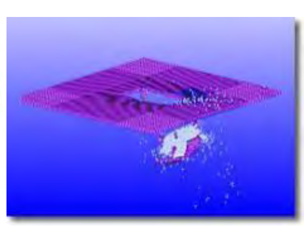
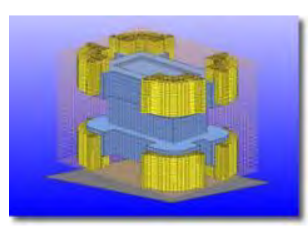
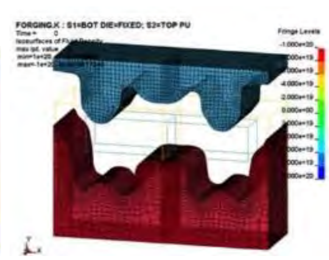
- JSTAMP is an integrated forming simulation system for virtual tool shop based on IT environment. JSTAMP is widely used in many companies, mainly automobile companies and suppliers, electronics, and steel/iron companies in Japan.

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.

Toyota Reveals TRD Camry For 2022 NASCAR Cup Series

May 05, 2021

As NASCAR Introduces Next Gen, Toyota Adds TRD Camry to Nameplate

CHARLOTTE, N.C. (May 5, 2021) – More than two years in the making, Toyota and TRD (Toyota Racing Development) today unveiled the manufacturer's Next Gen car that will take to the track in 2022 – the Toyota TRD Camry – in the NASCAR Cup Series (NCS).

Toyota has been working alongside NASCAR, as well as the other OEMs, on the development of the Next Gen vehicles for competition. The components of the Next Gen race cars are an unprecedented move for the sport, but one that looks to continue to evolve safety enhancements of the race cars and promote on-track competition.

Toyota, TRD (Toyota Racing Development, U.S.A.) and Calty Design have worked together to ensure as many body styling characteristics as possible are incorporated into the Toyota TRD Camry Next Gen so it resembles its production counterpart as closely as possible. From the grill to the spoiler and everywhere in between, the Next Gen TRD Camry race car has the most body styling attributes to its production counterpart than ever before in a NCS Camry. This is a practice that Toyota and TRD have employed since the development of the 2013 Camry in NCS competition.

"There has been a substantial amount of work put into the Toyota TRD Camry Next Gen car by all of our partners, but specifically everyone at TRD and Calty Design," said Paul Doleshal, group manager of motorsports and assets, TMNA (Toyota Motor North America). "We're thrilled to have the chance to showcase the TRD Camry to represent Toyota in the NASCAR Cup Series through this Next Gen project. This is our topline, track-inspired performance Camry and it seemed only fitting for it to take it's spot on the race track."

Toyota continues to be the only manufacturer in NASCAR to showcase three nameplates across NASCAR's three national series – the TRD Camry in the NCS, the Supra in the NASCAR Xfinity Series (NXS) and the Tundra in the NASCAR Camping World Truck Series (NCWTS). Since joining the NCS and NXS in 2007, Camrys have earned 155 NCS wins and three NCS championships along with 171 NXS wins and two NXS championships. And since Tundra's entry into the NCWTS in 2004, Toyota has claimed 204 victories and eight Truck Series championships.

"For Toyota and TRD, we're committed to the principle of continuous improvement and we believe that's reflected in this Next Gen TRD Camry," said David Wilson, president of TRD. "While we know the margins available with this new race car are smaller when it comes to adjustability, we know our race team partners and our team at TRD look forward to the challenge of learning about this car and discovering the performance opportunities that will help put the TRD Camry into victory lane."

The Toyota TRD Camry Next Gen will make its debut as the 2022 NASCAR Cup Series season kicks off at Daytona International Speedway on Sunday, February 20, 2022.

About Toyota

Toyota (NYSE:TM) has been a part of the cultural fabric in the U.S. and North America for more than 60 years, and is committed to advancing sustainable, next-generation mobility through our Toyota and Lexus brands. During that time, Toyota has created a tremendous value chain as our teams have contributed to world-class design, engineering, and assembly of more than 40 million cars and trucks in North America, where we have 14 manufacturing plants, 15 including our joint venture in Alabama (10 in the U.S.), and directly employ more than 47,000 people (over 36,000 in the U.S.). Our 1,800 North American dealerships (nearly 1,500 in the U.S.) sold nearly 2.8 million cars and trucks (nearly 2.4 million in the U.S.) in 2019.

Through the Start Your Impossible campaign, Toyota highlights the way it partners with community, civic, academic and governmental organizations to address our society's most pressing mobility challenges. We believe that when people are free to move, anything is possible. For more information about Toyota, visit https://pressroom.toyota.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

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

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

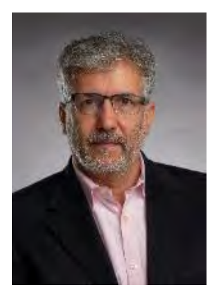
15. Composite Materials (1 day @ $500)
October 30

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).

FSI Based on CESE Compressible Flow Solver with Detailed Finite Rate Chemistry

Kyoungsu Im, Grant Cook, Jr., and Zeng-Chan Zhang

Livermore Software Technology, an ANSYS Company

Abstract

We have developed a new module of the modeling fluid structure interaction with the finite-rate chemistry in compressible CESE solver, which is based on the immersed bounday FSI method, and fully coupled with the LS-DYNA® structual FEM solver. In the CESE fluid structure interaction solver, we have two principal treatment methods, i.e., the immersed boundary method with a direct-forcing strategy and the moving mesh method. Although the moving mesh method is more accurate than the immersed boundary method, the latter is most efficient and robust when the problem involves large deformation such as a structure demolition by explosion. In the present report, we have demonstrated most practical fluid structure interaction problems by using the immersed boundary method with chemistry: i) shock-induced combustion in front of a spherical projectile moving at supersonic speed, ii) the blast refief wall simulation in methane and air mixture (CH_4/Air), and iii) the fracture of the shell and solid structures by high explosive spots in an H_2/O_2 premixed environment. The results are validated with existing experimental data and descriptions of the keyword setup are provided in details for users.

1 Introduction

The fluid structure interaction (FSI) problems at which either an internal or external fluid flow interacts with one or more solid stuructures play prominent roles in many scientific and engineering fields. However, a comprehensive study of such problems remains a challenge due to the strong nonlinearity and multidisciplinary nature [1-3]. Furthermore, analytical solutions for most FSI involved problems are not possible to obtain, whereas experiments in laboratory are limited in their scope. Thus, the numerical simulation models are greatly necessiated to investigate the fundamental physics involved in such complex interaction between fluids and solids.

LS-DYNA® is a well-known general-purpose highly nonlinear, transient dynamics finite element mechanical software capable of simulating complex real world problems, and is widely used in automobile, aerospace, construction, manufacturing, and bioengineering industries. By coupling with the LS-DYNA® FEM structural solver, we developed a CESE-FSI solver with detailed chemistry for solving the safety and explosion problems. Currently, we provide two FSI technics: a moving mesh method (MMM) and an immersed boundary method (IBM). In the MMM, the fluid mesh follows the structural interface motion and the interfaces between the fluid and the structure are treated as moving solid walls. The MMM is a natural extension of the CESE framework and thus it can be applied to solve more accurate problems such as a fluid boundary layer calculation. However, it takes more time to solve a problem since it requires additional calculation time for the mesh motions. Therefore, it is most suitable for small deformation problems. In the IBM, the fluid mesh is fixed and the structure moves through the CESE mesh. FSI interfaces are detected by the CESE solver and treated using a direct-forcing pulsing ghost-fluid approach. So, it is very robust and can handle large deformation problems such as explosions. To demonstrate the IBM FSI with the chemistry solver, the present report slected three practical applications: i) shock-induced combustion (SIC), ii) the blast relief wall (BRW), and iii) the fracture of the shell and solid structure problems.

One of proper problems using IBM FSI is the shock-induced combustion (SIC) in front of a spherical projectile which is moving supersonic speed. The SIC is the self ignited combustion phenomena of premixed gas induced by the flight of supersonic projectile in a combustible gas mixture. Thus, the SIC flow field is characterized by the hypersonic flight and finite rate exothermic chemistry behind the shock wave. Among the various features of SIC, periodically unstable regime around a blunt body would be a most interesting case due to ist naturally oscillating phenomena [4].

The flammable gas could be released due to internal structure ruptures in a off-shore plant and then, such gases will entrain air to form an explosive gas mixture. An accidental gas explosion may occur when the gas mixture meets a suitable ignition condition in a local position. In such a scenario, the BRWs or explosion relief panels reduce consierable damages in structures which may have the potential for explosion in an industrial plant located either on-shore or off-shore [5].

The fractures of the shell and solid structures by high explosive spots can possibly occur in nuclear reactors where the flammable hydrogen-air mixture (H_2/O_2 premixed environment) can be formed easily around the coolants.

The present paper will give the results of the explosion tests using CESE FSI solver with detailed chemistry models such as H_2/O_2 and CH_4/Air reactions.

2 FSI Model

Based upon the treatment of meshes, the numerical procedures to solve these FSI problems can be classified into two approaches: the conforming mesh methods and non-conforming mesh methods. The method with a conforming mesh considers the interface conditions as physical boundary conditions, and it is required that their meshes conform to the interface. Owing to the movement and/or deformation of the solid structure, mesh updating (even re-meshing) is needed as the solution is advanced. So, this method is extremly time consuming and thus not good choice to the explosion problems.

In the non-conforming mesh methods, the fluid-structure interface conditions are treated as constraints on the model equations so that a non-conforming mesh can be employed. As a result, the fluid and solid structural equations can be conveniently solved independently from each other with their respective grids. Moreover, re-meshing is not necessary. LS-DYNA® IBM is a representative method in this category. Here, a force-equivalent term is added to the fluid equations to represent the fluid-structure interaction and to avoid mesh update in the numerical procedure.

Such a IBM was originally developed by Peskin [6] for studying blood flow through a beating heart, and has since been extensively studied and applied to a wide variety of FSI problems. One of its variants is the direct forcing method [7]. By simply imposing the no-slip condition on the fluid momentum equations at the interface, this method directly evaluates the FSI force from the fluid equations with the incorporation of the known structural interfacial velocity through interpolation. One advantage of this method is that it avoids the numerical stiffness usually encountered in various penalty forcing techniques, also it can be used for compressible flows.

In CESE IBM FSI solver, the fluid equations are solved using the CESE method and the solid structural equations are solved by the LS-DYNA® FEM program, while for the elements near the fluid-structure interface it is treated with the direct forcing immersed boundary method, plus the ghost fluid method (GFM) [8]. The fluid solver gets the displacements and velocity of the interfaces from the structural solver and feeds back the fluid pressures to the structural solver as external boundary conditions as shown in

Fig. 1.

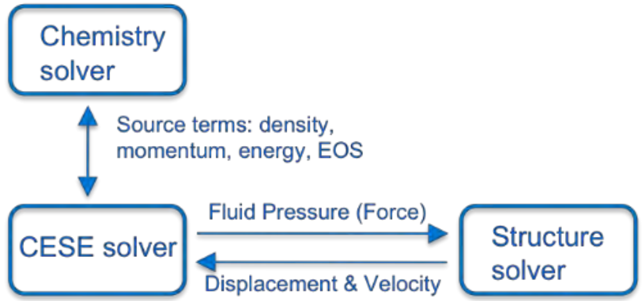


Figure 1 Interface data transfer

The followings are the main steps in the procedure:

1. Generate meshes for the fluid and the structure; those meshes are independent of each other. Then, initialize the flow field and structural state variables.
2. Calculate the shortest distance from the structure boundary to an individual solution element (SE) point. Using the distance, the SE points can be separated as the inner fluid one (A in Fig. 2), nearby fluid point (B in Fig.2), ghost fluid point (C in Fig. 2), or potential fluid point (D in Fig. 2).
3. Solve the solid structural equations using the LS-DYNA® FEM solver based on the structure loading and the fluid-structure interface boundary conditions obtained from the fluid solver.
4. Get the updated fluid-structure interface location and interface velocity from the structural solver.
5. Update the shortest distances (only for the near interface SE points) to get a new inner fluid point, nearby fluid point, ghost fluid point classification.
6. Update the fluid solution for inner fluid points using the CESE scheme.
7. Update the fluid solution for nearby fluid points using direct-forcing IBM, or the regular CESE scheme.
8. Use the ghost fluid method to treat the ghost fluid points since some of them may be needed in the next time step inner fluid point solution calculations.
9. Feedback the fluid pressure to the structural solver as external force acting on the fluid-structure interface as a boundary condition.
10. Go back to step 2 if the termination time has not been reached.

Note that in some cases, the sub-iterations between the fluid and structure solution may be needed in order to make the FSI solver more stable. Also, the conjugate heat transfer solver can be added in the above procedure, in such case, the step 4 need to get one more structural variable i.e., temperature, from the structure solver and also in step 9 the heat flux of the fluid should feedback at the FSI interface.

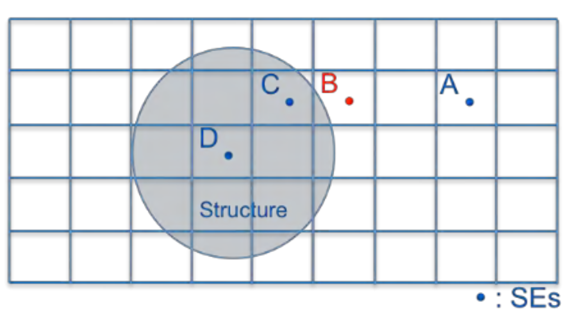


Figure 2 Different SE points in computational domain

The above FSI treatment takes advantage of both the LS-DYNA® FEM structural solver and the CESE method. It is very efficient and robust, and can be suitable for large deformation problems.

3 Results and Discussion

3.1 Shock-Induced Combustion

Figure 3 shows the schematic of the simulation domain of the SIC driving by supersonic bullet and its corresponding shadow image from Lehr's research [4]. Initially, the domain is filled with a stoichiometric hydrogen-air mixture at a pressure of 42663 Pa. The projectile body has a 15 mm diameter hemispherical projectile and cylindrical afterbody. The projectile is flying over the detonation velocity of the gas mixture and the reaction starts near the centerine of the body showing a shock-deflagration system.

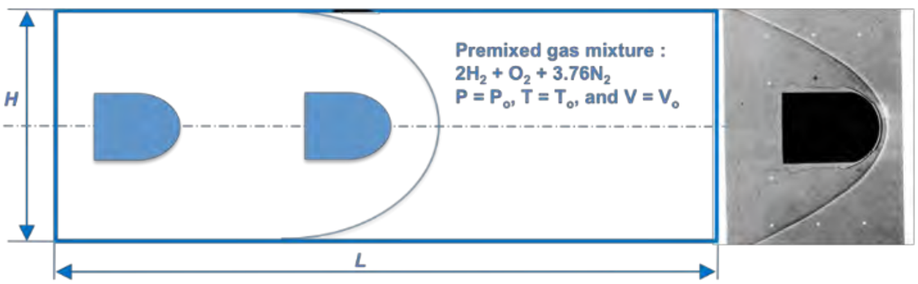


Figure 3 Schematic computational domain of the shock-induced combustion by superonic bullet.

3.2 Blast Relief Wall

The purpose of the blast reflief wall is to vent the combustion gases and pressures resulting from a deflagration within an enclosure so that structural and mechanical damage is minimized. Figure 4 shows a blast chamber with a hinged door presented flammable gas mixed with air. The selected gas mixed with air is methane(CH_4) since it is widely tested in offshore plants and also highly explosives. The combustible limits of the metane in terms of the mixture volume range 5~15 %. Test chamber was desigened as following the NFPA regulation [9]: a cube has the volume of 4 m^3 and the area of a blast relief wall having a material property of SUS 316 or Aluminum is 0.399 m^2. An electric ignitor of 100 J is set at a thrid of the position along the centerline of the chamber shown in Fig. 4.

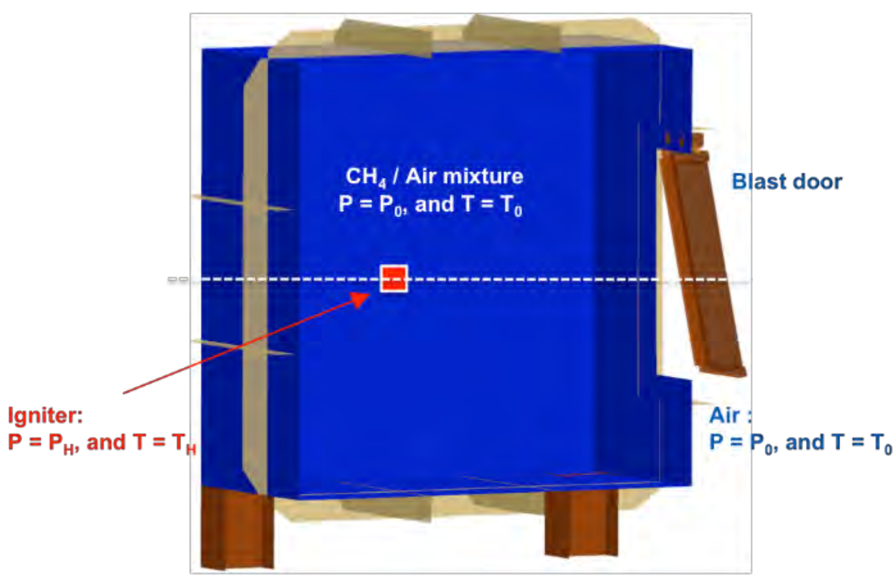


Figure 4 Simulation model of the blast relief wall and blast chamber.

3.3 Fracture of the Shell and Solid Structures

In a containment structure of the nuclear facilities, the hydrogen gas and other volatile radiaactive nuclides were collected and eventually reacted with oxygen, resulting in hydrogen explosion that destroyed the roof and deformed the internal structures. Fracture model of the shell and solid structure is selected here to test our FSI solver with detailed chemistry. Figure 5 shows the heme spherical dome structure constructed inside with bulidings which consist of the solid elements. A stoichiometric high explosive of the H_2/O_2 mixture with a pressure of 30 Mpa and a temperature of 3000K is set at the center position of the domain. The premixed gas mixture (H_2/air) initialized with the lower flammable limmit of the hydrogen gas (4 %) and air is designed outsied of the hemispherical dome.

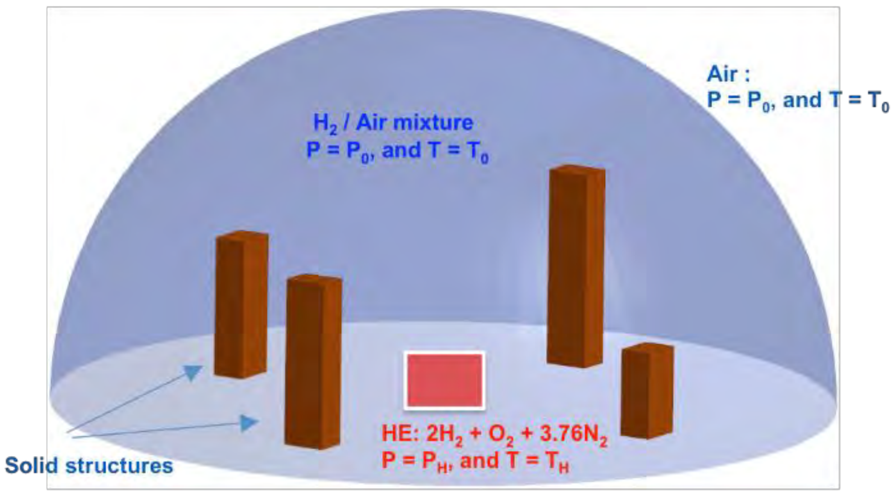


Figure 5 Fracture model of the shell and solid structures by a local high explosive gas.

4 Summary

In the present study, we have demonstrated the performance of the FSI with chemitry for three different practical problems: i) shock-induced combustion in front of a spherical projectile moving supersonic speed, ii) the blast wall simulation in methane and air mixture(CH_4/Air), and iii) fracture of the shell and solid structures by high explosives in H_2/O_2 premixed environment. The results between the numerical simulation and an experimental data set were compared, showing excellent agreement and also the keyword set up for users was illustrated in details. With these developments, we strongly believe that the present solver should provide for users to solve more practical applications in safety and explosion industries.

References

[1] Chakrabarti, S. K. (Ed.), “Numerical Models in Fluid Structure Interaction“, Advances in Fluid Mechanics, Vol. 42,WIT Press, 2005.
[2] Dowell, E. H. and Hall, K. C., “Modeling of fluid-structure interaction“, Annual Review of Fluid Mechanics, Vol. 33, 2001, pp. 445-490.
[3] Morand,H. J.-P. and Ohayon, R., “Fluid-Structure Interaction: Applied Numerical Methods“, Wiley, 1995.
[4] Lehr, H. F.,“Expeiments on Shock-Induced Combustion“, Astronautica Acta, Vol. 17, 1972, pp. 589-597.
[5] Hjertager, B. H., Bjorkhaug, M., and Fuhre, K., “Explosion Propagation of Non-homogeneous Methane-Air Clouds inside an Obstructed 50 m3 Vented Vessel”, J. Hazardous Materials, 19, 1988, pp. 139-153.
[6] Peskin, C. S., “Numerical analysis of blood flow in the heart,” Journal of Computational Physics, Vol. 25, 1977, pp. 220-252.
[7] Mohd-Yusof, J., “Combined immersed boundary/B-spline methods for simulations of flow in complex geometries,” Annual Research Briefs, Center for Turbulence Research, Stanford University, 1999, pp. 317-327.
[8] Fedkiw, R., Aslam, T, Merriman, B., and Osher, S., “A non-oscillatory eulerian approach to interfaces in multimaterial flows (the ghost fluid method),” J. Comput. Phys., 1999, 152:457.

NFPA68,“Standard on Explosion Protection by Deflagration Venting“, 2007 Edition.

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.

get it right®

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.

ANSYS 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 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.

Cloud - HPC Services - Subscription RESCALE

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

www.esi-group.com

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.

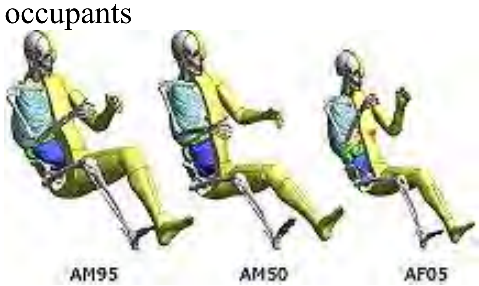
Key collaboration capabilities:

- 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

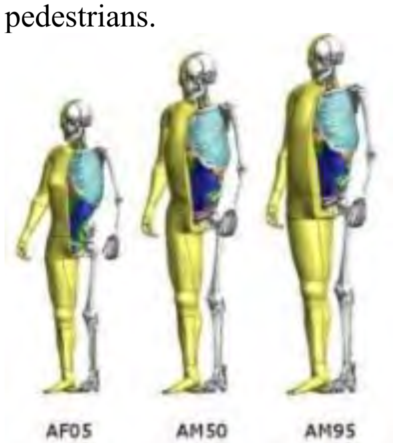
TOYOTA - Total Human Model for Safety – THUMS

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.

ATD - Human Models - Barrier

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.

Social Media

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