fea INFORMATION Engineering Solutions

Ansys LST

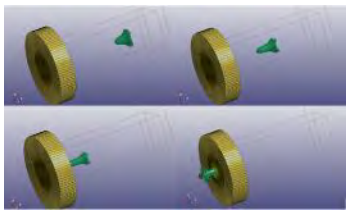
Issue 6, June 2021

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

ANSYS

DYNAmore

LST



OASYS

LS-DYNA@ New Feature and Application

- **On Structured ALE Mesh Trimming**
- **A new fatigue solver based on modal dynamics in Ansys LS-DYNA**

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

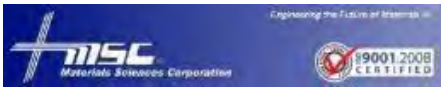


Table of contents

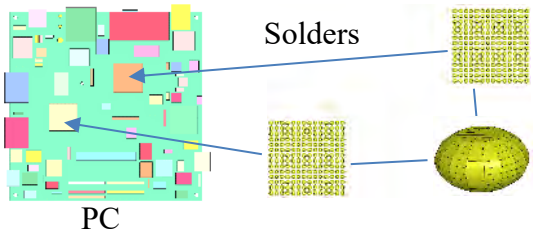
LS-DYNA® Computational and Multiscale Mechanics

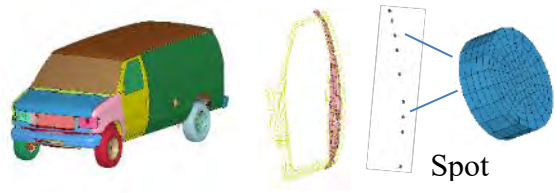
Intelligent Manufacturing, Advanced Material Design & Integrated Structural Analysis

LS-DYNA® integrates advanced finite element and machine learning algorithm for solving some of the most challenging multiscale problems in manufacturing processes, material design, and structural analysis. Such problems typically involve large deformation, material failure and separation, and/or crack propagation phenomena. The newly released features include two-scale co-simulation, machine learning based short fiber composite material model, RVE package for multiscale material modeling, ISPG for solder reflow process and multi-stage SPG analysis for failure induced manufacturing process.

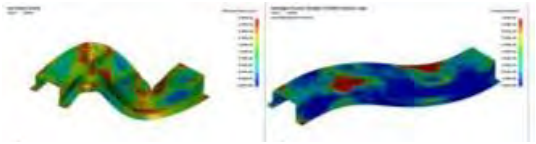
Two-scale co-simulation

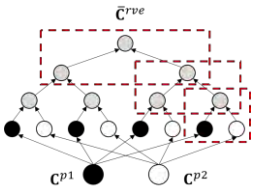
- Geometric multi-scale modeling
  - Sub-cycling
  - Non-conforming cross-scale coupling
  - MPI-based data exchange
- Applications
  - PCB Solders
  - Joints (rivet, spot weld, FDS)
  - Reliability, performance, crashworthiness



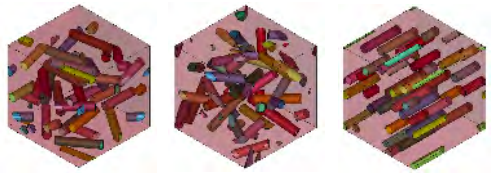


Analysis using DMN material





Dataset



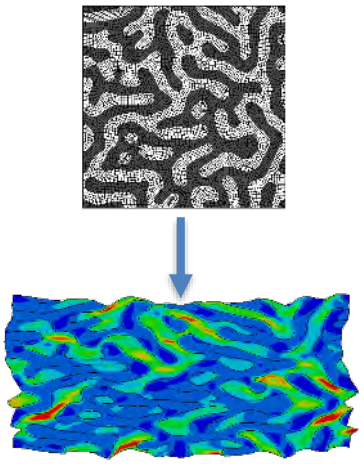
Short Fiber Reinforced Composite

- Data-driven material model
- Offline training data
  - RVE high-fidelity analysis
  - Fiber orientation
  - Fiber concentration
  - Deep material network (DMN)
- Online DMN based material model
  - User-defined base material: Fiber and matrix
  - Transfer learning for different fiber orientation and volume fraction
  - Nonlinear material behavior
  - Arbitrary loading path

Announcements

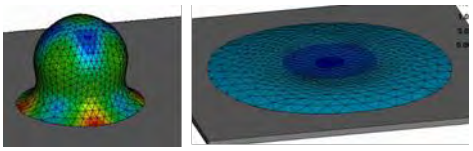
Representative Volume Element (RVE) Package

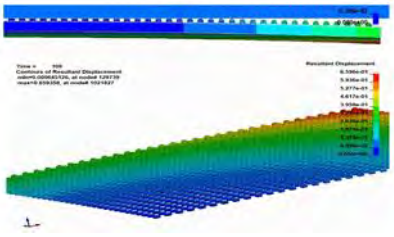
- Predict macroscopic constitutive behaviors (2D & 3D)
- Automatic periodic boundary condition set-up
- Support many nonlinear material models
- Capture finite deformations of microstructures
- Applications: multiscale virtual design & characterization
  - Fiber reinforced composites
  - Particulate composites
  - Laminar composites
  - Polycrystalline aggregates
  - Single phase or multiphase porous media



Incompressible Smooth Particle Galerkin (ISPG)

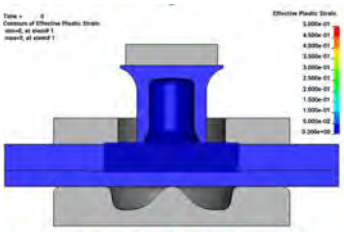
- Fully implicit formulation
- Model the surface tension and wall adhesion efficiently
- Capable to simulate the solder reflow with complex models, e.g. solder mask defined (SMD) pad & NSMD
- Coupled with implicit thermal and structure solvers
- Scalable computation for large scale thermal-mechanical PCB (warpage effect) with solder reflow

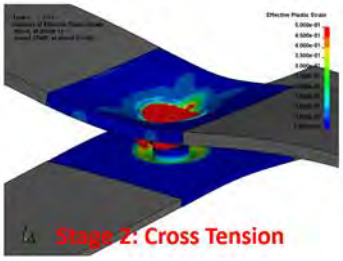


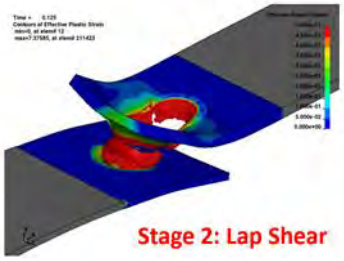


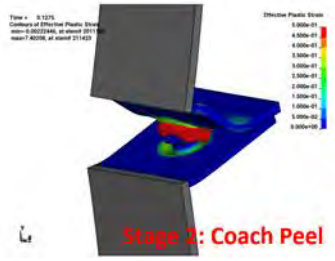
SPG Two-stage analysis

- Material failure in manufacturing process and performance test
- Seamless transition between two stages









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

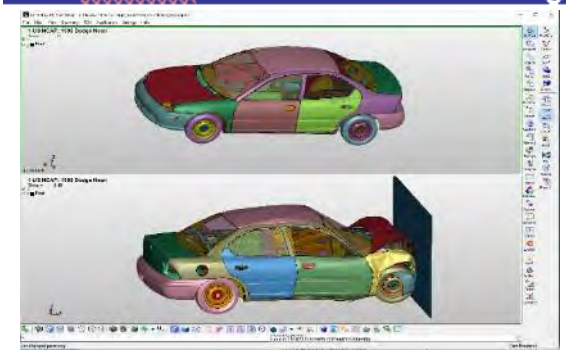
Other General Functions

- 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

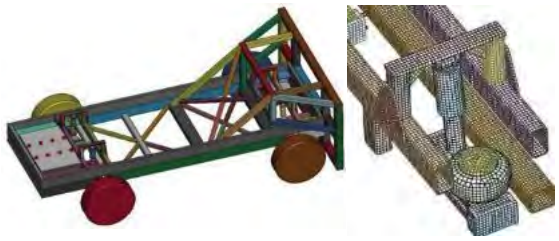
Applications

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

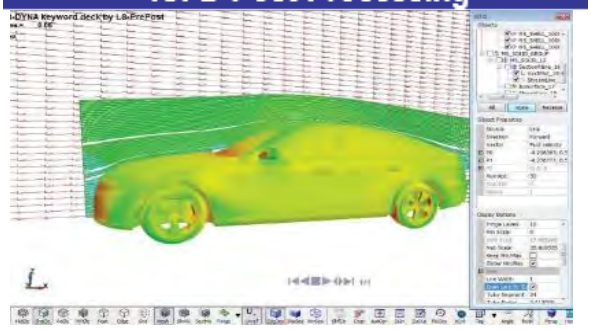
LS-PrePost Pre-and Post-Processing



LS-DYNA Geometry and Meshing



ICFD Post-Processing

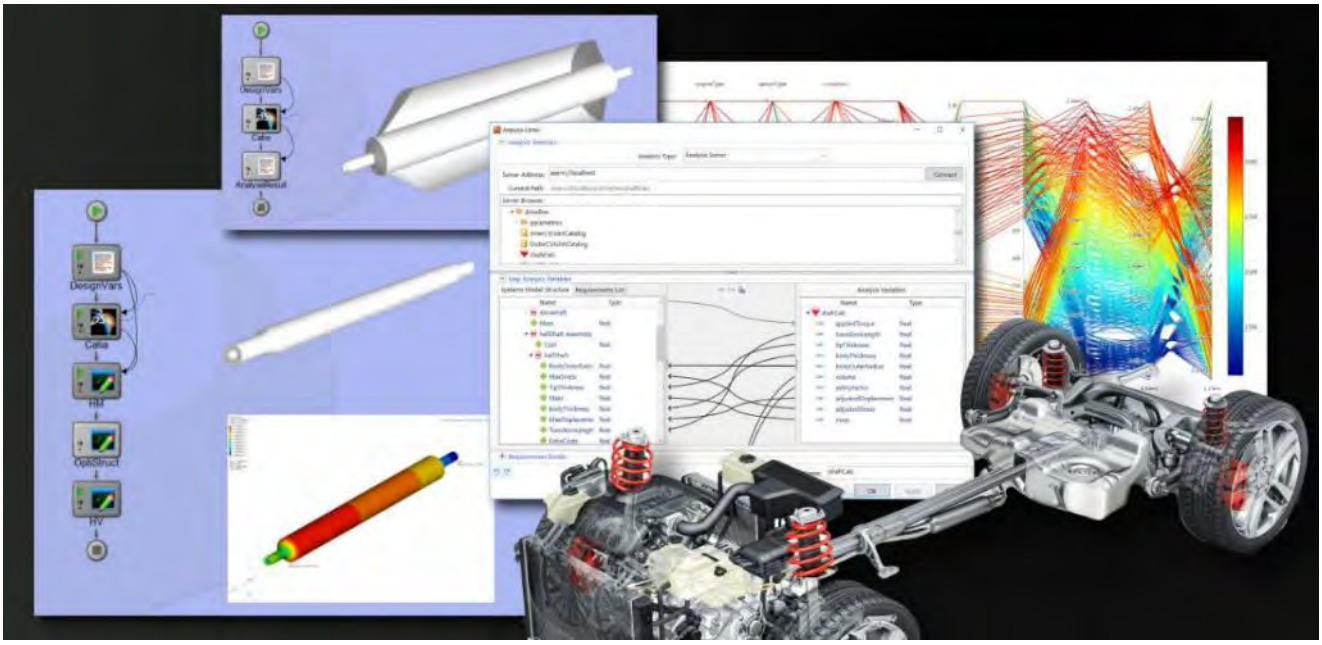


Ansys Blog

June 7, 2021

How to Design a Complex System With Model-Based Systems Engineering

Authors: Anthony Dawson, Anthony Dawson, Vice President and General Manager, Ansys
Jane Trenaman, Senior Director, Ansys, and former CEO of Phoenix Integration



The benefits of using simulation early and often in the product development process are clear. You can verify requirements and begin studying and exploring design alternatives before prototyping and investing time in designs that ultimately won't meet requirements. What may not be clear is how to integrate all the simulation tools that different engineering teams use during the product development process.

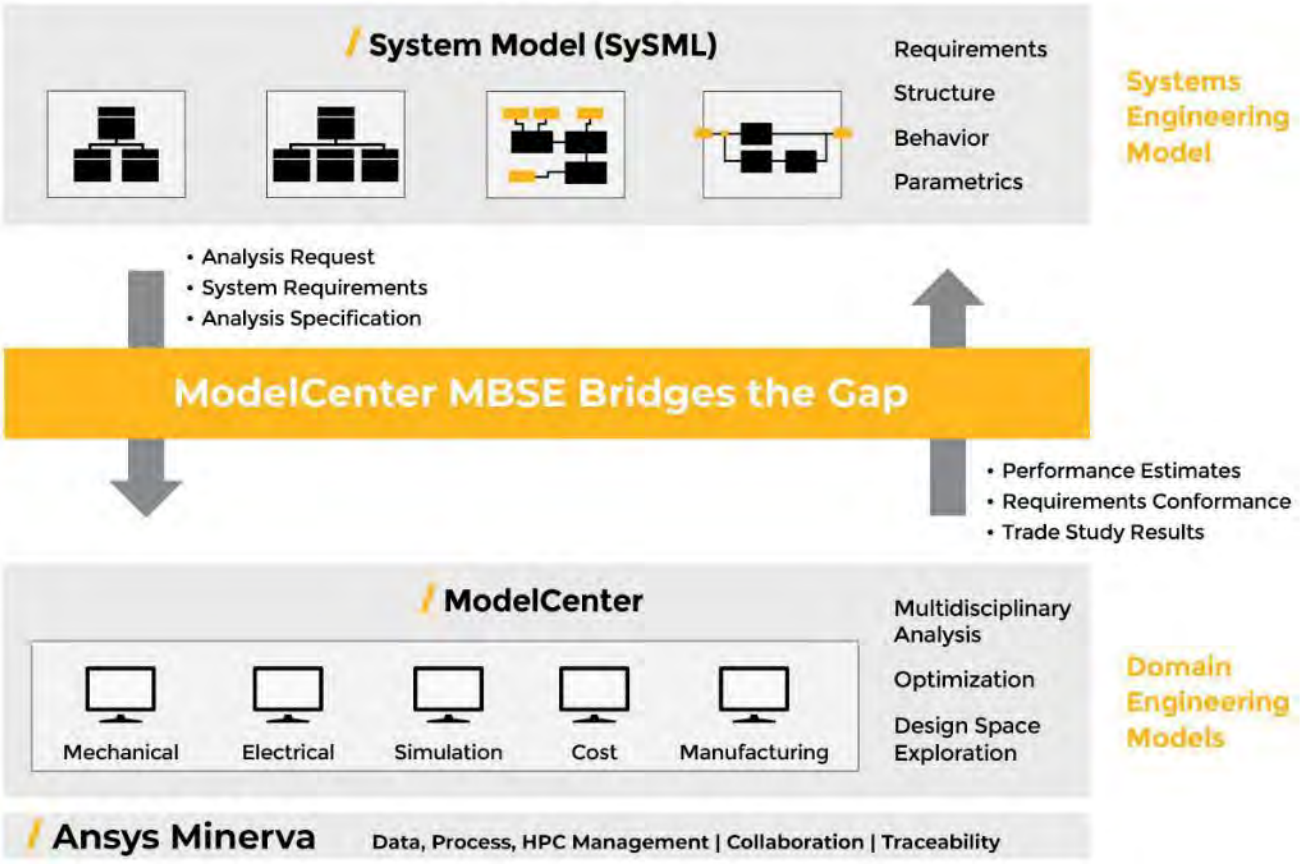
Without that integration, you can't verify requirements and fully evaluate trade-offs among performance, cost and risk. How do you know the larger ramifications of making each design change? Manual code execution and the use of ad-hoc scripts is slow and error prone. Using manual processes is asking for trouble, especially for more complex products where one bad decision could be the difference between success and failure.

Open Models Shut Down Inefficiency

To help address such challenges, Ansys has acquired Phoenix Integration, Inc., the developer of ModelCenter, the premier software for enabling model-based engineering (MBE) and model-based systems engineering (MBSE).

MBE is a holistic approach to product development that relies on digital models to drive all engineering activities – it begins with the definition of system requirements and continues throughout the product life cycle. It enables engineers to understand interactions and synergies between engineering subsystems and how design decisions for one part or assembly affect the behavior of the complete system.

MBSE uses a systems architecture model to manage complexity and to act as the single source of truth for describing an ever-evolving system design. ModelCenter allows users to link the systems model with multi-disciplinary analysis workflows to verify requirements and optimize the system design.



System MOdel (SySML) - ModelCenter MBSE Bridges the Gap System engineers and domain engineers don't always speak the same language. ModelCenter and Ansys Minerva bridge the gap.

The ModelCenter software suite is a vendor-neutral software platform for creating and automating multi-tool workflows, optimizing product designs and enabling MBSE. ModelCenter combined with Ansys Minerva makes MBE possible by giving engineers the ability to:

- Create and maintain a library of analysis models and engineering workflows
- Connect these workflows and analysis models with requirements and descriptive system models
- Automatically execute the workflows across different computers and operating systems
- Perform multi-run trade studies and ask "what-if" questions
- Visualize the design space and find the best designs
- Archive, manage and share the resulting data and metadata.
- Ensure traceability, interoperability and enterprise deployment

With the ModelCenter software suite and Ansys Minerva, engineering teams can capture, organize and share processes, models, input data, and the results and conclusions generated during the engineering design and analysis process. That information not only saves time and direct costs, it also allows engineers to gain valuable insights at a system level that can help organizations avoid liability exposure, product failures and recalls.

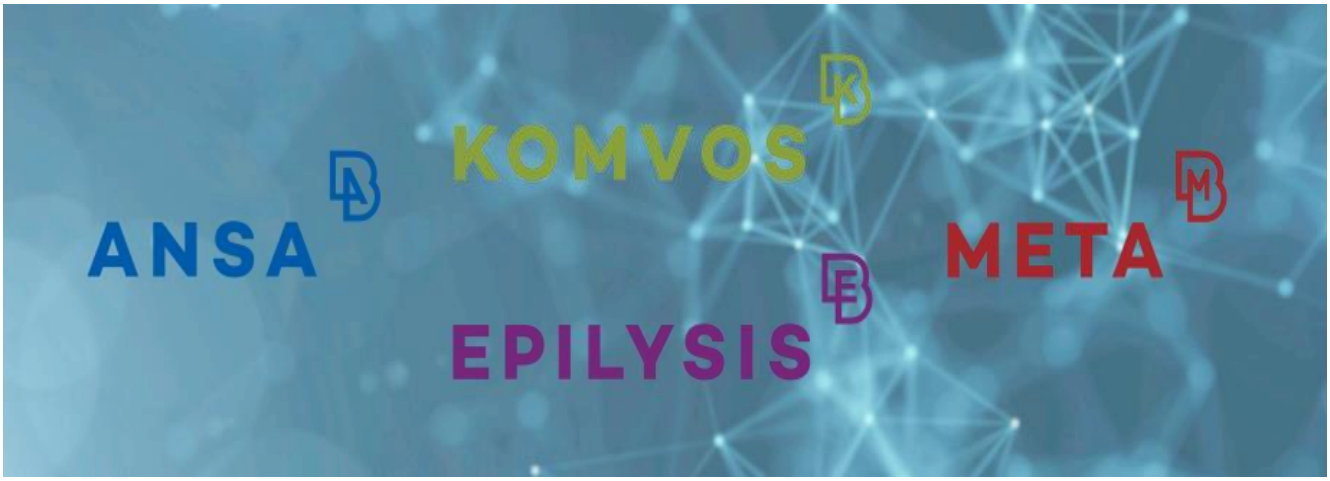
Tie Tools Together With MBSE

Engineering teams use many different tools during the product design and development process, spanning different domains and disciplines. Not all of those tools were designed with an open platform methodology. ModelCenter connects those tools from different vendors into a workflow that provides visibility across the process, enabling organizations to reduce development costs, improve engineering efficiency, stimulate innovation and design more competitive products.

"Companies across industries are benefitting from process integration and MBSE through improved communications, increased product quality and heightened productivity across their engineering teams," said Shane Emswiler, senior vice president of Ansys, via a press release. "Acquiring market leader Phoenix Integration will complement our acquisition of Dynardo in the process integration and design optimization space, and expand our capabilities in providing customers with strong MBE and MBSE offerings to further our pervasive engineering strategy. I am excited for Phoenix Integration to join the Ansys team."

Read in Ansys website

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



May 28, 2021

BETA CAE Systems announces the release of the v21.1.3 of its software suite

About this release

Thank you for contributing to bringing our software to its next level of maturity. You deserve the latest of the best. Today, BETA CAE Systems announces the availability of v21.1.3 of its software suite, the new second point release of ANSA/EPILYSIS/META and KOMVOS, incorporating numerous fixes in recently detected issues. Increase the reliability of your simulation work with this latest software release.

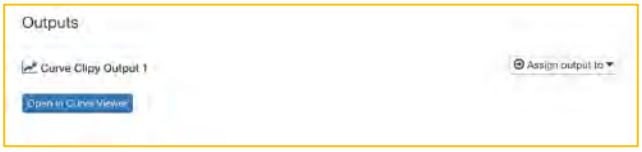
Don't miss some of the most notable improvements:

- License Management for ANSA and META
- Known issues resolved in ANSA
- Known issues resolved in EPILYSIS
- Known issues resolved in META
- Known issues resolved in KOMVOS
- New Documentation in ANSA
- Compatibility and Supported Platforms
- Download

More Detail from website

New Workflows Features

d3VIEW is consistently enhancing our platform for efficiency, better usability and more intelligent processes. Highlighted below are a few important additions to the Workflows application

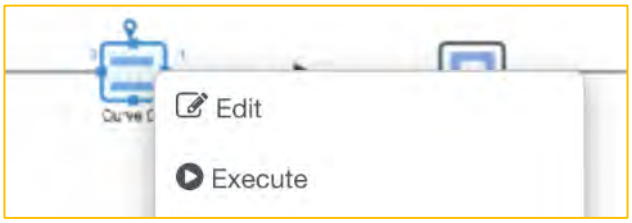


Assign Worker Outputs to Worker Inputs

Choose to assign executed worker outputs to other worker inputs on the canvas via the output worker.

Search for Workers on the Canvas in Execution Mode

During execution, use the new search bar at the top to find specific workers.

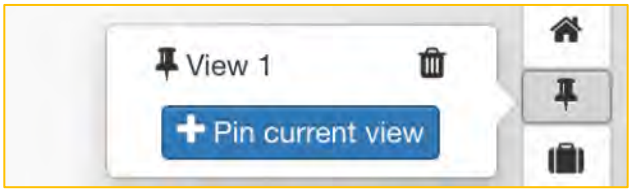


Execute Individual Workers

This new right-click option for individual workers support executing a single worker in a workflow.

New Decision HUBs

Decision HUBs are basically multiple decision workers in one, implementing conditions for multiple workers.



Pin Views in Workflows

Canvas views can now to pinned and saved for later.

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 Onsite

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

ARUP DYNA MORE DYNA MORE FRANCE DYNA MORE NORDIC DynaS+

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.

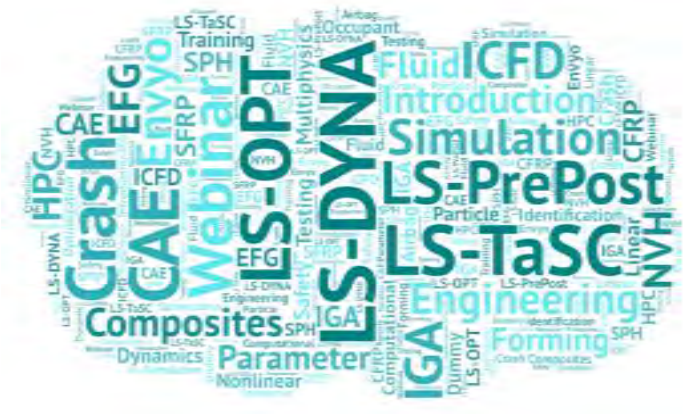
More information and contact

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 July and September

Webinars	
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
Simulation of thermoplastics	13-14 September
User Interfaces in LS-DYNA	17 September
Introduction to Isogeometric Analysis with LS-DYNA	23-24 September
Introduction to LS-PrePost	27-28 September
Introduction to LS-DYNA	29 September - 1 October

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.

Get Sustainable Practices Right, Virtually

How heavy industry leaders are committing to digitalization to achieve sustainability

ESI LIVE is a virtual event where progress in digital transformation becomes concrete. The Heavy Machinery edition on May 26, 2021, was one of such occasions where I had the pleasure of hearing from Caterpillar, Cummins, KONE, Volvo CE and more on where they are in their digital transformation journey. The most important lessons I learned in a nutshell: the rich diversity of sustainability goals and the strong commitment to key-enabling, advanced digital technologies in the heavy machinery industry.

1. Sustainable business models | Offering premium quality and excellent after-market solutions for their equipment is a key differentiator and revenue generator, particularly for OEMs of large investment goods who sell maintenance-free, productive, and safe work hours. For OEMs to be able to bring this value to their customer, they must evolve their maintenance activities towards condition-based, 24/7 connected services. How can they achieve this “zero downtime” promise for operators? By supplementing traditional condition monitoring of product assets with physics-based system simulation, machine learning, and artificial intelligence to build a virtual representation of their asset. Gian Luigi Di Lodovico, Head of Engineering Solutions at KONE, sees digital twins and advanced hybrid twins as being the backbone of any modern, sustainable asset management strategy – one that enables not predictive maintenance but the improved prescriptive maintenance, based on the current plant condition.

2. Environmental sustainability | With, often, more than half of OEMs global teams out in the fields for maintenance and service operations, the optimization of these activities – in terms of reducing labor, travel time and cost – is mission-critical in achieving ambitious greenhouse emission and carbon neutrality targets. The capabilities offered through advanced digital technologies are crucial because they help companies like KONE, for instance, minimize maintenance travel, which is directly linked to achieving their greenhouse emission goals and reducing the overall carbon footprint while also improving people flow and customer experience. Gian pointed out that already with a basic digital twin solution for predictive maintenance, KONE experiences up to a 30% decrease in call-outs from customers and up to 60% fewer issues because the OEM can detect potential issues in advance. What an achievement and potential if you consider the fact that the digital journey has just begun and KONE is working on advancing the digital twin into a hybrid twin! With this next step, they will be ready to evolve into prescriptive maintenance for their equipment and expect to further improve the numbers above. One aspect to keep in mind here: Condition-monitoring often requires the use of fast-running models to represent the physics of the targeted component or system. Nate Wieland, Sr Technical Team Leader, Virtual Product Development Division at Caterpillar, sees a significant amount of innovation potential arising from using reduced-order models (ROM). ROM is a simplification of a high-fidelity simulation model that preserves essential behavior and dominant effects for the purpose of reducing solution time, compute cost or storage capacity required for more complex models.

3. Sustainable development | The second group of major beneficiaries from combining engineering expertise and advanced virtual capabilities are R&D engineers who start exploring equipment performance fully virtually - with minimum physical prototype and test. Kieran Richards, Global Design Leader at Cummins Power Systems explains that they see Virtual Reality as an integral part of how they design and develop cleaner products and also as a tool of developing greener, more efficient facilities. By doing more virtually, Cummins can avoid issues, reduce changes, rework, waste and scrap, minimize test time and complexity whilst also decreasing the carbon-based emissions of their products and services. This is crucial for achieving the near-term goals of their “Planet 2050” agenda. With regards to agility and flexibility in times of insecurity, Kieran shared another very valid learning: since the pandemic, they had to increase the utilization of digital capabilities outside the centrally available office environment as many of their engineering teams have been working at home. The remote use of Virtual Reality allowed CUMMINS engineers to work more effectively in this distributed virtual world.

4. Sustainable, agile operations | Ultimately, Dr. Harri Kulmala, CEO of the Finish high-tech ecosystem, DIMECC, pointed out that the success of any digital and twin strategy stands and falls with the ability to quickly adapt to environmental changes. While it is crucial to maintain focus on your long-term vision, it’s even more important not to become too rigid in your long-term plans. Instead, remain agile to changes in your environment and be prepared for the next ‘Black Swan Phenomena’ (e.g. Coronavirus) - you cannot predict when, but you know it will certainly happen again. Gian from KONE’s recommendation for being successful with digital twin strategies is: “split investment into pieces, make the development and deployment in incremental steps, and get revenue incrementally.”

5. Diversity for sustainability | Truthfully, this is an old hat yet more valid than ever: we are together on this new digital road ahead of us, bringing in unique skills and expertise, sharing the same vision for the sake of our earth and people. Our ESI LIVE speakers are aligned on the fact that we can only exploit a significant amount of innovation potential and realize systemic changes if we collaborate across the industry, vendors and academia, share resources, competencies, and funding versus building everything on your own.

6. Sustainability needs collective engagement | In the hearts and minds of all our stakeholders, ESI stands for "Get it right". This is the essence of our brand and the guiding north star for all our activities including our corporate social responsibility. The commitment we share with our ESI Live contributors is to develop solutions that help our customers reduce their environmental footprint while progressing toward our Group's carbon neutrality as well as engaging employees and other stakeholders in the creation of a more sustainable world. One very concrete action to preserve our planet is, for instance, our commitment to the reforestation of the planet: We at ESI want to plant 10,000 trees by 2025. On behalf of all my ESI colleagues, I want to thank our 5 ESI LIVE speakers for contributing to this effort via the Reforest'Action program, by planting 5 new trees in Southern Portugal.

These were the lessons I learned from the ESI LIVE Heavy Machinery 2021 digital event (now available ON DEMAND). What about you? Do you agree with our speakers or have you had a different experience?

Also, don't forget to mark your calendar for the next ESI Live 2021 on November 4. You'll gain even more concrete, first-hand insights into the digital transformation journeys and 'real virtual' sustainable practices in automotive, aerospace, energy and heavy machinery industries. I look forward to seeing you there!

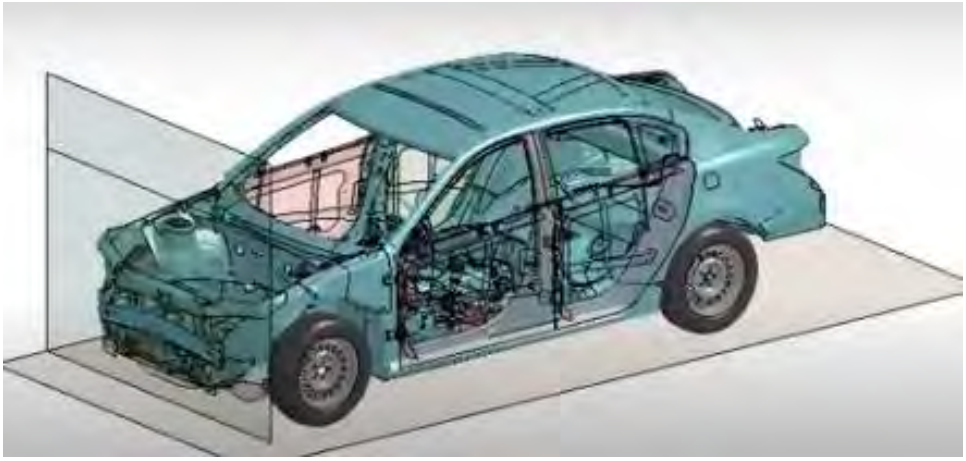
ETA has impacted the design and development of numerous products - autos, trains, aircraft, household appliances, and consumer electronics. By enabling engineers to simulate the behavior of these products during manufacture or during their use, ETA has been involved in making these products safer, more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

VPGSuite

Evaluate Product Designs in a Virtual Environment

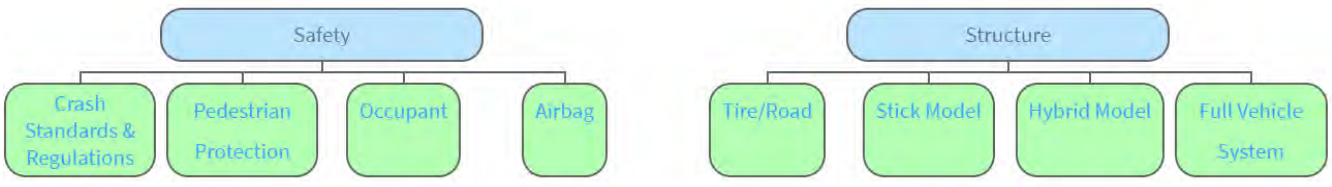
VPG stands for Virtual Proving Ground, which is a vertical application that can function as a standalone in the all-new VPGSuite or can be used as a plug-in with other preprocessing tools to setup proving ground simulations. VPGSuite allows users to quickly replicate real life, proving ground simulations in a virtual environment. This can involve crash and safety-related (standards/regulatory, occupant safety – positioning, seat belt, and pedestrian protection) or structure reliability-related (suspension, chassis, steering, vehicle dynamics, and durability). VPGSuite assists engineers with understanding true system performance under various testing conditions, providing early system-level design validation.

LS-DYNA is the default solver behind VPGSuite. The collaboration allows VPGSuite to become a robust tool that simulates various testing scenarios and drives the design based on the test results. Additional developments are underway to allow users to simulate proving ground simulations using other commercial codes such as Abaqus.



Benefits include:

- VPGSuite provides a simple way to edit and understand LS-DYNA with the ability to check the input file.
- Supplies users with the flexibility to use the 'unique' LS-DYNA model for a crash, safety, ride/handling, and road load test.
- Incorporates the multi-physics capabilities of LS-DYNA to simulate 'real world' physics issues related to wind turbines, solar panel structures, and aircraft matters.



Tel: +1 248-729-3010 Email: etainfo@eta.com

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

Read our latest Magazine and Welcome to our Town Hall Meeting - pdf format

Time to get your cup of coffee and choose what class to learn.

Ansys Innovation Courses: technical, interesting, free, online physics & engineering courses.

Among my favorites of the Structures Engineering Courses offered are: Structures Courses

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.

Shanghai Hengstar Technology Co., Ltd

Shanghai Enhu Technology Co., Ltd

Online workshop on "Toward adaptive vehicle safety system: development and application of parametric human FE models"

Shanghai Hengstar Technology will organize a Webinar with topic "Toward adaptive vehicle safety system: development and application of parametric human FE models" on July 27 2021.

Contents:

1. Occupant Protection: Restraint design optimization, Adaptive restraint design, Active and passive safety integration, Tactical vehicle safety;
2. Impact/Injury Biomechanics: Parametric Human Modeling, Statistical Injury Data Analysis, The injury mechanism and safety designs for various vulnerable populations (e.g. elderly, obese occupants, children, pregnant female, wheelchair users, etc.)

Instructor:

Jingwen Hu (Research associate professor, University of Michigan**)**

Dr. Jingwen Hu is a researcher at the Biosciences Group of the University of Michigan Transportation Research Institute (UMTRI). He also hold a joint appointment at the Department of Mechanical Engineering at University of Michigan, Ann Arbor, MI. His research interests primarily focus on impact/injury biomechanics in motor-vehicle crashes by a multidisciplinary approach using combination of experimental, computational, and epidemiological procedures. One of the highlights of his recent research is the development of parametric computational human models representing a diverse population. Such models have been used to study the injury mechanism and safety designs for various vulnerable populations, such as children, elderly, pedestrians, pregnant women, and obese occupants.

Duration and Date: (1 hours Webinar)
July 27 (9:00AM-10:00AM)
Language: Mandarin
Contact: Xixi Fei
Tel. :021-61630122 Mobile:13524954631 Email:Training@hengstar.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.

JSTAMP Functions

Address various tasks in tool shop

JSTAMP represents the Sheet metal forming process virtually by numerical simulation. Users can examine the simulation result, output it to CAD, and directly use the CAD as a countermeasure by using JSTAMP.

Designers can avoid the challenges of trial and error. JSTAMP provides an adequate result and reduces the lead time and cost of tool design.

JSTAMP provides comprehensive support throughout the design process from the first trial to the final stage. The feature for addressing complicated process stages, low formability materials, and latest technologies covers various tasks in the Sheet metal forming process.

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.

Buckling analysis of Water tower

Objective:

To perform an eigenvalue buckling analysis for a water tower to predict the buckling load of the structure using ANSYS Mechanical.

Pre-processing:

The CAD model was subjected to meshing such as the columns and cross members were modelled with beam elements.

The cross sections of the beams were extracted using ANSYS Spaceclaim tool.

All the cross members and columns were assignments Elasto-plastic Structural steel material.

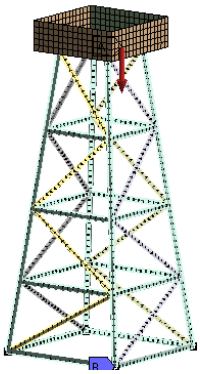
Loading conditions:

Considering the capacity of the water tank, the load was approximated as a point mass with gravity.

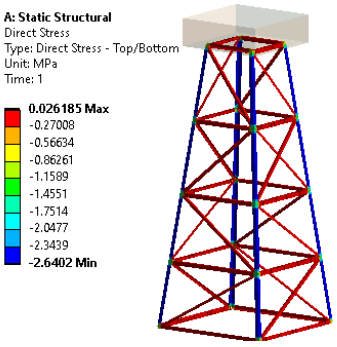
Mass of tank = 1000 kg

Boundary constraints:

All the bottom ground nodes were fixed in all directions.

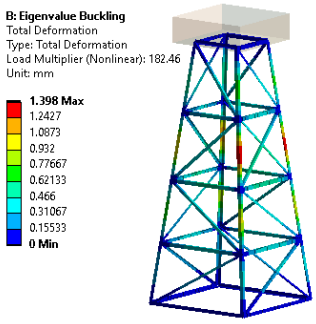


Beams stress results:



Eigenvalue buckling deformation:

Static structural is linked with Eigenvalue buckling system.



Buckling load factor of structure = 182.46

It was found that the buckling load of water tower = 9810 * 182.46 = **$1.789 * 10^6$ N**

Ending remarks:

- The results calculated by the Eigenvalue Buckling analysis are buckling load factors that scale all of the loads applied in the Static Structural analysis.
- Hence, it's typical to apply unit loads in the static analysis that precedes the buckling analysis.
- You can apply a nonzero constraint in the static analysis. The load factors calculated in the buckling analysis should also be applied to these nonzero constraint values.
- However, the buckling mode shape associated with this load will show the constraint to have zero value.

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

On Structured ALE Mesh Trimming

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 the mesh trimming feature.

Mesh regularity.

S-ALE solver expects a regular, box-shaped rectilinear mesh. This regularity enables an automated mesh generation by the solver, instead of pre-processor. And more importantly, this mesh regularity enables a simpler algorithm leads to a reduced simulation time and memory usage. It comes with a price though.

In the generic ALE solver, ALE mesh is generated by the user and could be of arbitrary shape. Users have the flexibility to mesh the ALE domain to better fit their needs. For example, to model a spherical blast one could generate a spherical ALE mesh domain. Another example is to model fuel sloshing in a rigid body container. One could choose to make the ALE outer surface conform to the Lagrange container. And then constrain the normal directional flow by using *ALE_ESSENTIAL_BOUNDARY.

The above two examples are without any fluid structure interactions. Now let us go to cases with FSI as these are what S-ALE targets to solve. Still on fuel sloshing problem, what we care is to see how the fuel tank deforms under a sudden fluid impact so container has to be flexible and FSI is needed. The shape of the container, for example, is somewhat very irregular. Let us say its height on the left side is much less than the right side, only ¼. To reduce the number of ALE elements, one might want to mesh the left side with less ALE elements. Just to make sure it covers the Lagrange container with several

extra cushion layers of elements. As ALE runs are comparatively much more expensive, stinginess becomes a virtue.

From the above example, one could understand the motivation behind the development of mesh trimming feature. Its major objective is to keep the element cost low, by trimming the mesh to better fit the domain of interest. In most cases, to better fit the geometry of Lagrange structures coupled to ALE fluids. Simply put, mesh trimming is to remove the elements far away from the structure or some geometric entities.

A first glance.

Let us see how it is done. Below is the format and description of the keyword *ALE_STRUCTURED_MESH_MESH_TRIM. There could be multiple trimming cards which are executed in the order of their appearances.

*ALE_STRUCTURED_MESH_MESH_TRIM							
MSHID	OPTION	OPER	OUT/IN	E1	E2	E3	E4

MSHID is the Mesh ID of the S-ALE mesh, defined in *ALE_STRUCTURED_MESH card. OPTION is to provide geometries to trim the S-ALE mesh and could be chosen from the following six: PARTSET, SEGSET, PLANE, CYLINDER, BOXCOR, BOXCPT and SPHERE. OPER has two choices: 0 to trim or 1 to keep. IN/OUT is to prescribe inside or outside; 0 – outside, 1 – inside. E1,E2,E3,E4 are to provide the geometric data and have different meanings for each different OPTION.

A picture is better than thousand words. The figure below shown a S-ALE mesh trimmed by using OPTION=”SEGSET”, OPER=”0” (trim) and OUT/IN=”0” (outside) with an offset of “10”. The segment set is the tube-like surface of red color. S-ALE mesh is on top and around the red tube, transparent and of a color of light blue.

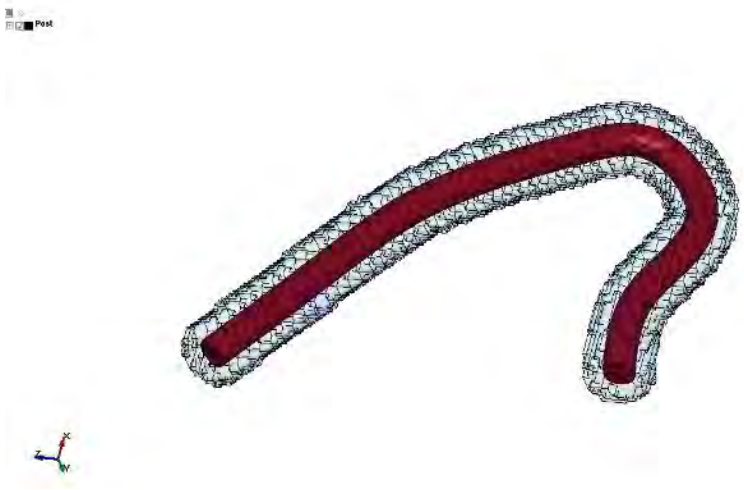


Figure 1. S-ALE mesh trimmed around “glue” (the red tube)

This model is done by Mukul Atri, Ansys ACE India, mukul.atri@ansys.com. The input deck could be found at https://ftp.lstc.com/anonymous/outgoing/hao/sale/models_R121/trim/tube_trim.k

Let us go through its setup to illustrate the usage of mesh trimming card. The initial mesh is a box mesh of 75x63x68 elements spans from (-150,-125,-135) to (150,125,135). There are two ALE multi-materials in this mesh – "glue" and "vacuum". The segment set #2 is created from the outer surface of a Lagrange dummy part; And used later to do the volume filling for ALE fluid "glue". We want to study how "glue" flows under compression (not included in this model for simplicity).

```
$=========================================================================
*ALE_STRUCTURED_MESH
$   meshid      dpid      nbid      ebid
         1         2
$    cpidx     cpidy     cpidz      nid0     lcsid
         1         2         3
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$     cipd
         1
$        n                 x
         1              -150
        76               150
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$     cipd
         2
$        n                 x
         1              -125
        64               125
*ALE_STRUCTURED_MESH_CONTROL_POINTS
$     cipd
         3
$        n                 x
         1              -135
        69               135
$=========================================================================
*ALE_STRUCTURED_MULTI-MATERIAL_GROUP
$# mmgname       mid     eosid                                          pref
  glue          1001      1001
  vacuum        1002
*MAT_ALE_VISCOUS
      1001  1.000E-5   -1.0E10    2.0E-6       8.0   2.16E-2              0.201
*EOS_LINEAR_POLYNOMIAL
      1001       0.0    1000.0       0.0       0.0       0.0       0.0       0.0
       0.0       0.0
*MAT_VACUUM
      1002   1.0E-08
$=========================================================================
```

After setting up the above cards, we have a box mesh of 321,300 S-ALE elements, as shown below. There is quite some waste. All space not taken by "glue" (which occupies the same space as the red Lagrange tube) is going to be filled with "vacuum". Most of the space, "glue" will never flow into it. There is no point to keep all those elements far away from "glue". So using a box mesh mostly filled with vacuum is really something we strongly dislike.

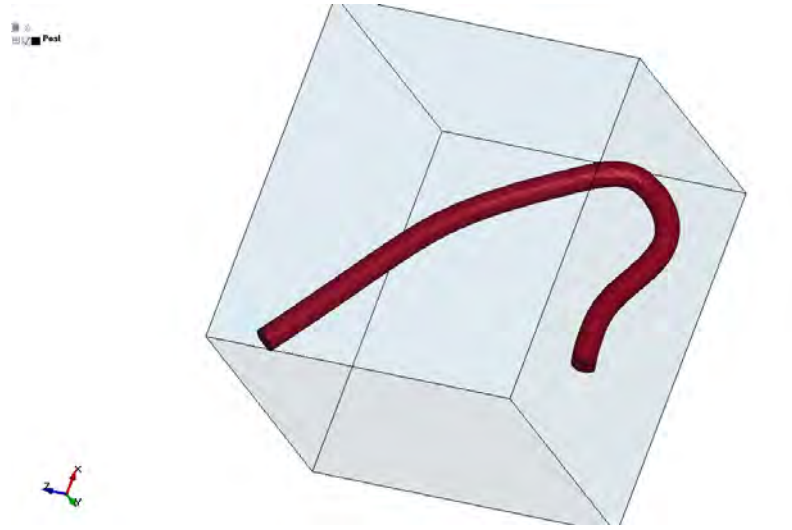


Figure 2. S-ALE box mesh untrimmed

To eliminate this waste, it is easy and straightforward. Simply adding the card below.

```
*ALE_STRUCTURED_MESH_TRIM
$     mshid     option       oper     out/in      segid     offset
          1     SEGSET          0          0          2       10.0
```

What it does is to remove S-ALE elements far away (10.0) from the outer surface of the red tube. The number of S-ALE elements is reduced to 14,173 from 321,300. Impressive, right? We would not expect the cost saving is always this significant, but still for special cases it could be quite surprisingly good.

*ALE_STRUCTURED_MESH_TRIM

Let us describe the design of *ALE_STRUCTURED_TRIM to get a better understanding of its usage.

*ALE_STRUCTURED_MESH_MESH_TRIM							
MSHID	OPTION	OPER	OUT/IN	E1	E2	E3	E4

1. Multiple entries allowed. Processed in the order of their appearances.
2. Five basic geometries: PLANE, CYLINDER, BOXCOR, BOXCPT and SPHERE.
3. Complicated geometries provided using PARTSET or SEGSET.
4. Operation be either trim (0) or keep (1) (or call it untrim, bring the trimmed elements back from previous trims)

5. OUT/IN: To operate on the elements outside (0) or inside (1) of the geometry For PARTSET and SEGSET options, “outside” is defined as the region to which the segment normal points.

Most of the above points could be easily understand by our users. Maybe with one exception: One might wonder what the purpose of operation “keep” is. Let us illustrate its usage by using an airbag model. The full input deck could be found at the following directory. https://ftp.lstc.com/anonymous/outgoing/hao/sale/models/meshmotion/airbag1/

The S-ALE mesh is shown in the figure below. The mesh is composed of two boxes. The upper one has 7 layers of elements along z direction from 0.0315 to 0.2415; each layer contains 16x16 elements and spans from (-0.36, -0.36) to (0.36, 0.36). The bottom one has 3 layers from -0.0885 to 0.0315; each layer contains 6x6 elements and spans from (-0.135, -0.135) to (0.135, 0.135).

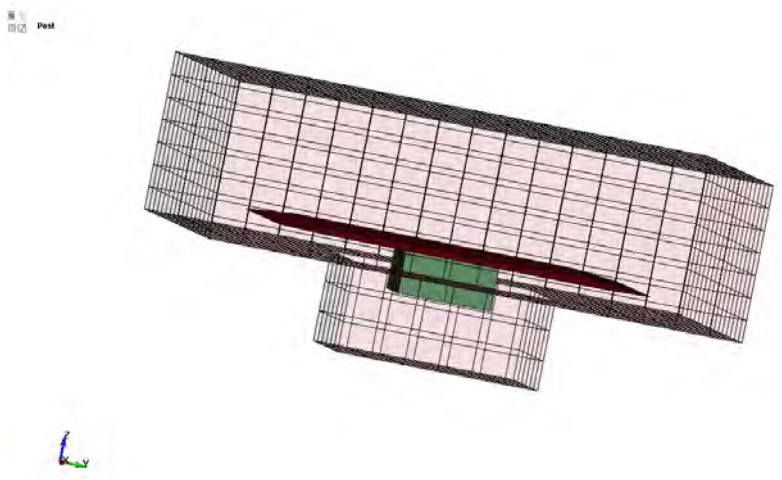


Figure 3. S-ALE mesh trimmed to have less elements at the bottom

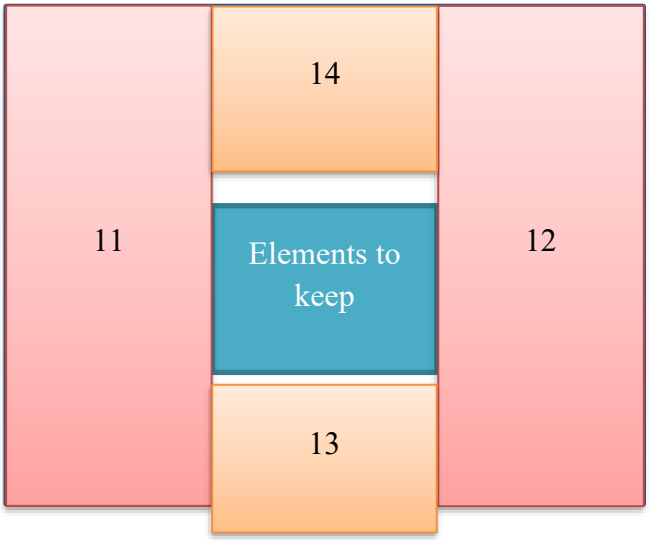
As always, let us define the ALE mesh first.

```
*ALE_STRUCTURED_MESH
$    mshid       pid       nbid      ebid
         1        11     100001    100001
$     nptx      npty       nptz      nid0     lcsid
      1001      1001       1003       272      1001
*DEFINE_COORDINATE_NODES
      1001       272        254       233         1
*ALE_STRUCTURED_MESH_CONTROL_POINTS
      1001
$                 x1                   x2
                   1                -0.36
                  17                 0.36
*ALE_STRUCTURED_MESH_CONTROL_POINTS
      1003
$                 x1                   x2
                   1              -0.0885
                   4               0.0315
                  11               0.2415
```

Now we need to trim the mesh at the bottom away from the center. The most suitable choice is to use BOXCPT option to define the geometry. For now, let us forget about the "keep" operation and do the trim with "trim" operation only. The bottom mesh starts from node [6, 6] to node [12, 12] in the xy plane. So this means we need to trim 4 times, each time with a box of elements. One way to do it is as follows.

```
*ALE_STRUCTURED_MESH_TRIM
$    mshid   command      oper    out/in     boxid
         1    BOXCPT                            11
         1    BOXCPT                            12
         1    BOXCPT                            13
         1    BOXCPT                            14
*DEFINE_BOX
$    boxid      xmin      xmax      ymin      ymax      zmin      zmax
        11         1         6         1        17         1         4
        12        12        17         1        17         1         4
        13         6        12         1         6         1         4
        14         6        12        12        17         1         4
```

These four boxes are shown in the figure below.



The result fits our expectations and the process seems OK. But somehow you have a gut feeling that there must be a better way to do it, right?

*ALE_STRUCTURED_MESH_TRIM is designed to do addition or subtraction between geometries. That is where "keep" operation comes into play. The above card could be simplified to:

```
*ALE_STRUCTURED_MESH_TRIM
$    mshid   command      oper    out/in     boxid
         1    BOXCPT                             1
         1    BOXCPT         1         1         2
*DEFINE_BOX
$    boxid      xmin      xmax      ymin      ymax      zmin      zmax
         1         1        17         1        17         4        11
         2         6        12         6        12         1         4
```

It contains two operations. The first is to trim any element outside (OUT/IN=0) of the box 1 at the top. After this, all elements at the bottom 3 layers are deleted. That is not what we wanted – we still need the center 6x6 elements at the bottom. So let us reverse the trim operation, but only at those 6x6 elements. That is what the second card does here. It is to "keep" (untrim) the elements inside (OUT/IN=1) the box 2 which spans from node [6, 6] to [12,12] in the bottom 3 layers.

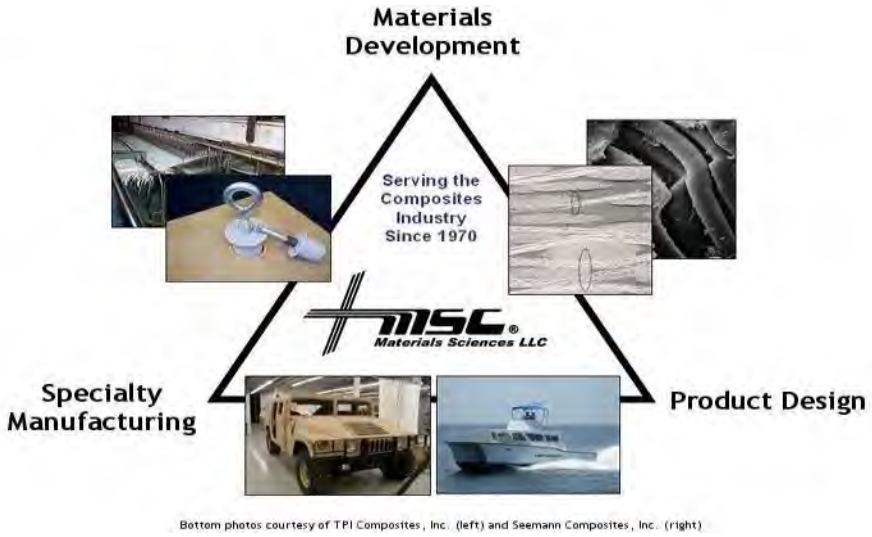
One can easily imagine the added flexibility it achieved by adding this "keep/untrim" operation. Most cases a complicated geometry could be described using addition/subtraction between 2 or at most 3 simple geometries. For a more detailed description of other options/flags in this keyword, please refer 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.

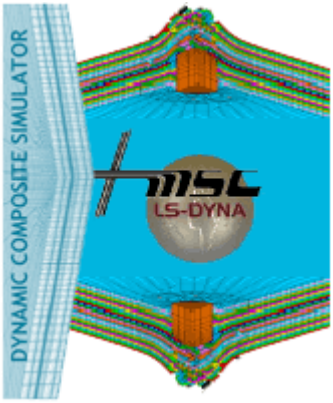
To prompt LS-DYNA ALE usages, Structured ALE solver introduced a new, user-friendly, streamlined three-step setup. We hope this effort could help users, new or old, to perform their work more efficiently and smoothly.

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.



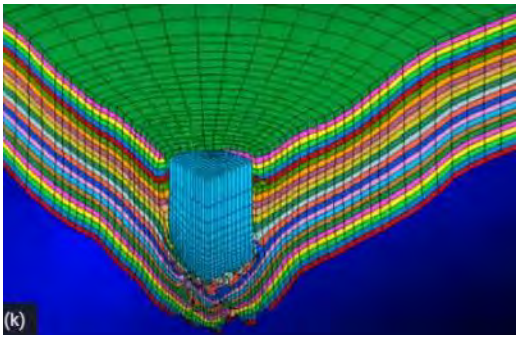
Bottom photos courtesy of TPI Composites, Inc. (left) and Seemann Composites, Inc. (right)

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 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.

D3PLOT VIEWER

New feature:

D3PLOT viewer

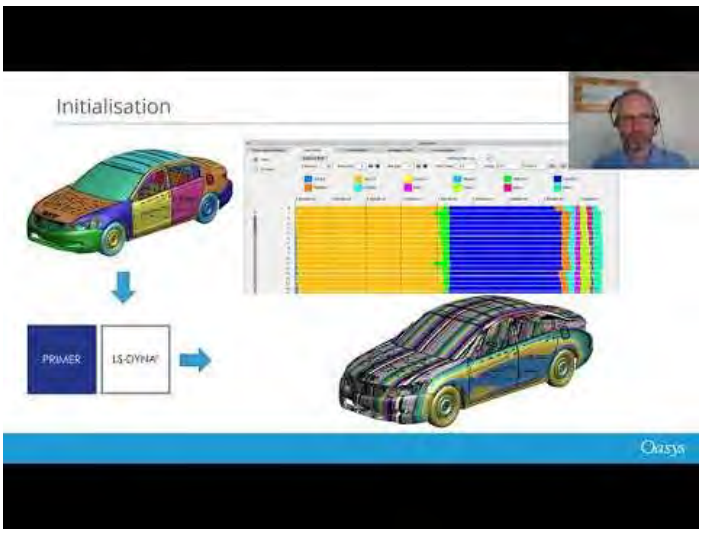
D3PLOT Viewer has the power to transform the way that you review, communicate, and deliver engineering analysis:

Explore in 3D with your team in meetings and design reviews

Share animated 3D models with designers and suppliers

Give your clients 3D project deliverables

D3PLOT Viewer is free to use - for your team, your partners and your clients. No installation, registration or set-up is required. Find out more on our website here.



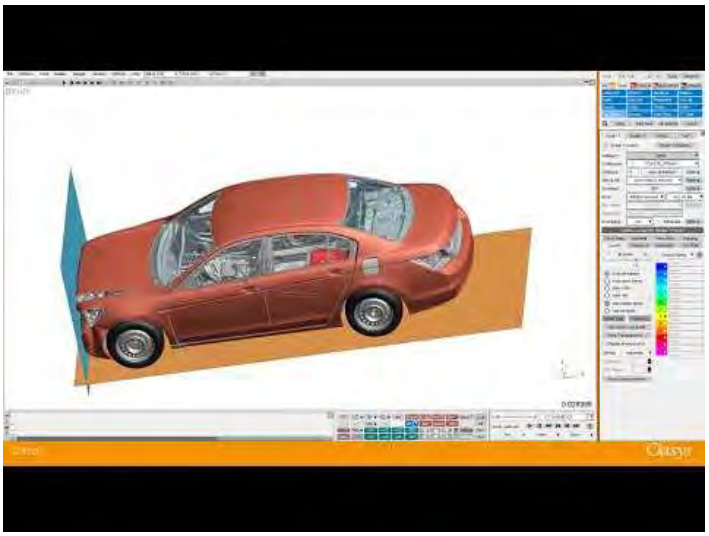
Webinar to watch again:

Oasys Suite 18.0 Keynote

Following the release of Oasys Suite 18.0, you can now re-watch the Oasys keynote presentation from our recent Virtual Update Meeting:

"New features to accelerate your workflow with the Oasys LS-DYNA Environment 18.0".

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



Top Tip video: Did you know?

Did you know that you can define cut sections within Oasys D3PLOT which can be useful when looking at models with lots of parts or complex deformations?

← Click on the image to watch.

Upcoming Oasys LS-DYNA training courses

Featured Introduction to Oasys PRIMER (online)	7 Jul, 2021	3 days 09:30 - 12:30 BST
Featured Introduction to Oasys POST (online)	21 Jul, 2021	3 days 09:30 - 12:30 BST

The Oasys LS-DYNA training courses are 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

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

FINITE ELEMENT ANALYSIS
Predictive Engineering

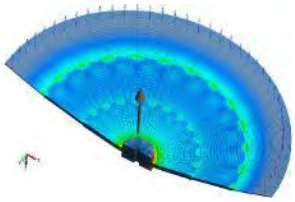
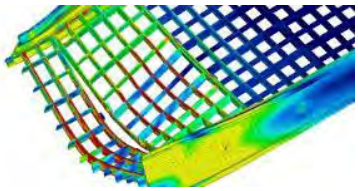
LS-DYNA has been one of Predictive's core analysis tools pretty much since we got started in 1995. It is an amazing numerical workhorse from the basic linear mechanics (think ANSYS or Nastran) to simulating well nigh the impossible. At least that is the way I feel at times when the model is not solving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

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.

Description: ANSYS LS-DYNA is the most commonly used explicit simulation program, capable of simulating the response of materials to short periods of severe loading. Its many elements, contact formulations, material models and other controls can be used to simulate complex models with control over all the details of the problem.

Industries

Available Versions: 2021 R1 / 2020 R2 / 2020 R1 / 2019 R3 / 2019 R2 / 2019 R1 / 19.2 / 19.1 / 19.0 / 18.2 / 18.1 / 18.0

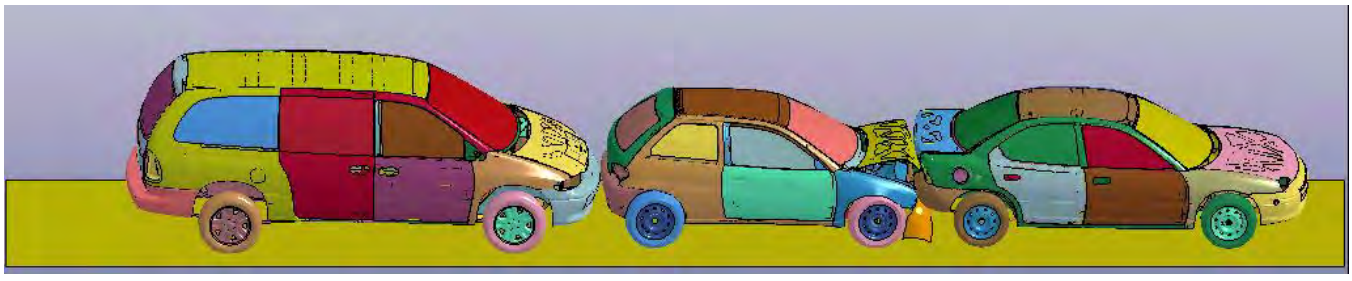
Licensing: Short Term, Bring Your Own

LS-DYNA Examples

LS-DYNA is 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. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration. It is used for analyzing high speed, short duration events where inertial forces are important. Typical uses include automotive crash, explosions, manufacturing etc.

Large LS Dyna models can be computationally intensive and Rescale provides a platform with easy work flow to run such models across multiple cores. Given below are some benchmark LS Dyna models on Rescale's platform, ScaleX.

Read more in Rescale Documentation Home



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

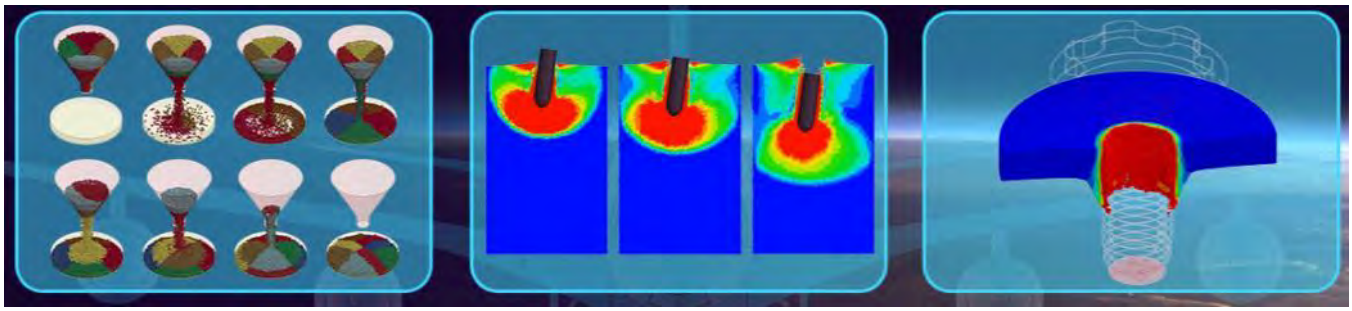
Shanghai Fangkun Software Technology Ltd

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

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

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

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



Contacts

Address: Room 2219, Building No.1, Global Creative Center, Lane 166, Minhong Road, Minhang District, Shanghai

Postcode： 201102 **Tel：** 4008533856 021-61261195

Sales Email： sales@lsdyna-china.com **Technical Support Email：** support@lsdyna-china.com

LS-DYNA Training Plan in 2021

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

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

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

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

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

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

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

FE analysis

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

CFD analysis

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

EM analysis

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

Metal sheet

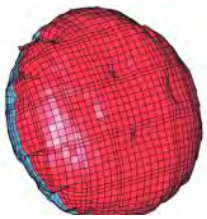
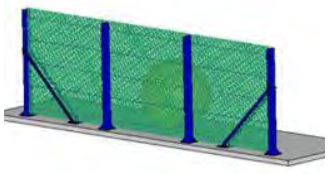
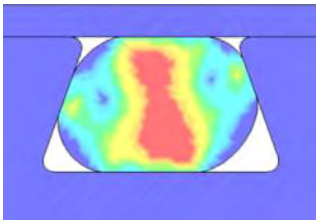
- JSTAMP is an integrated forming simulation system for virtual tool shop based on IT environment. JSTAMP is widely used in many companies, mainly automobile companies and suppliers, electronics, and steel/iron companies in Japan.

Pre/ Post

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

Biomechanics

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

Dodge Partners With Universal Pictures on 'F9,' the Ninth Installment of 'The Fast & Furious' Franchise, Opening Nationwide on June 25

Diane Morgan June 11, 2021

- Co-branded Dodge and "F9" promotional campaign includes television, social and digital media extensions, multicultural and in-theater placements
- Sixty-second "Superpower" spot launches across Dodge social and digital media channels today, June 11
- Campaign includes Atlantic Records collaboration, with Dodge vehicles playing a supporting role in the official music video for the new track "I Won," which is featured in the film
- The Dodge Charger SRT Hellcat Widebody is featured in the newest chapter of one of the most popular and enduring film franchises of all time

June 11, 2021 , Auburn Hills, Mich. - Dodge, which has been a part of Universal Pictures' groundbreaking, blockbuster "Fast & Furious" franchise since its inception, announced today its promotional partnership with Universal Pictures for "F9," which arrives in U.S. theaters June 25. In support of the film's release, Dodge is launching a multi-tier marketing campaign, "Superpower," which includes television, social and digital media extensions and in-theater placements.

The co-branded campaign promotes the blockbuster "Fast" franchise, which has earned more than $5 billion worldwide, and Dodge//SRT, the world's fastest and most powerful muscle cars. The Dodge Charger SRT Hellcat Widebody is featured in "F9," in addition to Dom Toretto's vintage 1960 classic Dodge Charger. The 60-second "Superpower" video launches today, June 11, across the Dodge brand's social media channels. A 30-second version of the "Superpower" spot will run across television.

"Dodge vehicles have played a pivotal role in the 'Fast' saga, helping to fuel this incredible franchise for nearly two decades," said Olivier Francois, Global Chief Marketing Officer, Stellantis. "Since the beginning, Universal Pictures has shown its commitment to our partnership by creating stories where some of the world's most iconic and significant performance vehicles become beloved characters and an important part of the 'Fast' family."

"'Fast & Furious' heroes don't wear capes, they drive muscle cars," said Tim Kuniskis, Dodge Brand Chief Executive Officer – Stellantis. "With Dodge, horsepower is our superpower, and Dodge and the Brotherhood of Muscle have become central characters in the enduring 'Fast & Furious' family, racing high-octane joy to generations of performance enthusiasts and cementing America's muscle car brand as a key component of this iconic film franchise."

Cars, movies and music come together again as part of the summer multimedia campaign, with the Dodge brand and Atlantic Records combining efforts in support of "F9." The official music video for the new track "I Won" (featuring Ty Dolla $ign, Jack Harlow & 24kGoldn), included in "F9," features Dodge vehicles, including the 2021 Dodge Charger SRT Hellcat Redeye Widebody and the 2021 Dodge Challenger SRT Super Stock.

Dodge vehicles featured in the 60-second "Superpower" video include:

- Dodge Charger SRT Hellcat Redeye Widebody (White Knuckle with Satin Black graphics)
- Dodge Charger Scat Pack Widebody (Indigo with Satin Black graphics)
- Dodge Challenger SRT Hellcat Redeye Widebody (TorRed)
- Dodge Challenger SRT Hellcat Widebody (Octane Red with Satin Black graphics)
- Dodge Challenger R/T Scat Pack Shaker Widebody (Smoke Show with Bumblebee graphics)
- Dodge Durango SRT Hellcat (DB Black with Redline Black dual stripes)
- Dodge Durango R/T Blacktop RWD (Reactor Blue)

The campaign was created in partnership with Universal Pictures and GSD&M.

F9

"F9" is the ninth chapter in the "Fast & Furious" saga, which has endured for two decades and has earned more than $5 billion around the world.

Vin Diesel's Dom Toretto is leading a quiet life off the grid with Letty and his son, little Brian, but they know that danger always lurks just over their peaceful horizon. This time, that threat will force Dom to confront the sins of his past if he's going to save those he loves most. His crew joins together to stop a world-shattering plot led by the most skilled assassin and high-performance driver they've ever encountered: a man who also happens to be Dom's forsaken brother, Jakob (John Cena, the upcoming "The Suicide Squad").

"F9" sees the return of Justin Lin as director, who helmed the third, fourth, fifth and sixth chapters of the series when it transformed into a global blockbuster. The action hurtles around the globe—from London to

Tokyo, from Central America to Edinburgh, and from a secret bunker in Azerbaijan to the teeming streets of Tbilisi. Along the way, old friends will be resurrected, old foes will return, history will be rewritten, and the true meaning of family will be tested like never before.

The film stars returning cast members Michelle Rodriguez, Tyrese Gibson, Chris "Ludacris" Bridges, Nathalie Emmanuel, Jordana Brewster and Sung Kang, with Oscar® winner Helen Mirren, with Kurt Russell and Oscar® winner Charlize Theron. "F9" also features Grammy-winning superstar Cardi B as new franchise character Leysa, a woman with a connection to Dom's past, and a cameo by Reggaeton sensation Ozuna.

"F9" is produced by Neal H. Moritz p.g.a., Vin Diesel p.g.a., Justin Lin p.g.a., Jeff Kirschenbaum p.g.a., Joe Roth, Clayton Townsend p.g.a., and Samantha Vincent. Universal Pictures presents an Original Film/One Race Films/Perfect Storm production in association with Roth/Kirschenbaum Films, a Justin Lin film. www.thefastsaga.com

Universal Pictures

Universal Pictures is a division of Universal Studios (www.universalstudios.com). Universal Studios is part of NBCUniversal. NBCUniversal is one of the world's leading media and entertainment companies in the development, production and marketing of entertainment, news and information to a global audience. NBCUniversal owns and operates a valuable portfolio of news and entertainment networks,

a premier motion picture company, significant television production operations, a leading television stations group and world-renowned theme parks. NBCUniversal is a subsidiary of Comcast Corporation.

Dodge//SRT

For more than 100 years, the Dodge brand has carried on the spirit of brothers John and Horace Dodge. Their influence continues today as Dodge shifts into high gear with muscle cars and SUVs that deliver unrivaled performance in each of the segments where they compete.

2021 marks the year that Dodge is distilled into a pure performance brand, offering Hellcat-powered, 700-plus-horsepower SRT versions of every model across the lineup. For the 2021 model year, Dodge delivers the drag-strip dominating 807-horsepower Dodge Challenger SRT Super Stock, the new 797-horsepower Dodge Charger SRT Redeye, the most powerful and fastest mass-produced sedan in the world, and the new 710-horsepower Dodge Durango SRT Hellcat, the most powerful SUV ever. Combined, these three muscle cars make Dodge the industry's most powerful brand, offering more horsepower than any other American brand across its entire lineup.

In 2020, Dodge was named the "#1 Brand in Initial Quality," making it the first domestic brand ever to rank No. 1 in the J.D. Power Initial Quality Study (IQS). The Dodge brand also ranked No. 1 in the J.D. Power APEAL Study (mass market). These results are historic because it marks the first time a domestic brand has earned top spots in both J.D. Power studies in the same year.

Dodge is part of the portfolio of brands offered by leading global automaker and mobility provider Stellantis. For more information regarding Stellantis (NYSE: STLA), please visit www.stellantis.com.

Read from website

LS-DYNA - Resource Links

LS-DYNA Multiphysics YouTube

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

FAQ LSTC

ftp.lstc.com/outgoing/support/FAQ

LS-DYNA Support Site

www.dynasupport.com

LS-OPT & LS-TaSC

www.lsoptsupport.com

LS-DYNA EXAMPLES

www.dynaexamples.com

LS-DYNA CONFERENCE PUBLICATIONS

www.dynalook.com

ATD –DUMMY MODELS

www.dummymodels.com

LSTC ATD MODELS

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

AEROSPACE WORKING GROUP

http://awg.lstc.com

Training - Webinars

Participant's Training Classes

Webinars

Info Days

Class Directory

Directory

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

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.

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).

A new fatigue solver based on modal dynamics in Ansys LS-DYNA®

Yun Huang, Tom Littlewood, Zhe Cui, Ushnish Basu, David Benson

Ansys, Inc.

Abstract

A series of time domain and frequency domain fatigue solvers have been implemented in LS-DYNA during the past few years to answer the requests from customers [1]. These fatigue solvers have important applications in the durability analysis of automotive, cranes, machines, engines and other products. To further accelerate the fatigue damage computation, a new method to run fatigue analysis based on modal dynamics has been implemented in LS-DYNA since release R13. This paper aims at introducing this new analysis method to LS-DYNA users. The necessary keywords for this method are described. Two simple examples are included to illustrate the effectiveness and efficiency of this new method.

Introduction

Fatigue failure is very common for industry structures and components. According to "Fatigue of Metals: part I", 90% of service failure of mechanical parts is due to fatigue damage. Thus, it is important to have a tool to compute the expected fatigue life of structures during the design phases. To answer such requests from customers, a series of fatigue solvers have been implemented in LS-DYNA in both the time and frequency domains [2-11]. In the time domain, one can run fatigue analysis using the stress / strain results saved in elout or d3plot database (***FATIGUE_ELOUT /*FATIGUE_D3PLOT**). In the frequency domain, fatigue analysis can be performed based on the structural response in the random vibration environment (***FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE**) and the sine sweep vibration environment (***FREQUENCY_DOMAIN_SSD_FATIGUE**).

For a large-scale model, fatigue analysis can be expensive since we need to get accurate stress history (of course, with high resolution). As a result, it not only takes a long CPU time to run the computation, but also requires a huge hard drive space to save the databases.

To speed up the fatigue analysis, a new option has been added to the fatigue solver (***FATIGUE_MODAL_DYNAMIC**) in LS-DYNA to run fatigue analysis based on the stress history data reconstructed from modal coordinates and modal stresses at each time step. To reach this goal, the model has to be solved using the modal dynamic method (***CONTROL_IMPLICIT_MODAL_DYNAMIC**). Please note that this method is more suitable for the case with linear dynamic response, where the stress response is almost linear and elastic, and the modal analysis of the original model is available.

Workflow and Keywords

As mentioned in Section 1, this method is based on modal dynamic analysis. As mentioned in [12], transient linear dynamic analysis can be performed very efficiently using a modal basis, since the number of dof can be reduced significantly. Based on the orthogonality of eigenvectors, the original system can be simplified to a smaller size equation system involving only the modal coordinates. The new equation system is decoupled for classical global damping conditions. After solving for the modal coordinates, the dynamic response of the original structure can be obtained using modal superposition method.

It is convenient to run the fatigue analysis based on modal dynamics in two steps:

- Run transient modal dynamic analysis to get "moddynout" and "d3eigv" with modal stresses;
- Run fatigue analysis with "**restrt**" =1, to read in "moddynout" and "d3eigv", to reconstruct the stress history for the whole model or for selected parts / elements, and then run fatigue damage computation.

```
*FATIGUE_MODAL_DYNAMIC
$    stres     index    restrt
         0         0         1
$     filename
dyn.results/moddynout
dyn.results/d3eigv
```

To run the transient modal dynamic analysis, the keywords needed in addition to the keywords used in a standard transient analysis (connectivity, boundary condition, material models, etc.) include.

***CONTROL_IMPLICIT_GENERAL**

***CONTROL_IMPLICIT_EIGENVALUE**

***CONTROL_IMPLICIT_MODAL_DYNAMIC**

Particularly,

- In ***CONTROL_IMPLICIT_EIGENVALUE**, one needs to turn on the flag "mstres" to get modal stress in d3eigv.
- In ***CONTROL_IMPLICIT_MODAL_DYNAMIC**, one needs to set "dtout" which defines the modal dynamics file (moddynout) output interval. This file saves the modal coordinates at each time step. It will be used later to recover the stress history during the transient response.

```
*CONTROL_IMPLICIT_MODAL_DYNAMIC
$   mdflag      zeta   md_strs      dtout      integ       nsid
         1     0.029         1      0.001
d3eigv
```

For fatigue part, the following keywords are needed:

***MAT_ADD_FATIGUE**
***FATIGUE_MODAL_DYNAMIC**
***DATABASE_D3FTG**

***MAT_ADD_FATIGUE** defines fatigue properties (SN curve or EN curve) for the material models in the input deck. ***FATIGUE_MODAL_DYNAMIC** directs LS-DYNA to run fatigue analysis with two steps: 1) reconstruct stress history of the structure using the modal coordinates provided in "moddynout", and the modal stress provided in "d3eigv"; 2) compute cumulative fatigue damage ratio by the rainflow counting method, using the stress history from 1), and the fatigue properties of materials defined by ***MAT_ADD_FATIGUE**. The computed cumulative fatigue damage ratios are dumped to a binary plot database "d3ftg", as directed by the keyword ***DATABASE_D3FTG**. This database is accessible by the software LS-PrePost. The cumulative fatigue damage ratio plot and the expected fatigue life plot can be provided by LS-PrePost.

Examples

In this section, two examples are provided to demonstrate the effectiveness and efficiency of this new method.

A cantilevered shell strip

A simple cantilevered shell strip model is used in this example. The model is constrained at the left end and is subjected to nodal force excitation at the right end. See Figure 1.

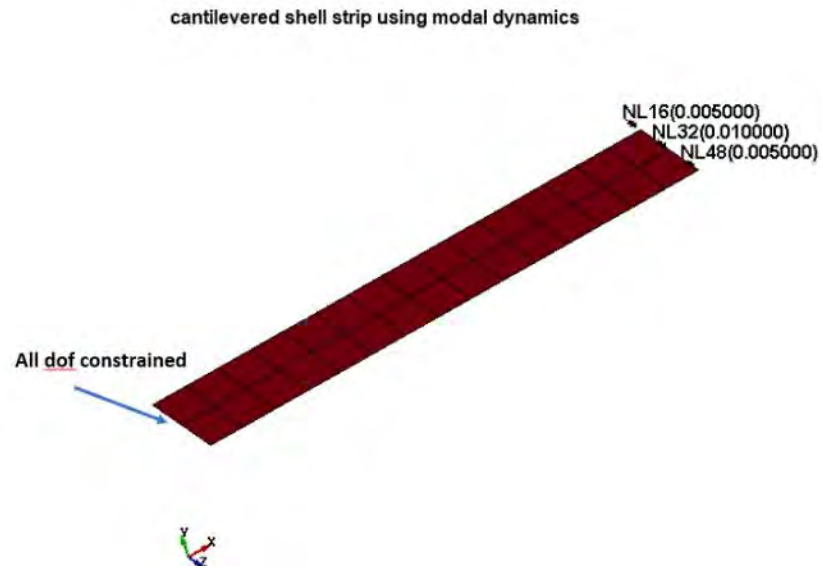


Fig.1: a cantilevered shell strip subjected to impulse nodal force excitation

For the 1st step, this problem is solved using transient modal dynamic analysis. The modal coordinates are dumped to moddynout for every 0.001 second. For the 2nd step, the keyword ***FATIGUE_MODAL_DYNAMIC**, ***MAT_ADD_FATIGUE**, and ***DATABASE_D3FTG** are added to the input file. "restrt" in ***FATIGUE_MODAL_DYNAMIC** is set to be 1, so that LS-DYNA can read in the modal coordinates from "moddynout" file, and the modal stress vectors from "d3eigv", to reconstruct the stress history along with time.

For this model, the fatigue results given by ***FATIGUE_MODAL_DYNAMIC** and by ***FATIGUE_ELOUT** are both provided for cross validation purpose.

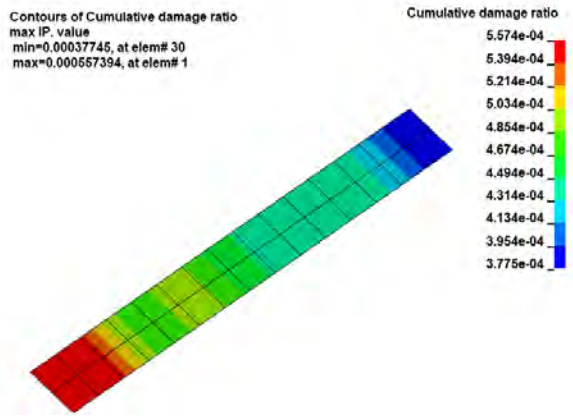


Fig.2: Cumulative damage ratio by ***FATIGUE_ELOUT**

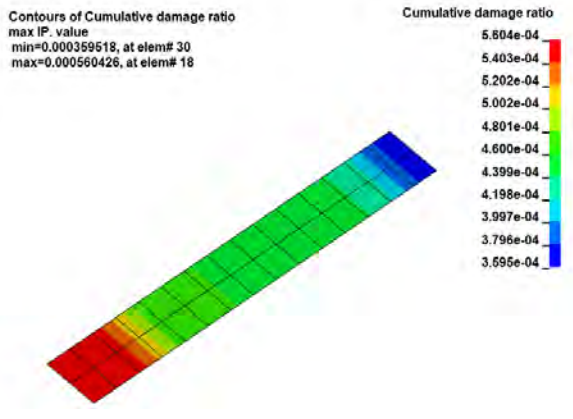


Fig.3: Cumulative damage ratio by ***FATIGUE_MODAL_DYNAMIC**

As we can see from the Figures above, the cumulative damage ratio results by ***FATIGUE_ELOUT** and ***FATIGUE_MODAL_DYNAMIC** are very close. For the maximum cumulative damage ratios given by the two methods, their relative difference is only around 0.5%.

A metal support deck

For this example, a metal support deck which is used in automotives is considered. This deck is connected to other parts by the edge of the two holes and is subjected to a high frequency sinusoidal nodal force excitation at the center of the model (see Figure 4).

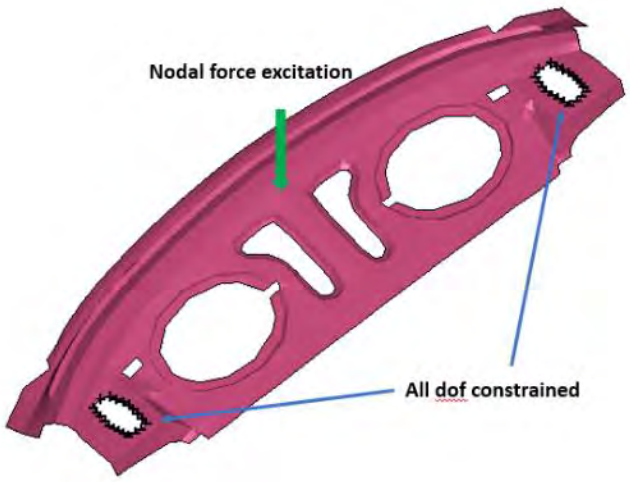


Fig.4: a metal support deck for automotive

The cumulative damage ratio results at two time instants are given in Figure 5 and Figure 6.

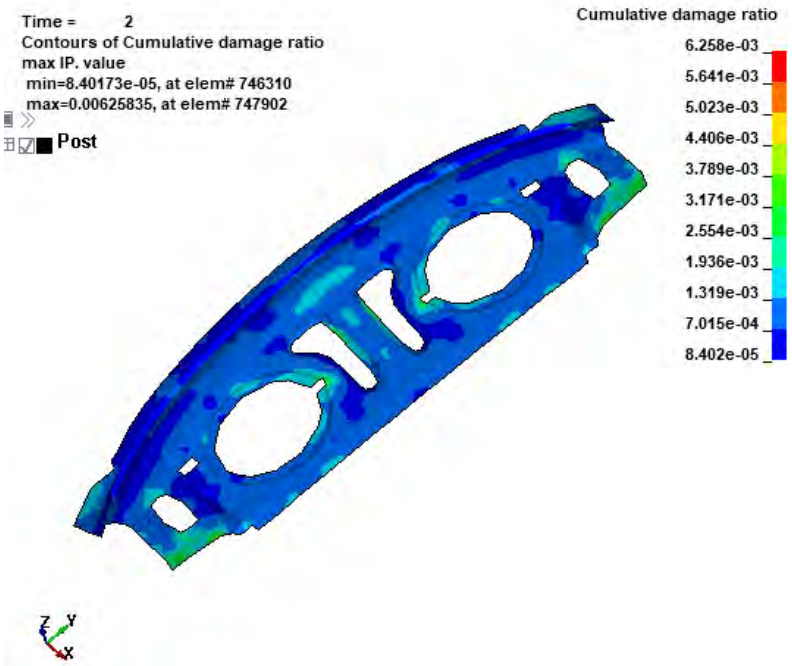


Fig.5: Cumulative damage ratio at time instant 2 second.

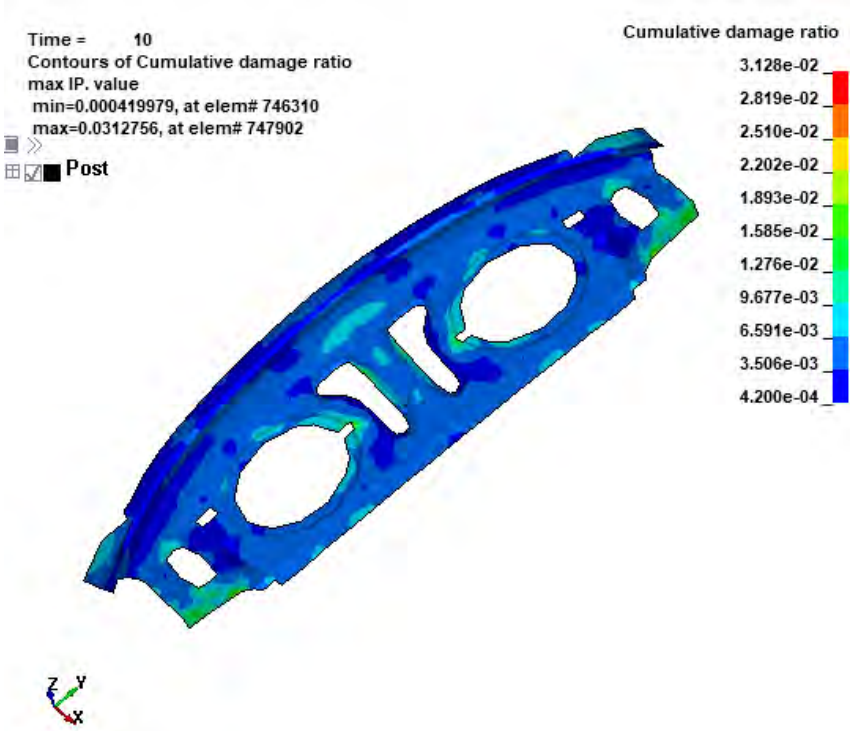


Fig.6: Cumulative damage ratio at time instant 10 second.

One can see that the fatigue damage grows along with time, even though its distribution pattern has no much change.

For this problem, we compared the CPU time and the Hard Drive usage by different fatigue analysis methods in LS-DYNA. The results are listed in Table 1.

Table 1. CPU cost and Hard drive usage using different methods.

Method	CPU time (seconds, 1cpu)	Hard Drive usage
***FATIGUE_MODAL_DYNAMIC**	255	78 MB moddynout
***FATIGUE_ELOUT**	659	7 GB binout
***FATIGUE_D3PLOT**	462	15 GB d3plot

It is clear that the new method by ***FATIGUE_MODAL_DYNAMIC** is more efficient than the others, for both the CPU cost and the hard drive usage. This advantage makes the new method more appropriate for the fatigue analysis of large scale problems.

Summary and future work

This paper presents a new fatigue solver in LS-DYNA, which is based on stress reconstruction using modal coordinates and modal stress in transient modal dynamic analysis. This method is proved to be more efficient than the other methods which rely on the stress results from elout or d3plot databases.

For the future work, we are going to change the moddynout file from ASCII format to binary format, to further reduce the Hard drive usage, and to preserve more digits in modal coordinates, so that one can get more accurate stresses and fatigue damage ratio results.

Literature

[1] LS-DYNA Keyword Users' Manual, Livermore Software Technology (LST), An Ansys Company, 2021

[2] Arnaud Ringeval, Yun Huang, Random Vibration Fatigue Analysis with LS-DYNA. 12th International LS-DYNA Users Conference, Dearborn, Michigan, 2012

[3] Yun Huang, Zhe Cui, Philip Ho, Chengju Zhang, Application of LS-DYNA in Structural Fatigue Analysis and Post-Processing with LS-PrePost. 14th International LS-DYNA Users Conference, Dearborn, Michigan, 2016

[4] Jithesh Erancheri, Ramesh Venkatesan, Durability Study for Tractor Seat Using LS-DYNA. 14th International LS-DYNA Users Conference, Dearborn, Michigan, 2016

[5] Yun Huang, Zhe Cui, Advances in Fatigue Analysis with LS-DYNA, 15th International LS-DYNA Users Conference, Dearborn, Michigan, 2018

[6] Jong S. Park, Ramakrishna Dospati, Ye-Chen Pan, Amit Nair, Random Vibration Fatigue Life Simulation of Bolt-on Metal Brackets using LS-DYNA, 15th International LS-DYNA Users Conference, Dearborn, Michigan, 2018

[7] Zhe Cui, Yun Huang, Recent Developments in Time Domain Fatigue Analysis with LS-DYNA. 4th China LS-DYNA Conference, Shanghai, China, 2019

[8] Tengteng Wang, Random Vibration Fatigue Analysis for Bracket of PAB Module with LS-DYNA. 4th China LS-DYNA Conference, Shanghai, China, 2019

[9] Yun Huang, Anders Jonsson, Marcus Lilja, Multiaxial fatigue analysis with LS-DYNA. 16th International LS-DYNA Users Conference, 2020

[10] Hwawon Lee, Parvath Police, Amit Nair, “Random Vibration Fatigue Analysis Model Development from Explicit to Implicit in LS-DYNA”, 16th International LS-DYNA Users Conference, 2020

[11] Yun Huang, Stefan Hartmann, David J. Benson, Random vibration fatigue analysis based on IGA model in LS-DYNA. Ansys TechCon 2020

BETA CAE Systems. www.beta-cae.com

BETA CAE Systems - ANSA

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

BETA CAE Systems μETA

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

Solutions for:

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

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

Inventium Suite™

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

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

PreSys

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

VPG

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

DYNAFORM

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

ESI Group www.esi-group.com

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

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

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

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

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

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

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

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

HYCRASH

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

JSTAMP/NV

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

JMAG

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

LST

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

LS-DYNA

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

LS-PrePost

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

LS-OPT

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

LS-TaSC

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

LST, AN ANSYS COMPANY Dummy Models

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

LST, AN ANSYS COMPANY Barrier Models

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

Material Sciences Corporation www.materials-sciences.com

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

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

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

MSC/LS-DYNA Composite Software and Database -

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

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

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

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

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

Oasys PRIMER

Key benefits:

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

www.oasys-software.com/dyna

Oasys D3PLOT

Key benefits:

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

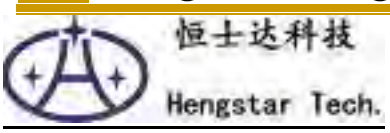
www.predictiveengineering.com

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

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

Our History

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



Shanghai Hengstar www.hengstar.com

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

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

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

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

Consulting

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

Cloud - HPC Services - Subscription JSOL

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

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

JSOL Corporation is cooperating with chosen cloud computing services

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

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

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

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

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

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

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

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

Platform Computation Cloud - CreDist.Inc.

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

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

Cloud - HPC Services - Subscription RESCALE

www.rescale.com

Rescale: Cloud Simulation Platform

The Power of Simulation Innovation

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

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

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

True On-Demand, Global Infrastructure

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

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

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

ScaleX Enterprise: Transform IT, Empower Engineers, Unleash Innovation

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

Rescale Cloud Simulation Platform

www.rescale.com

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

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

Industry-Leading Security

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

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

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

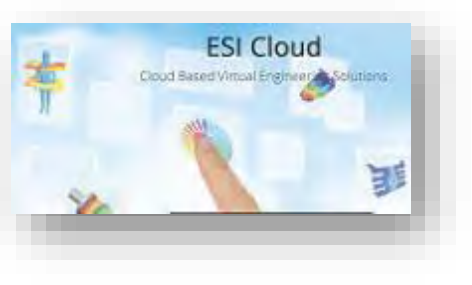
LSTC - DYNAmore GmbH JSOL Corporation

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

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

ESI Cloud Based Virtual Engineering Solutions

www.esi-group.com



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

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

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

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

Virtual Performance Solution:

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

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

The benefits of VPS hybrid on ESI Cloud include:

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

VPS On Demand

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

Key solution capabilities:

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

Result visualization

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

Key result visualization capabilities:

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

Collaboration

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

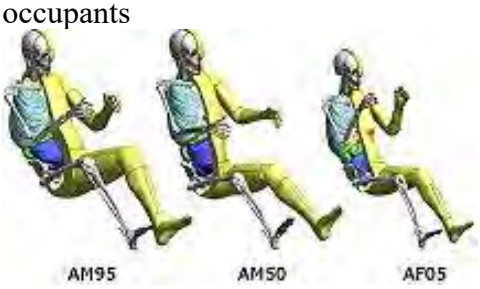
Key collaboration capabilities:

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

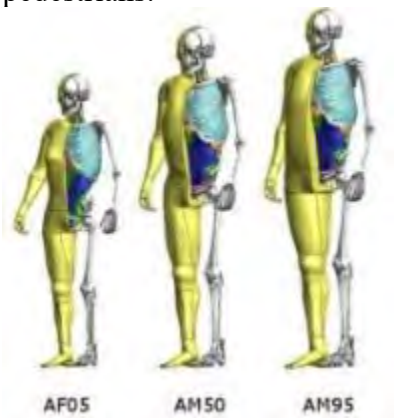
TOYOTA - Total Human Model for Safety – THUMS

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

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



and as standing model to represent pedestrians.



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

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

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

For information please contact: THUMS@lstc.com

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

LST, An ANSYS Company – Dummy Models

Crash Test Dummies (ATD)

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

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

e-mail to: atds@lstc.com

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

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

Models In Development

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

Planned Models

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

LST, An ANSYS Company – Barrier Models

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

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

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

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

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

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

FACEBOOK

BETA CAE Systems
CADFEM
ESI Group

TWITTER

BETA CAE Systems
ESI Group
ETA
CADFEM

LINKEDIN

BETA CAE Systems
CADFEM
DYNAmore Nordic
ETA
ESI Group

YOUTUBE

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

GOOGLE+

BETA CAE Systems