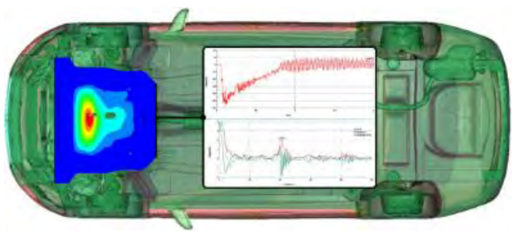
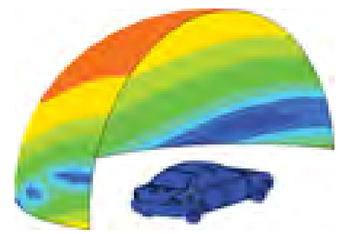
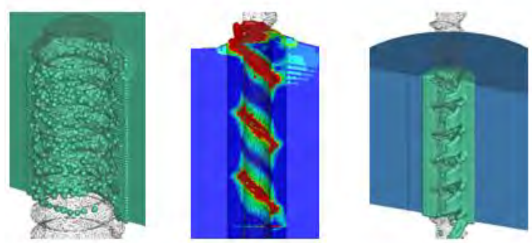
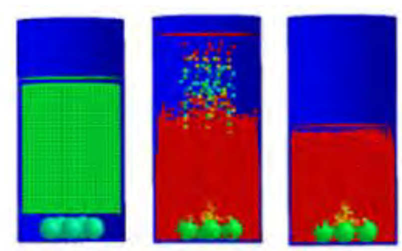
Issue 7, July 2021

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









Ansys LS-DYNA Special Collection

- LS-DYNA® Advanced CFD Analysis
- LS-DYNA® Discrete Element Method (DEM)
- LS-DYNA® Computational and Multiscale Mechanics
- LS-DYNA® SPG updates and new features
- LS-DYNA® NVH and Fatigue Analysis /Vibration, acoustic and fatigue solution package
- LS-PrePost® an Advanced Pre- and Post-processor
- LS-TaSC® New Release Version 2021 R2

LS-DYNA® New Feature and Application

- RVE Analysis in LS-DYNA for High-fidelity Multiscale Material Modeling

FEA Information Engineering Solutions

www.feapublications.com

The focus is engineering technical solutions/information.

Livermore Software Technology, an ANSYS,Inc. 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

BETA SIMULATION SOLUTIONS

Ansys LST

DYNA MORE

eta Innovation Starts Here

DatapointLabs

恒士达科技 Hengstar Tech.

esi get it right®

KAIZENAT Keep Improving

FEA Not To Miss

LANCEMORE

Oasys LS-DYNA ENVIRONMENT

MSC Materials Sciences Corporation

JSOL

rescale

Finite Element Analysis Predictive Engineering

DELL

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

仿坤软件 LS-DYNA China

Table of contents

LS-DYNA® Advanced CFD Analysis

LS-DYNA® Incompressible CFD (ICFD) tool combines state-of-the-art numerical techniques that allow robust, scalable, and accurate simulations of fluid flows. Its ability to couple with the structural, thermal, and Discrete Element Method solvers make it an excellent option for multi-physics problems.

Applications:

- Fast transient external aerodynamics
- Cooling analysis
- Resin Transfer Molding for manufacturing of composites
- Turbomachinery
- Fluid-Structure Interaction inthe biomedical field

Features:

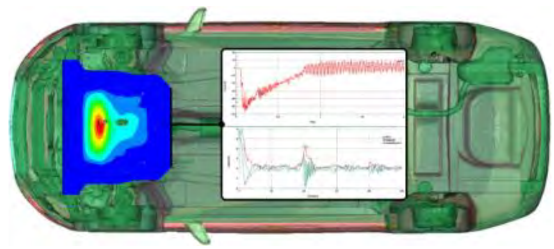
- FEM based
- Large library of RANS and LES turbulence models
- Automatic meshing andre-meshing
- Free surface flow
- Non-Newtonian flows
- Non-inertial reference frames and sliding mesh
- Porous mediamodels

Learn more at:

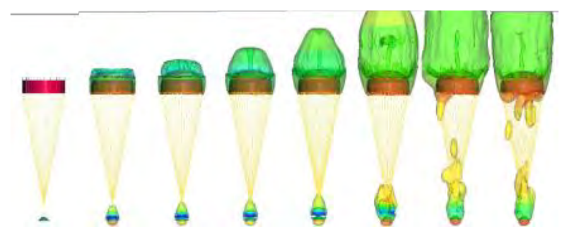
www.lstc.com/applications/icfd

YouTube:

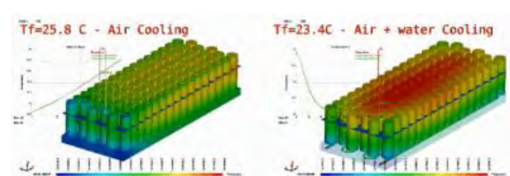
www.youtube.com/user/980LsDyna



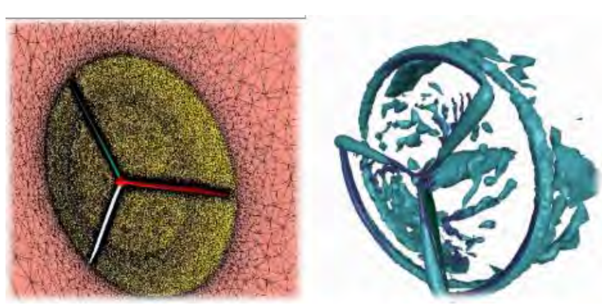
Accurate prediction f aerodynamic forces for flutter analysis.



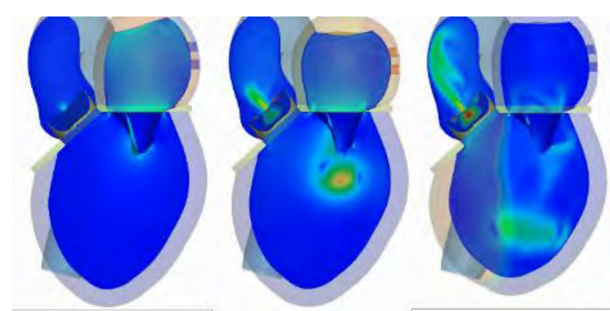
Finite Element Analysis of parachute deployment using porous membrane materials.



Battery cooling and general porpoise conjugate heat transfer applications.



Sliding mesh solver for Fluid Structure Interaction analysis of rotating structures.



Strong Fluid Structure Interaction (FSI) coupling with a focus on bio-medical applications

LS-DYNA® Discrete Element Method (DEM)

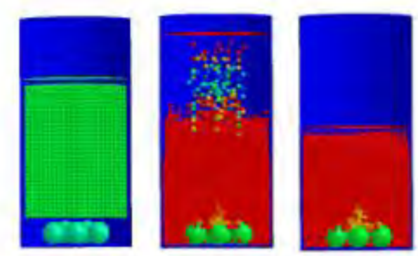
Discrete Element Method is a discontinuous approach for modeling particulate media by considering the interaction of each individual particle with other particles, structure, force field and surrounding media. In DEM, both translational and rotational trajectories of individual particles are calculated by evaluating their contact based on Newton's equation of motion. Due to its power in investigating phenomena occurring at the length scale of the diameter of particles, DEM becomes a perfect tool for use in design and optimization of particle handling and process operations. The addition of the bond models in LS-DYNA® DEM further expands its application to the area of cemented granular material such as rocks and concretes.

Applications:

- Granular flow, e.g., hopper and funnel flow
- Powder filling and packaging
- Particle sorting, mixing, and segregation
- Soil mechanics, e.g., mine blast, bulldozing
- Tire rotating on sand or gravel
- Storage and transportation
- Non-spherical particle simulation
- DES Particle coating
- Uniaxial, Brazilian, and triaxial tests for elastic material

Features:

- DEM particles are geometrically rigid and deformation is accounted for by allowing a small overlap when determining the contact force
- The bonded model enables the capture of microstructure evolution, deformations, dynamics and forces within the particulate system in detail
- Capillary forces are considered for wet particles
- Multiple coupling options are available between DEM parts and/or between DEM and FEM, beam, CFD or other mesh free methods (such as SPH)
- Injection of non-spherical particles is enabled through the bonded model
- Adaptive conversion of solid elements to DEM particles when handling severe deformations



Particle Mixing and Packing

Non-spherical Particle & DES coating

Particle Blast Simulation

Youtube Channel:

https://www.youtube.com/user/LSTCandDYNAmore

Twitter:

https://twitter.com/LSTCandDYNAmore

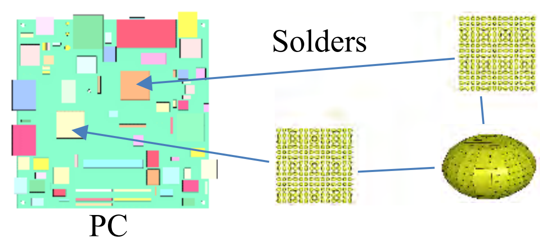
LS-DYNA® Computational and Multiscale Mechanics

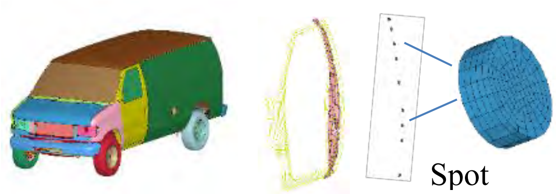
Intelligent Manufacturing, Advanced Material Design & Integrated Structural Analysis

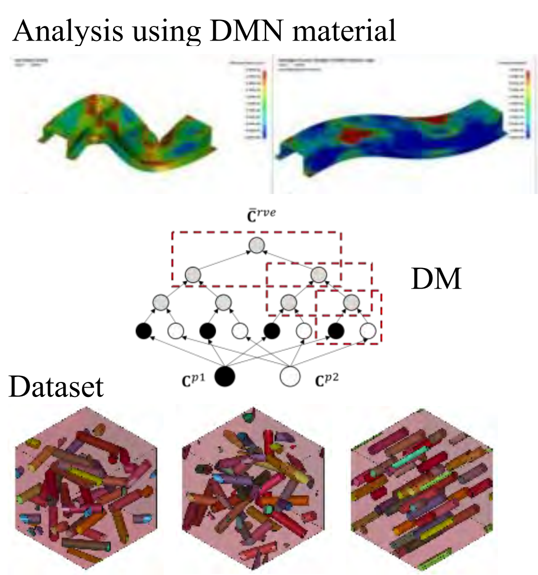
LS-DYNA® integrates advanced finite element and machine learning algorithm for solving some of the most challenging multiscale problems in manufacturing processes, material design, and structural analysis. Such problems typically involve large deformation, material failure and separation, and/or crack 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





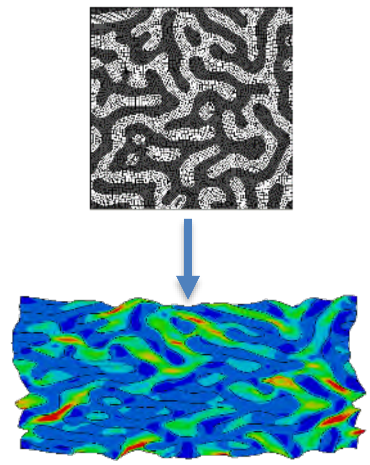


Short Fiber Reinforced Composite

- 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

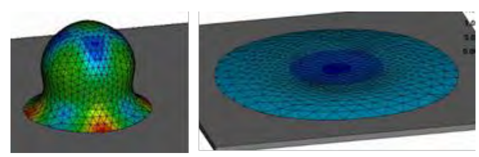
Representative Volume Element (RVE) Package

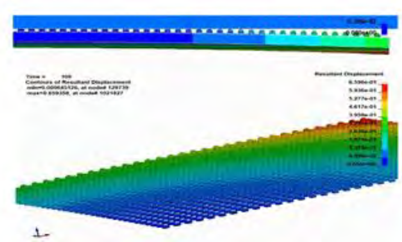
- Predict macroscopic constitutive behaviors (2D & 3D)
- Automatic periodic boundary condition set-up
- Support many nonlinear material models
- Capture finite deformations of microstructures
- Applications: multiscale virtual design & characterization
  - Fiber reinforced composites
  - Particulate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



Incompressible Smooth Particle Galerkin (ISPG)

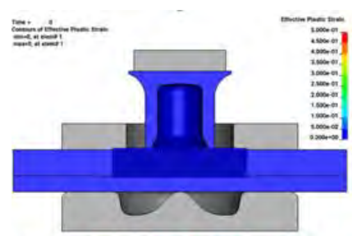
- Fully implicit formulation
- Model the surface tension and wall adhesion efficiently
- Capable to simulate the solder reflow with complex models, e.g. solder mask defined (SMD) pad & NSMD
- Coupled with implicit thermal and structure solvers
- Scalable computation for large scale thermal-mechanical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

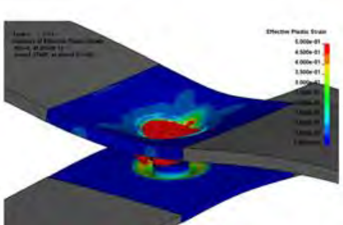


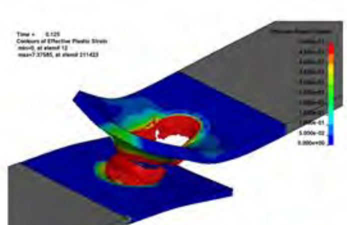


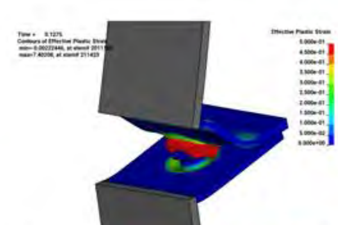
SPG Two-stage analysis

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









LS-DYNA® SPG updates and new features

Smoothed Particle Galerkin (SPG) method is a new generation numerical method for dynamic material failure simulation. SPG theory was proposed by Livermore Software Technology (LST) R&D team in 2014. The first LS-DYNA version was released in 2017. A momentum consistent SPG (MCSPG) formulation was developed in 2019 which improves the original version in both accuracy and efficiency and enables coupled thermal mechanical analysis as well. The 2021 version supports more features with more efficient computation.

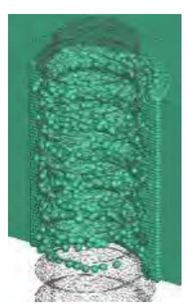
One of the new features is the two-stage SPG simulation for mechanical joining process (e.g., flow drill screwing – FDS, self-piercing riveting – SPR) and joint performance analysis. The 1st stage simulates the joining process, and the 2nd stage analyzes the joint strength (e.g., pullout, lap shear, etc.) while the histories (stress, strain, displacement / coordinate) from the 1st stage are used as initial conditions. It is straightforward to perform a two-stage SPG analysis. Users only need to include *INTERFACE_SPG_1 in the 1st stage calculation to output desired histories, and *INTERFACE_SPG_2 in the 2nd stage calculation to read desired histories. Details can be referred to LS-DYNA keyword manual or https://www.lstc-cmmg.org/mmps.

Different from conventional FEM, SPG does not erode elements during the material failure simulation. It preserves the desired conservation properties in mass and momentum, leading to an accurate solution for a wide range of material failure applications in various industries, such as automobile, electronics, civil, wood, and biomedical.

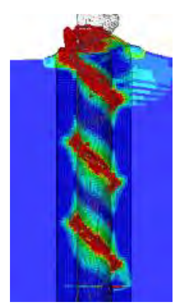
Typical SPG applications include but not limited to screwing, machining, joining, and many other manufacturing process simulations involving material failure and removal, thermal mechanical effect, and severe deformation. SPG can perform various material failure analyses for metal, concrete, wood, foam, rubber, composites, etc.

Screw driving analyses

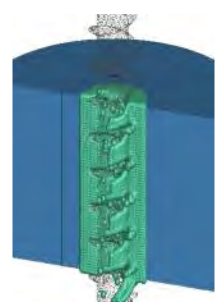
1. Isotropic/orthotropic ductile and brittle materials
2. Thread forming
3. Driving force and torque
4. Different geometry of screws
5. Joint strength after driving



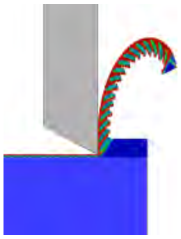
Metal taping



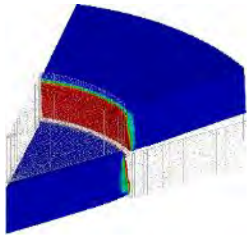
Concrete drilling



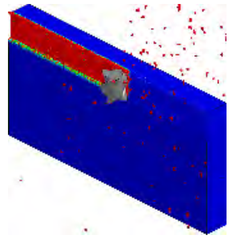
Wood screwing



Cutting



Blanking



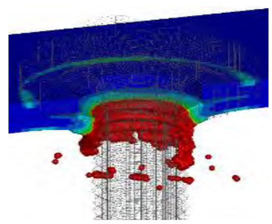
Grinding

Manufacturing process analyses

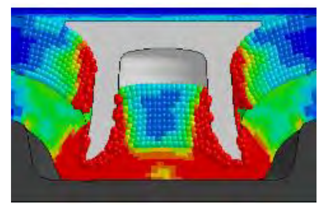
1. Ductile (metal) materials
2. Coupled thermal mechanical
3. Applied force
4. Shear banding and debris forming

Mechanical joining process

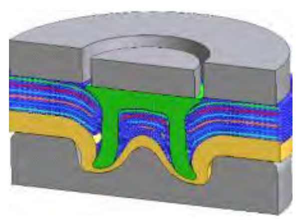
1. Similar / dissimilar materials
2. Thread forming or mechanical locking
3. Joining force and/or torque
4. Joint strength after joining
5. Thermal effect available



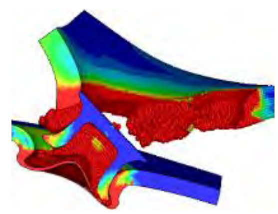
FDS



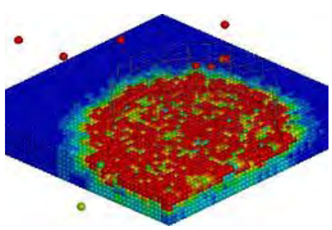
SPR



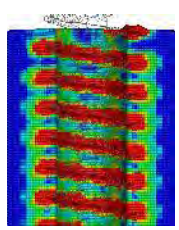
SPR of LFRP



Lap shear of SPR

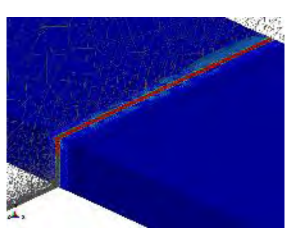


Wafer grinding



Dental screwing

Needle insertion



Ultrasonic cutting

New area applications

1. Semi-conductor material
   Wafer grinding
2. Biomedical materials
   Dental screwing, drilling
   Joint strength
3. Hyper-elastic material
   Ultrasonic cutting of rubber
   Cutting force

LS-DYNA® NVH and Fatigue Analysis
Vibration, acoustic and fatigue solution package

NVH and Fatigue problems are very common in many industries (e.g. cabin noise of vehicles and fatigue damage and failure of metal and composite structures). LS-DYNA NVH and Fatigue Analysis modules provide solutions for these problems. Particularly they can provide more realistic and reliable simulation and analysis of these problems in working with other multiphysics solvers embedded in LS-DYNA and other Ansys products (for example, fatigue simulation with pre-stress due to metal forming; E-motor noise simulation with Ansys Maxwell and Ansys Mechanical); This type of analysis is crucial to the comfort, safety and integrality of vehicles and other structures.

Solvers:

- FRF (Frequency Response Function)
- SSD (Steady State Dynamics)
- Random vibration
- Response spectrum analysis
- Acoustics (by finite element / boundary element / spectral element methods and modal methods)
- PML for wave absorbing boundary
- Fatigue analysis

Application:

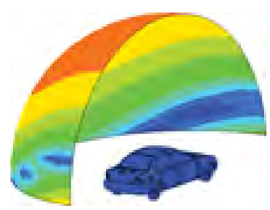
- Assessment of dynamic properties of structures
- Energy transfer path analysis
- Shaker table testing simulation
- NVH of vehicles and aircrafts
- Noise control of machines and engines
- Fatigue analysis of structures
- Safety evaluation of buildings under earthquake induced ground motion
- Underwater acoustics with fluid-structure coupling
- Ultrasonic wave from piezoelectric sensors

Features:

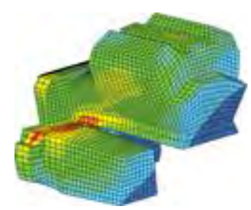
- Seamless vibro-acoustic analysis
- Seamless vibro-fatigue analysis
- Options to run:
  - Equivalent Radiated Power (ERP),
  - Acoustic Transfer Vectors (ATV),
  - Incident waves,
  - Acoustic eigenvalue analysis.
- Element and panel contribution for acoustic sensitivity studies
- Multiple fatigue analysis methods

Displacement RMS of Body In White

Cabin noise computation by FEM acoustic solver

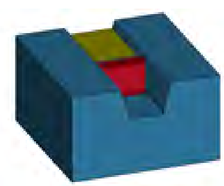


Vehicle radiated noise by BEM acoustic solver

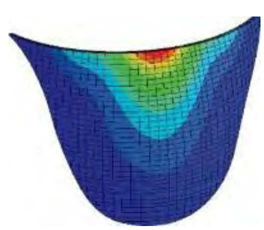


ATV plot for engine by BEM acoustic solver

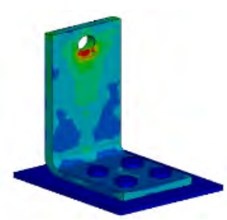
Ultrasonic sensor by SEM acoustic solver



Dam-reservoir system with PML foundation



Response spectrum analysis of dam under earthquake excitation



Fatigue damage analysis of a bolt

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

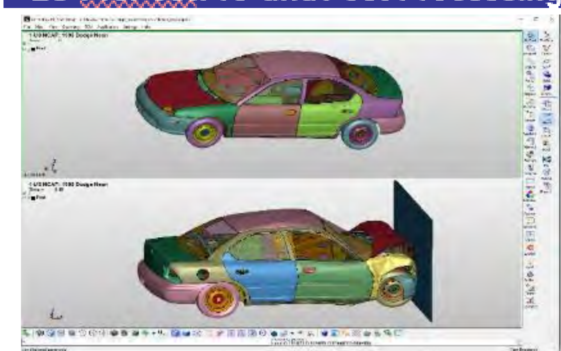
Other General Functions

- 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

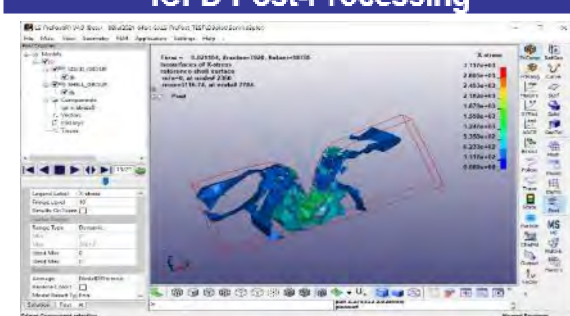
- Airbag folding
- Comprehensive model checking including contact initial penetration check
- Dummy positioning
- Metal forming process setup
- Seatbelt fitting

LS-PrePost Pre-and Post-Processing



LS-DYNA Geometry and Meshing

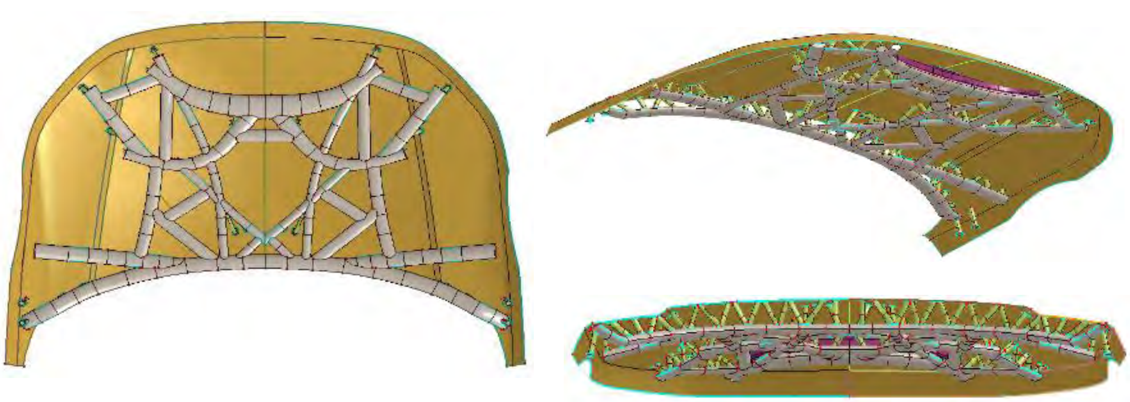
ICFD Post-Processing



LS-TaSC® New Release Version 2021 R2

With the 2021 R2 release, LS-TaSC continues to expedite the optimization design process, enabling complex large nonlinear MDO problems to be tackled efficiently. A summary of top new features in LS-TaSC 2021 R2 is described as below.

1. ***Efficiency Improvements for the Multidisciplinary Design Optimization*:** Following the computation capabilities of the previous release in addressing constrained multidisciplinary topology optimization problems, 2021 R2 aims to improve the computational efficiency in multipoint analysis in the optimization design workflow. Disciplines with analytical design sensitivity information such as NVH load cases will no longer do a full multipoint analysis – the values for the sibling designs will be predicted instead. This enables the function evaluation time for NVH analyses at sibling points to be decreased enormously in each iteration.
2. ***Facilitating Various Needs for NVH Design Optimization*:** The 2021 R2 supports computations using the design sensitivity information in NVH load cases for different design needs, such as a frequency gap "$f_3 - f_2$", and a normalized frequency constraint "$f_2/1000$". This also enables a frequency for a particular design purpose to be used as an objective for NVH design optimization.
3. ***Support for Structures using Rubber Materials*:** The 2021 R2 now supports structural designs for using materials *MAT_MOONEY-RIVLIN_RUBBER, *MAT_HYPERELASTIC_RUBBER, and *MAT_OGDEN_RUBBER.
4. ***Minimum Member Size Control*:** The latest release provides better control of minimum member size for parts, which can be selected from the method panel.
5. ***Better Support for Design of Head Injury Criterion*:** The 2021 R2 now enables to extract and define the HIC (Head Injury Criterion) responses directly from the constraint panel.
6. ***Support for STL Outputs*:** The 2021 R2 now provides the STL outputs of the isosurface plots of the optimized designs. The STL outputs can be used to create CAD version of the optimal designs through third-party tools.

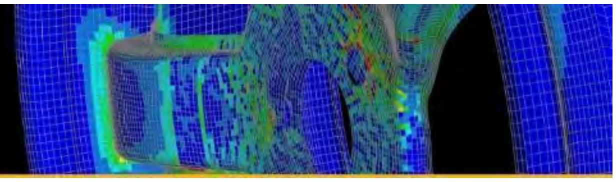


The CAD model of a LS-TaSC hood design as created using ANSA. CAD model courtesy of BETA CAE Systems.

New version download: https://ftp.lstc.com/user/ls-tasc/2021R2/

Ansys

Ansys Blog



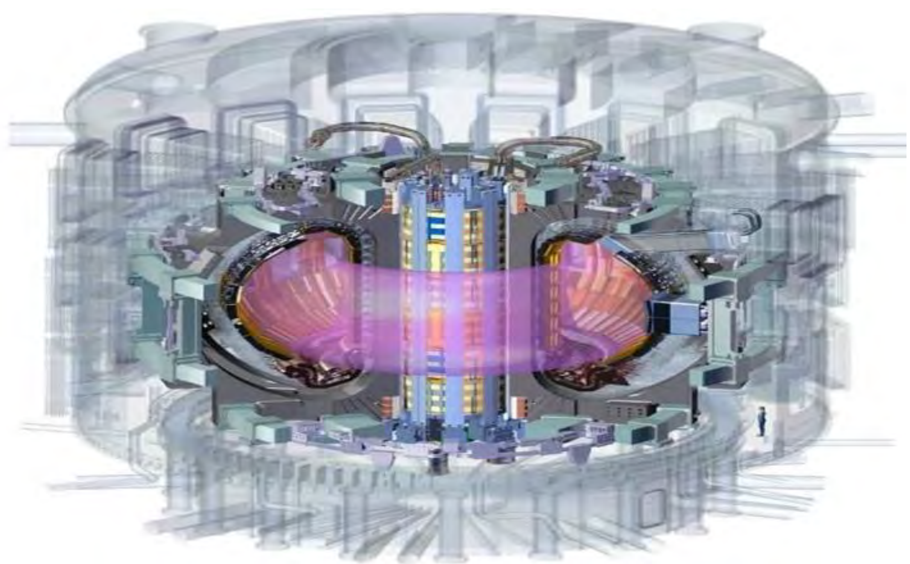
July 9, 2021

Energizing Highly Sustainable Next-Gen Nuclear Power Plants with Ansys Simulations

Authors: Anthony Laurent Arduin
Strategic Account Manager, Ansys

In 2025, the International Thermonuclear Experimental Reactor (ITER), the world's largest nuclear fusion reactor experiment, will begin a 15-year period of experimentation to demonstrate the scientific and technological feasibility of fusion energy. Engineers from around the world have collaborated for decades to develop this modern marvel, which is designed for record-breaking fusion power gain. Fusion is the energy that powers the Sun and other stars.

Could fusion be the ideal energy source? It just might be. Fusion generates little emissions or radioactive waste, while releasing nearly four million times more energy than a chemical reaction – such as the burning of coal, oil or gas – and four times as much energy as nuclear fission reactions (at equal mass). But there's a catch: To produce a self-sufficient fusion reaction, the plant must heat up hydrogen isotopes to more than 150 million° C, which then forms an electrically charged gas known as plasma. To maintain this tremendous heat and reliably contain this gas, ITER Organization engineers needed to create the ITER's electromagnetic (EM) structure design, relying on Ansys simulation solutions to cut material demands and costs.



Credit © ITER Organization, iter.org

"Ansys simulation solutions will continue to help our team to satisfy the required safety and accuracy levels for this first-of-a-kind initiative," said Bernard Bigot, Director-General of the ITER Organization in recent press release. "For ITER to achieve hydrogen fusion at industrial scale requires unprecedented levels of engineering precision, so it is incredibly important that our simulation software is highly reliable and efficient. Ansys has consistently delivered that capability to us for many years, enabling our team to safely push boundaries, dream bigger and deliver Earth's biggest fusion reactor."

Demonstrating the Feasibility of Fusion

The ITER project will demonstrate the feasibility of fusion to generate commercial electricity. If it succeeds, the world may soon harness a practically limitless source of clean energy, offering many of the advantages of traditional fission nuclear plants, but tremendously safer. Think of it as a "holy grail" of clean energy as it would revolutionize how humanity produces electricity.

Diving a bit deeper, ITER would provide an electricity supply that isn't dependent on what most renewables are based on, such as the wind or sun, nor does it produce CO2. And as the world moves toward electrifying transportation, the need for electricity will radically increase. The more that can be done with less carbon, the easier it will be for world leaders to meet climate goals and slash greenhouse gases.

The implications for the energy sector are vast because nuclear fusion plants could replace potentially all coal-fired and nuclear power plants.

Overcoming ITER's Engineering Challenges

To create the mechanical design of the main components within the immense ITER device, engineers needed to optimize the machine's mass and structure to ensure correct usage of construction material. Additionally, they needed to ensure that everything performed well from an EM standpoint because the plasma produces currents that fluctuate over time, creating significant EM forces within the structures. Using Ansys simulations helped ITER engineers visualize the current flows and then transfer the forces that they calculated into structural models. For example, they used Ansys Fluent to optimize the cooling of various components.

Fluent also plays a key role in building a set of documents that ITER engineers use to ensure a robust system design and that the system adheres to stringent project and industry safety requirements. Additionally, Ansys Mechanical plays a critical role in helping engineers build the structural supports that secure ITER's base.

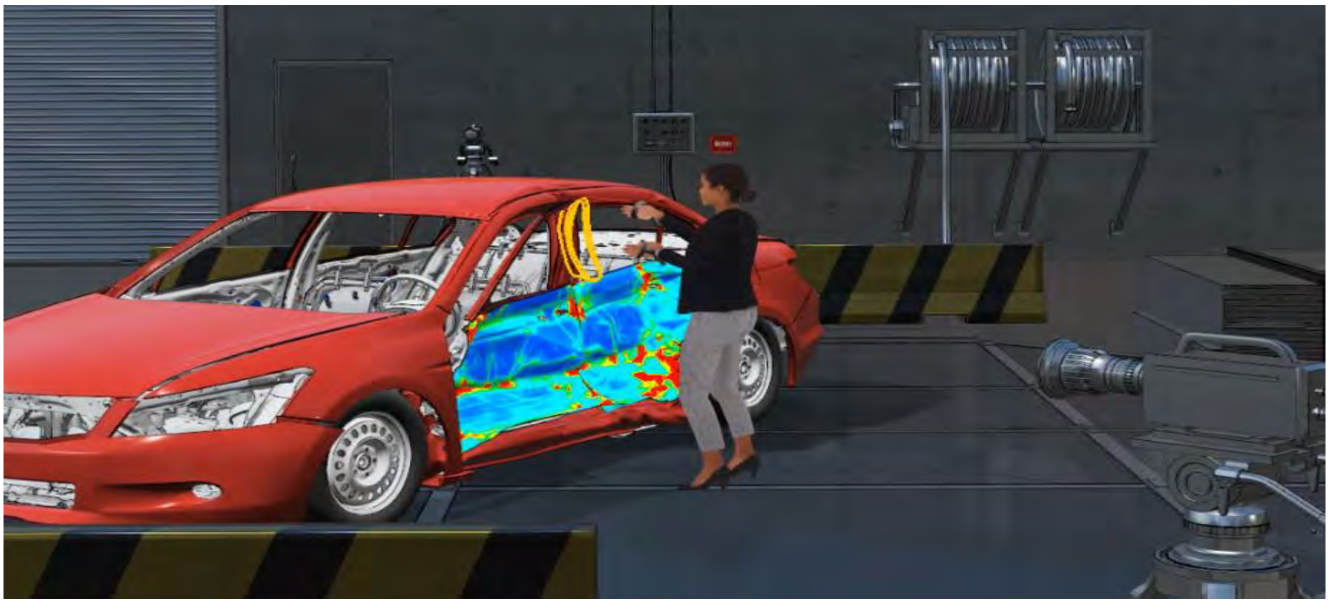
"To power the sun and the stars, light atoms fuse at very high pressures and temperatures. Replicating this process on Earth with ITER will help solve the world's energy demands, however, engineers must overcome extremely difficult design challenges," said Prith Banerjee, chief technology officer at Ansys. "Using Ansys simulations, ITER engineers are rapidly building a structurally sound fusion power reactor, drastically reducing the EM structures' material content and substantially decreasing the plant's development cost – driving the delivery of clean, sustainable energy for our planet."

ITER's highly innovative engineering team is effectively using Fluent and Mechanical to turn the dream of clean nuclear power into a reality. Check out how engineers are using Fluent and Mechanical to substantially enhance product development across countless industries.

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…

July 5, 2021

BETA CAE Systems announces the release of the v22.0.0 of its software suite



About this release

BETA CAE Systems is thrilled to announce the release of the major version v22.0.0 of its product line. With solutions for every stage of the product development process, the new version offers all means to manage the emerging, increased complexity in engineering simulation.

The new series is the proof of how the key-selection of innovative tools and functions can accelerate your processes, boost performance and stand up to the most demanding needs of the simulation and analysis market, across a range of industries.

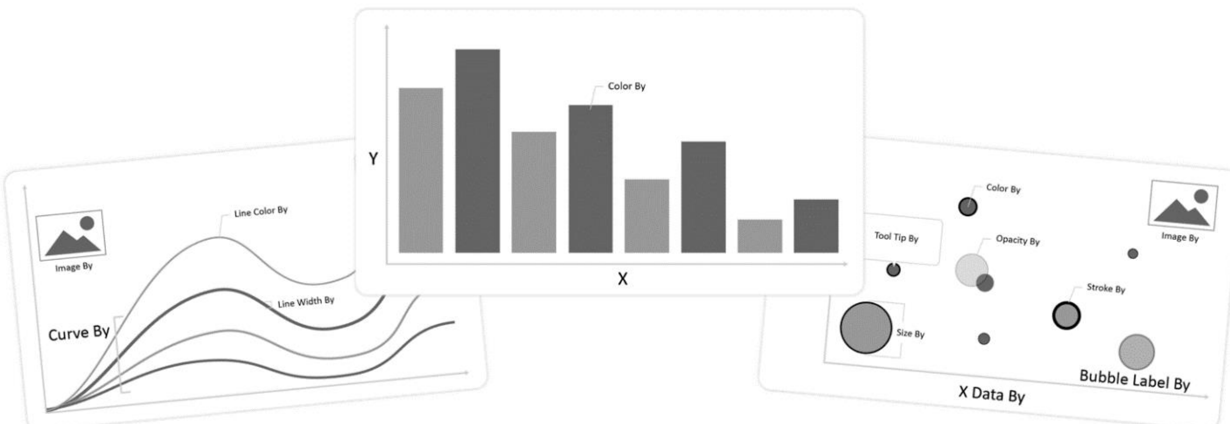
Do not miss:

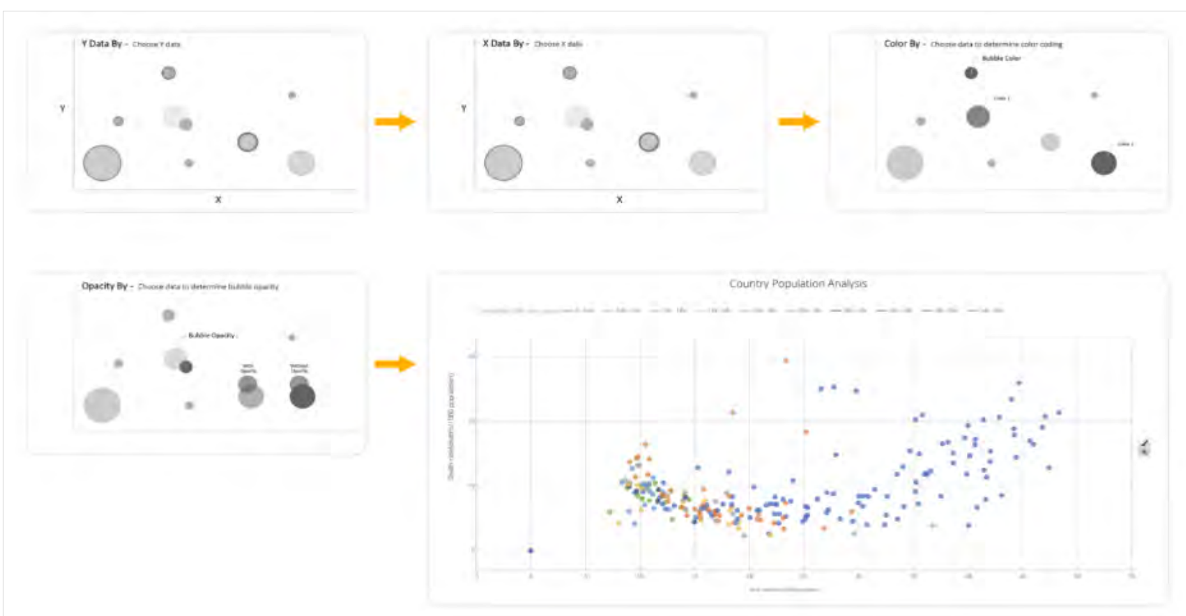
- The revamped GUI in ANSA, that enhances user experience through a series of features, focusing on highlight performance, real-time information on elements, module buttons' re-organization and more.
- The fast, effortless Mesh generation, through the total Mesh redesigned layout with new, dedicated toolsets and functions re-organization.
- The accelerated performance of CFD algorithms on Volume Mesh generation.
- The enriched Optimization capabilities in ANSA, as well as in SOL200 area with EPILYSIS.
- The constantly expanding VR capabilities in ANSA and META.
- The increased Collaboration capabilities through META and Web Interface.
- The progressing Machine Learning integration in KOMVOS through ANSA.

More Detail from website

Progressive Data Visualization in Simlytiks

In an effort to make visualizing data in the Simlytiks application more efficient, d3VIEW has created detailed placeholders for previewing visualizations before creation. Each visualizer option has its own illustration placeholder detailing its function.Preview some of these for vertical bar chart, scatter plot and curve plot below.





www.d3view.com | support@d3view.com

Author: Christian Frech christian.frech@dynamore.de

Online and Onsite Event! Register today!

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

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

Invitation
We kindly invite you to the 13th European LS-DYNA Conference at October 5-6, 2021 in Ulm, Germany, and online.

Schedule
4 October, 6 p.m. Registration and Get Together
5 October, 8 a.m. Start of the conference on-site
5 October, 8 p.m. Gala Dinner
6 October, 8.30 a.m. Conference day 2
6 October, 9 a.m. Streaming day 1
7 October, 9 a.m. Streaming day 2

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.

Registration
We are looking forward to numerous registrations and to welcome you again at our conference. Please register here:
www.dynamore.de/en/conf21-reg

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.

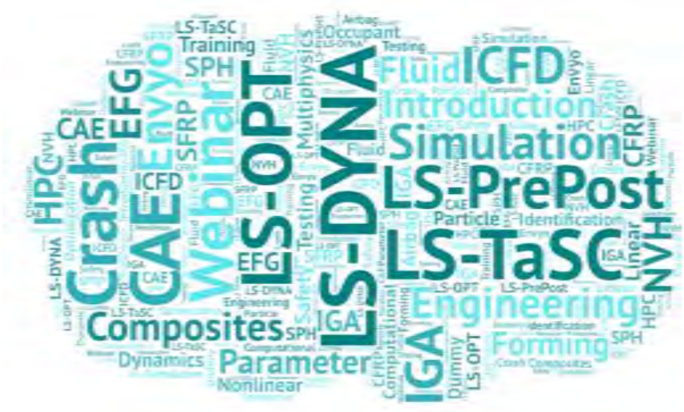
Exhibiting and sponsoring
Please request further information.

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 DYNAmore DYNAmore FRANCE DYNAmore NORDIC DynaS+

Author: Christian Frech christian.frech@dynamore.de

Webinars and on-demand Video-Seminars 2021



Online trainings in August/September/October

Webinars	
Transferring phase transformation data from MAT_244 to MAT_254	6 August
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
Introduction to Welding Simulation	12-13 October
Dummy/Pedestrian Impactor Modeling	14-15 October
Accompanying seminars at the 13th European LS-DYNA Conference (on-site if possible)	
Introduction to SPG Method for Manufacturing and Material Failure Analysis	4 October
Electromagnetism in LS-DYNA	4 October
Explosives Modeling for Engineers	4 October
Element Types & Nonlinear Aspects	7 October
NVH, Frequency Domain, Fatigue	7 October
Resistive Heating and Battery Modeling	7 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.

Predictive Maintenance of Conveyor Belts with Digital Twins

An interview with expert Jörg Arloth
Wednesday, June 30, 2021/By Peter Larsson

Profitability is one of the fundamental goals in the development and operation of complex machines and systems. New, data-based methods (big data, machine learning) now enable smart services (e.g. predictive maintenance) and help meet this requirement by avoiding unplanned downtimes and reducing operating and maintenance costs. I spoke to my ESI colleague Jörg Arloth, Account Manager Mining, and asked about the current state of the art.

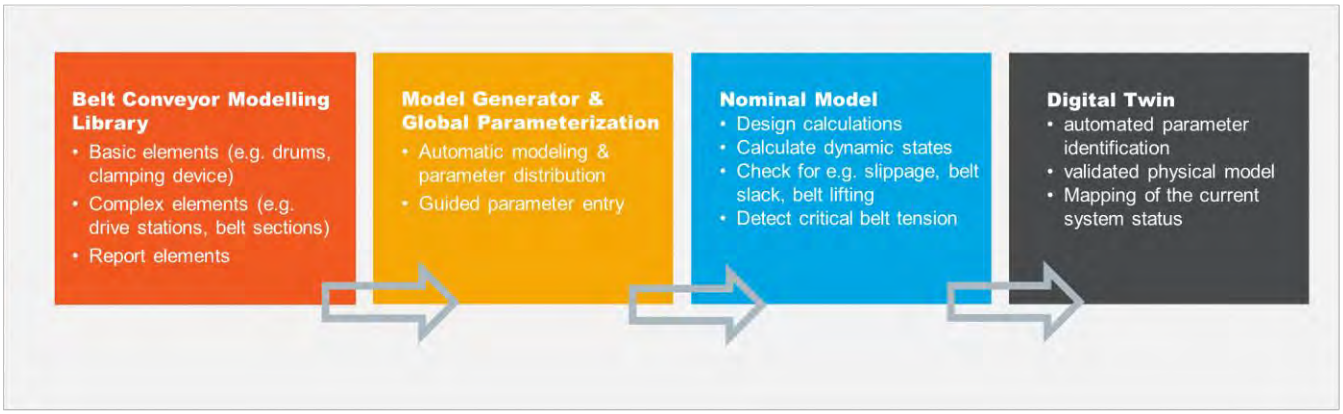
Q. Big data and machine learning are trending in digitization in all industries. Looking at manufacturers and operators of mining machines, where do you currently see most of the starting points?

Jörg. A current challenge is bringing together data-based methods and expert knowledge from real-life practice and assigning existing measurement data to the causes of defects or wear and transfer them to other systems. Only then can the development of data-based pattern recognition be crowned with success. The physical system simulation, combined with proven workflows, offers a solution that facilitates practical application and supports communication between experts from various fields. Models can be created automatically; desired error behavior can be integrated in a targeted manner and a wide variety of virtual operating data can be generated.

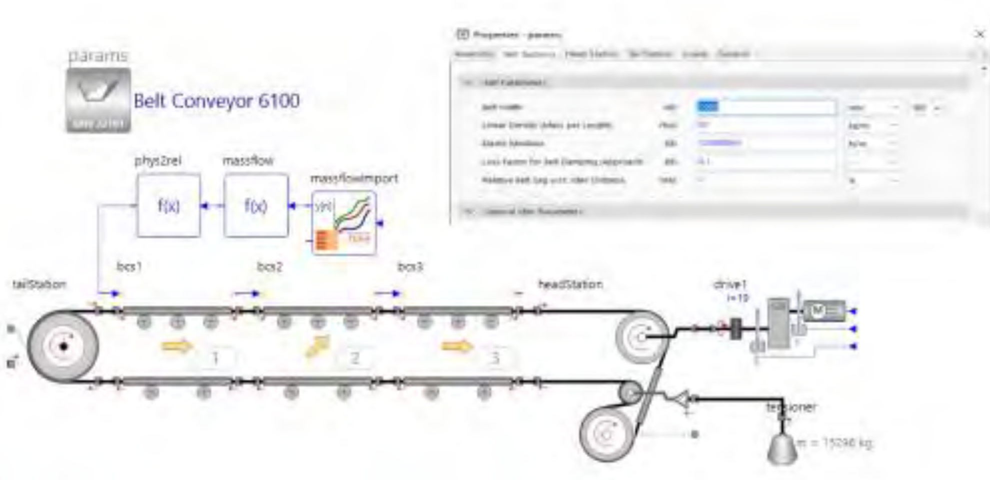
Q. In your lecture at the belt conveyor conference, you outlined the workflow of an SRA-based condition analysis of belt conveyors to support predictive maintenance strategies. The starting point and basis of this concept is a digital twin using the example of a belt conveyor. How exactly is that supposed to work?

Jörg. If, as with belt conveyors, the system model can be generalized, it can be created automatically. With system simulation, the generalization is achieved by applying the belt conveyor library in ESI SimulationX

From the included elements, a script-based model generator creates the system model. I briefly sketched the workflow from the library to the digital twin:



The resulting simulation model describes the structure of the system and is parameterized, centrally, in a data container. The picture on the left shows a simulation model of a belt conveyor that is then loaded with a standardized mass flow that was generated from real measurement data and thus represents a real loading process over time (load).

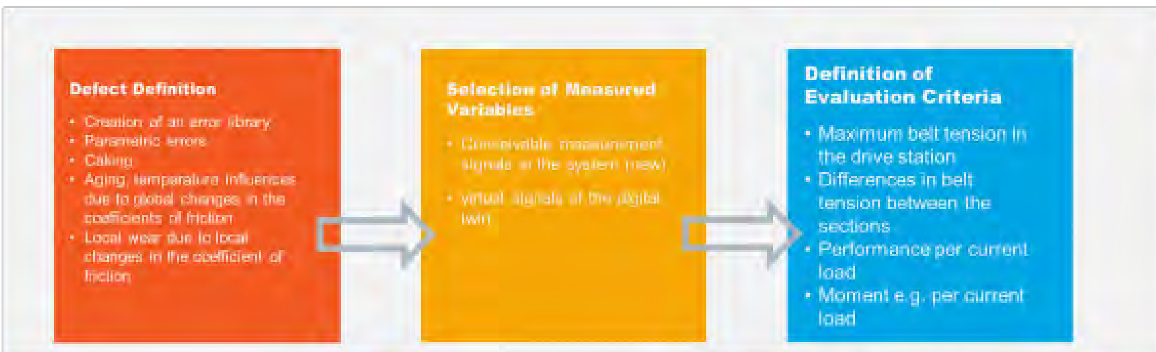


Q. A very important criterion for the quality and reliability of the created digital twin is data consistency. What can you say about the data measured in real-life scenarios versus the one you obtained using simulation?

Jörg. That is a very important aspect. To validate our simulation model, we took the data from a conveyor system that was in operation. If you compare the real measured values with the simulation data in the picture on the right, you can see a sufficiently large correspondence between the simulation and measurement data. The quality criterion met has been met.

Q. Let's move on to another important point - the SRA analysis. What exactly is behind the new technology?

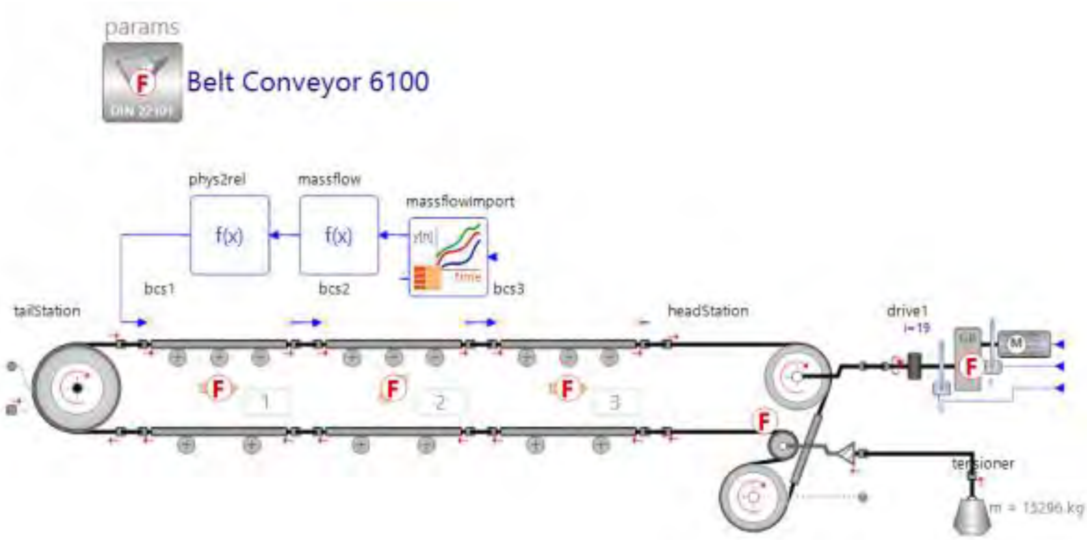
Jörg. In the image below, I show you an example of our SRA analysis approach for the belt conveyor application where you can see how new ideas can be developed, tested and evaluated in the simulation model. This is key for ideas to be checked completely virtually, upfront, in development instead of inventing complex and unrealistic measurement strategies. As a side effect, such a digital approach may even generate the need for new measurement technology.



Q. So, looking at performance in real-life operations, you have collected a multitude of possible errors and put them together in a library?

Jörg: Exactly. The fault model we generated is very much what you see here. All model elements with parametric errors are marked with a red "F" and are part of an automatically generated error library.

You can clearly see the structural view of the belt conveyor model with implemented parametric errors. This includes errors that can be assigned directly, such as caking on the drums, but also complex errors that are characterized by a change in the coefficient of friction. This can be done globally in order to mark general aging and wear. But it can also be done locally in order to map steady-state influences or effects originating from the moving belt.



In addition to the usual measurement signals, such as speed (motor, rotating drums) and drive torque, the current total load is measured using virtual sensors and the belt tension forces of the individual model sections in this test. This is to clarify the question of whether such different failure effects could be detected and identified, even if several failures happen to occur at the same time.

For example, a broad range of features are continuously monitored, like the maximum belt tension on the drive drum, the normed power and the normed drive torque per current load as well as the normed belt tensile forces based on the nominal behavior (based on a nominal model). This means that engineers could calculate a large number of variants (defect size and combination) and their impact on the conveyor's performance.

Q. Machine learning algorithms are used to analyze the effects of errors on the model. How exactly does this contribute to a predictive maintenance strategy?

Jörg. Correct. We use machine learning algorithms to evaluate the results of the variant calculation. The quality of decision trees, for example, shows clearly the dependencies between the generated features and the errors and how precisely threshold values can be defined. While caking can be detected very easily using the quotient of different engine and drum speeds, it is much more difficult to assign aging effects. Due to the different loading conditions during operation, very robust threshold values must be selected.

To illustrate this better, let's look back at the image above where you see the standardized belt tension forces of the individual sections. The starting point of the idea is that the belt tension forces can be measured for each section in the model (here BCS1, BCS2, BCS3, HeadStation). The belt tension forces under consideration are localized running up onto the respective belt section. Without developing or installing appropriate sensors, engineers know what conclusions they shall draw from these signals. For this purpose, the belt tension forces are normalized using the calculated belt tension forces of a nominal model.

Q. That was a lot of theoretical and technical facts. In conclusion, how would you summarize the topic for our speed readers?

Jörg. Intelligent (smart) services and the machine learning algorithms used for them require the provision of sufficient quantities of operational data. This already applies to the preparation of service strategies for new systems that do not have any field data at this point in time. Data-based analyses of systems, as described in the article, are an effective means of generating such realistic amounts of data. The inclusion and evaluation of sources of error and their interactions represent, on the one hand, an opportunity to include the experience of operating and service personnel and make them usable on a broad basis, and, on the other hand, to protect the system from sensor overloading. The SRA-based workflow shown supports data acquisition and the development of new solutions, both for increasing system availability and for predictive maintenance strategies.

For more insight on delivering safer, cleaner, and more productive machinery with Virtual Prototyping visit our dedicated Heavy Machinery page.

Do you want to leverage discussions on key technologies to face not only challenges related to product design, but also product manufacturing, product assembly, and product operations? Watch our ESI LIVE Heavy Machinery.

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

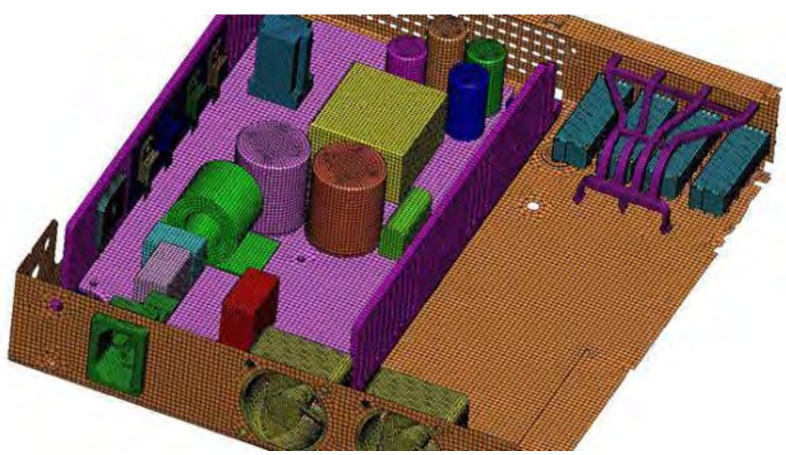
eta Innovation Starts Here

PreSys

Pre/Post Processor

PreSys is an advanced multi disciplinary CAE pre/post processing tool for the development of finite element analysis models. User can generate a solver specific input file ready to be analyzed from an existing CAD data and then post process the results in a single integrated GUI. It offers an intuitive user interface with many streamlined functions, allowing for fewer operation steps with a minimum amount of data entry. PreSys allows the user to analyze product designs, view simulation results, and predict how the product will perform in a given circumstance. PreSys works the way you do – The menus, toolbars, and many interface features can be customized. Including a model explorer feature that provides streamlined data navigation. Developed by the leader in the creation and implementation of new computer-aided engineering (CAE) tools and methodology, PreSys is ETA's 4th generation pre/post-processor. It delivers the capability to handle finite element modeling with clarity and ease. Presys also have two dedicated modules for setup a drop test and fluid-structure interaction apart from regular pre and post processing.

PreProcessing



A good quality FE model is a key foundation to a quality simulation. Presys provides extended features to develop a robust FE Model. The advanced graphical capabilities and efficient menu structure means significant modeling time reduction. Though primarily Presys is designed to prepare and modify LS Dyna models quickly. Presys supports all the LS Dyna keywords along with descriptions of each field which makes editing and understanding those fields quite easy.

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 - July pdf format

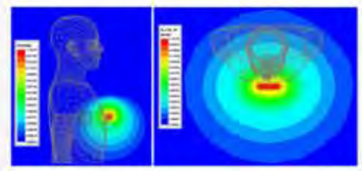
Time to get your cup of coffee (and let's have chocolate chip cookies - shake them first, so the calories fall out) and make sure you didn't miss these ANSYS blogs on health care.

June 10, 2021

Women in Technology: Overcoming Gender-Based Stereotypes in Biomedical Engineering

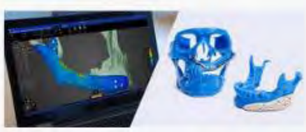
May 18, 2021

Pharmaceutical Industry Simulates Lyophilization to Speed Manufacturing



May 10, 2021

Simulate How Permanent Magnets Impact Medical Implants



Blog

December 16, 2020

Repairing Bone Loss with Simulation-Generated, Patient-Specific Implants | Ansys

Blog

February 6, 2020

Explore the Important Challenges of Medical Device Design | Ansys

Blog

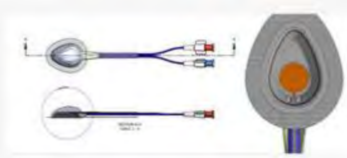
February 20, 2019

Overview of the Regulatory Pathway for In Silico Testing of Medical Devices | Ansys

Blog

September 25, 2019

What is PMMA and How Is It Used in the Medical World? | Ansys



Blog

October 5, 2018

Simulation Optimizes a Chemotherapy Implant to Treat Pancreatic Cancer | Ansys

Blog

March 13, 2017

3 Milestones on the Way to the Medical Digital Twin | Ansys

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

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.)

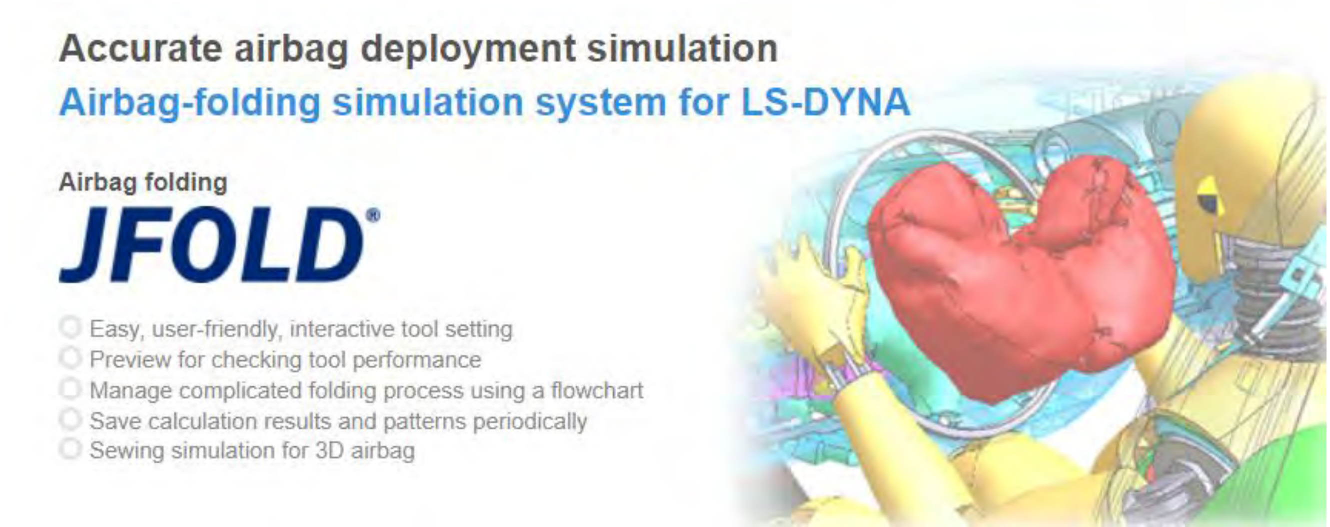
Instructor：

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.

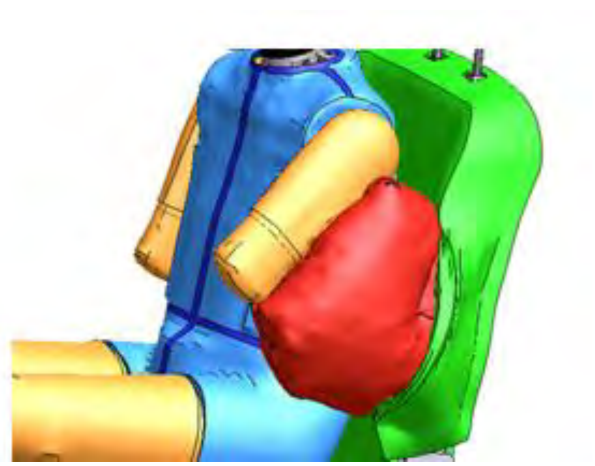
Duration and Date: (1 hours Webinar)
July 27 (9:00AM-10:00AM)
Language: Mandarin
Contact: Xixi Fei
Tel. :021-61630122 Mobile:13524954631 Email:Training@hengstar.com

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.



JFOLD Features

Towards more accurate airbag deployment simulation



JFOLD was developed to fold airbags for automotive crash simulation. JFOLD can be used to generate a folded airbag model using LS-DYNA simulation, regardless of the complexity of the geometry.

Airbags are one of the important safety devices for protecting the occupant during an accident: airbags are folded compactly and stored in the interior. The deployment behavior of an airbag depends on the pattern through which it is folded. The risk of occupant injury during airbag deployment, the out-of-position problem, considerably affects the occupant's safety performance.

Recently, the demand for more accurate airbag deployment simulation to improve the occupant's safety has been increasing. Building a folded airbag model with complicated geometry was an issue for CAE engineers to address.

JFOLD can manage the complicated folding process of an airbag using a flowchart in an easy-to-understand tree view. Users can build, manage, and view the airbag models in various folding patterns. The intuitive and interactive GUI facilitates the operation of defining the position and behavior of the folding tools.

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:

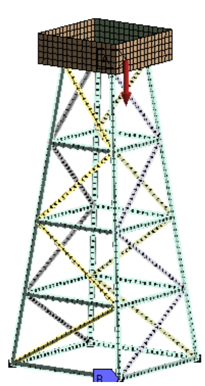
The CAD model was subjected to meshing such as the columns and cross members were modelled with beam elements.
The cross sections of the beams were extracted using ANSYS Spaceclaim tool.
All the cross members and columns were assignments Elasto-plastic Structural steel material.

Loading conditions:

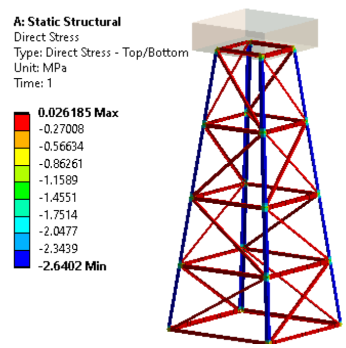
Considering the capacity of the water tank, the load was approximated as a point mass with gravity.
Mass of tank = 1000 kg

Boundary constraints:

All the bottom ground nodes were fixed in all directions.

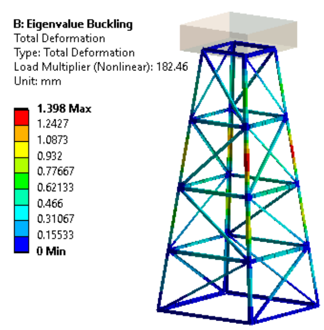


Beams stress results:



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:

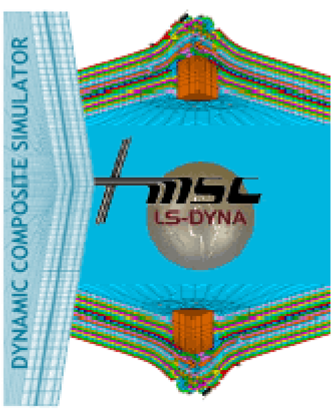
- The results calculated by the Eigenvalue Buckling analysis are buckling load factors that scale all of the loads applied in the Static Structural analysis.
- Hence, it’s typical to apply unit loads in the static analysis that precedes the buckling analysis.
- You can apply a nonzero constraint in the 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**

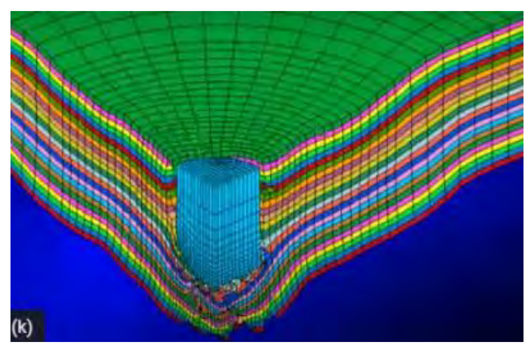
Providing engineering services to the composites industry since 1970. During this time, we have participated in numerous programs that demonstrate our ability to perform advanced composite design, analysis and testing; provide overall program management; work in a team environment; and transition new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

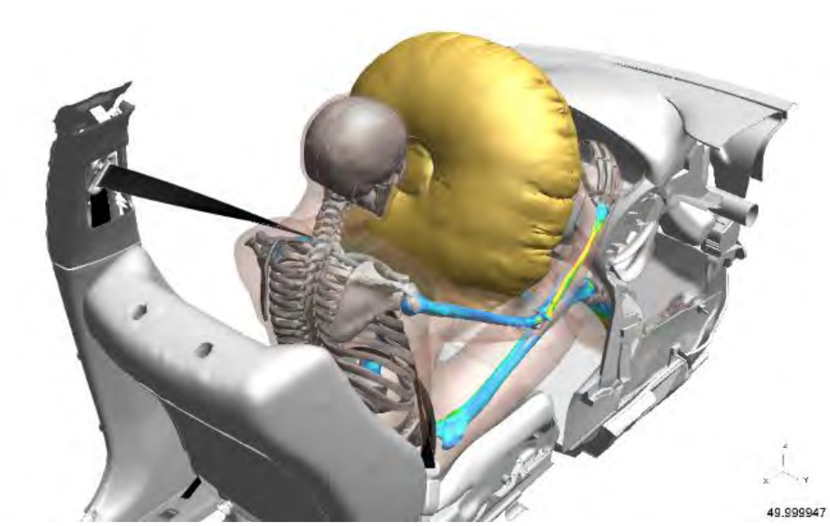
Dyna Fact Sheet (PDF)

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.

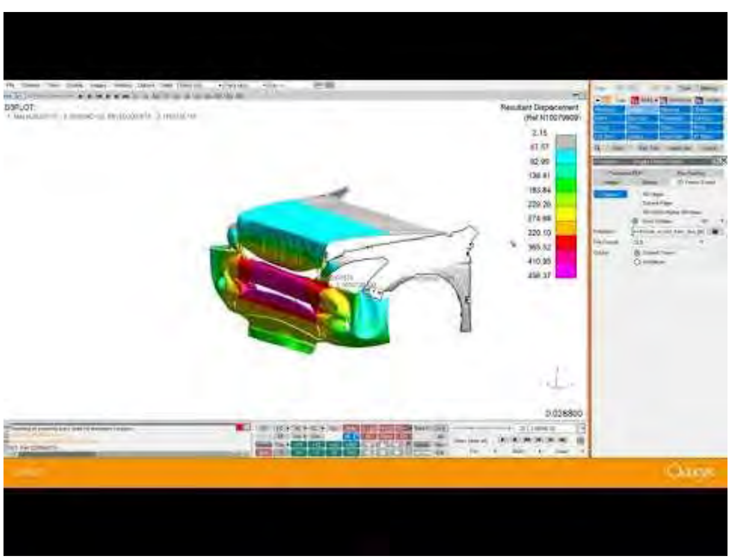
Oasys Ltd is the software house of Arup and distributor of the LS-DYNA software in the UK, India and China. We develop the Oasys Suite of pre- and post-processing software for use with LS-DYNA.



Oasys HBM positioning trees for all THUMS models

Toyota Motor Corporation now offers the THUMS model free of charge as part of its efforts towards a safe mobility society. Oasys human body model (HBM) positioning trees are now available for all royalty-free THUMS models (v4, v5, and v6 models).

Please visit our website for further information.



Top Tip video: Did you know?

Oasys D3PLOT Viewer

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 here to **view a** playlist of all the **Top Tip** videos.

LS-DYNA®

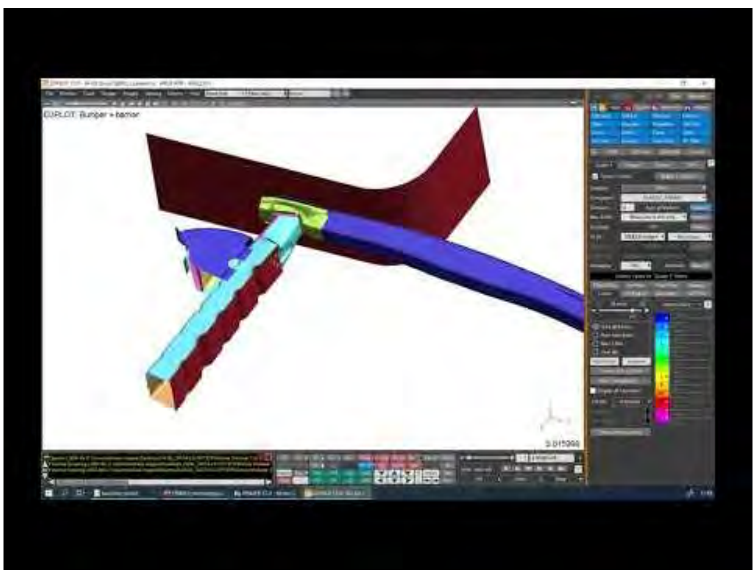
Upcoming webinar:
LS-DYNA: Recent Updates

12:30 BST on 7th September 2021

Featuring Richard Sturt
Arup Fellow

This webinar will cover a selection of recent developments in LS-DYNA including occupants, implicit, materials and multiphysics functionality.

Register here.



Webinar to watch again:
Pre-processing in LS-OPT with Oasys PRIMER

This webinar will demonstrate how to setup an Oasys PRIMER pre-processing stage in LS-OPT, which has been made simpler in V7 with the addition of native support for Oasys PRIMER. We will set up an Oasys PRIMER stage to apply part morphing, allowing the parts shape to be incorporated into an optimisation study.

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

Predictive Engineering

www.predictiveengineering.com

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

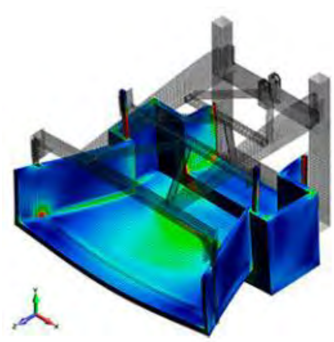
LS-DYNA has been one of Predictive's core analysis tools pretty much since we got started in 1995. It is an amazing numerical workhorse from the basic linear mechanics (think ANSYS or Nastran) to simulating well nigh the impossible. At least that is the way I feel at times when the model is not solving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

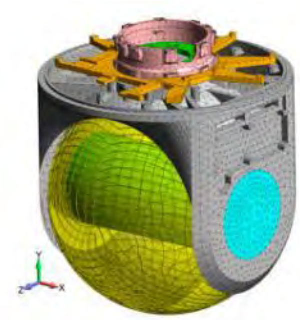
Predictive Engineering – Western States ANSYS LS-DYNA Distributor

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

Rescale

www.rescale.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

Shanghai Fangkun

http://www.lsdyna-china.com

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

仿坤软件 LS-DYNA China

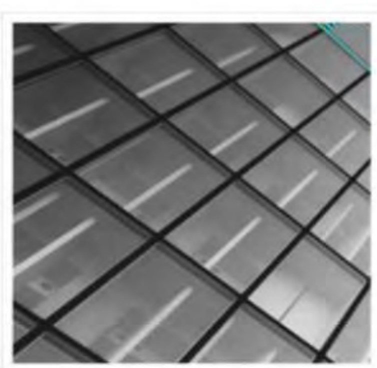
Shanghai Fangkun Software Technology Ltd

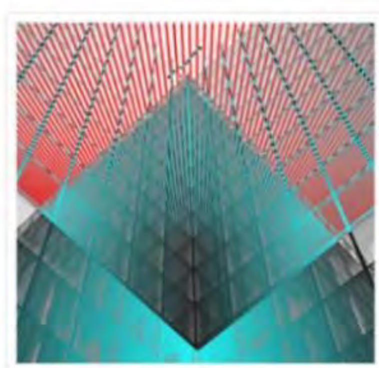
Shanghai Fangkun Software Technology Ltd. was authorized by ANSYS Inc as the domestic master distributor of LS-DYNA software. Shanghai Fangkun is fully responsible for domestic sales, marketing, technical support of LS-DYNA. By integrating and managing a wide range of resources such as LS-DYNA agents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

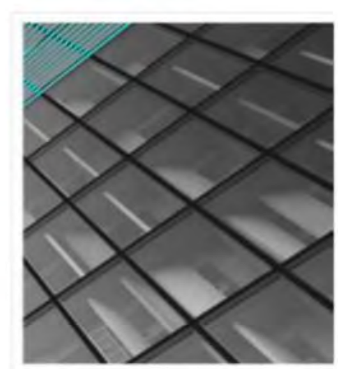
Based on the strong technical support and developing capability from ANSYS Inc, Shanghai Fangkun attracts a group of top LS-DYNA application engineers and commit to provide LS-DYNA technical support in the automotive industry, electronics industry, rock-soil, aerospace, general machinery and other industries.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

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

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







Contacts

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

Postcode： 201102 Tel：4008533856 021-61261195

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

LS-DYNA Training Plan in 2021

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

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

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

Terrabyte

www.terrabyte.co.jp/english

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

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

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

FE analysis

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

CFD analysis

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

EM analysis

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

Metal sheet

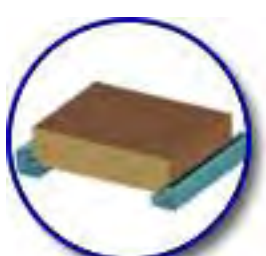
- JSTAMP is an integrated forming simulation system for virtual tool shop based on IT environment. JSTAMP is widely used in many companies, mainly automobile companies and suppliers, electronics, and steel/iron companies in Japan.

Pre/ Post

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

Biomechanics

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

RVE Analysis in LS-DYNA for High-fidelity Multiscale Material Modeling

Haoyan Wei, Dandan Lyu, Wei Hu, CT Wu

Computational & Multiscale Mechanics Group,
Livermore Software Technology, ANSYS,
Livermore, CA 94551, USA

1. Introduction

In modern engineering designs, advanced materials (e.g., fiber/particle-reinforced polymers, metallic alloys, laminar composites, etc.) are widely used, where microscale heterogeneities such as grains, inclusions, voids, micro-cracks, and interfaces significantly affect the macroscopic constitutive behaviors. Obviously, an accurate description of the multiscale material behaviors is of great importance to the success of material design and structural analysis.

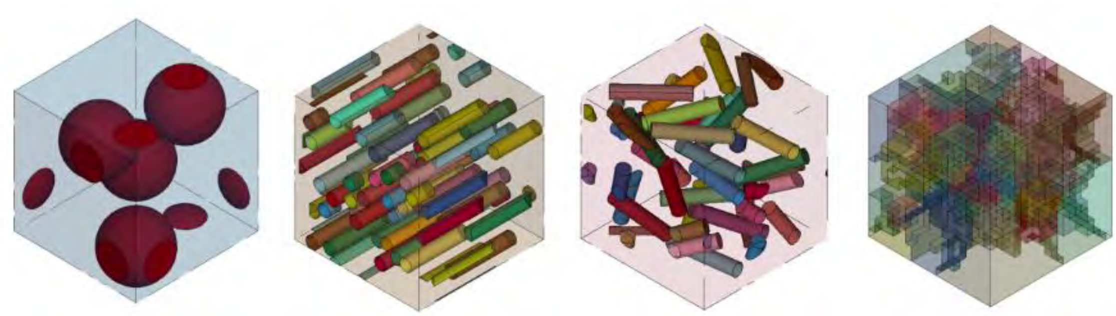


Figure 1. Illustration of the Representative Volume Element (RVE) for different composite materials

The *Representative Volume Element (RVE)* analysis method provides a rigorous means to obtain homogenized macroscopic material properties at the upper length scale from the properties of the material constituents and structures at a lower length scale. Recently, we have developed an RVE module (keyword: *RVE_ANALYSIS_FEM) in the multiphysics simulation software LS-DYNA to enable high-fidelity virtual testing of numerically re-constructed material samples at user-specified characteristic length scales. In this article, a brief introduction to this new feature will be given.

2. Basic Theory of RVE Analysis

RVE is a material volume that statistically represents a sampling of microstructural heterogeneities included at a material point of a continuum body [1]. The size of RVE should be chosen to be large enough so that any volume of an increased size can be considered equally representative. Let us consider an RVE (refer to Figure 2 for a 2D illustration) in its undeformed initial configuration Ω, for which the external boundary can be expressed as $\partial\Omega = \cup_{\alpha=1}^{d} \partial\Omega_\alpha$, where $d = 2$ for 2D RVE models, and $d = 3$ for 3D RVE models. Further, we use $\boldsymbol{X}_\alpha^+ \in \partial\Omega_\alpha^+$ and $\boldsymbol{X}_\alpha^- \in \partial\Omega_\alpha^-$ to denote the microscale material points located on a pair of opposite external surfaces $\partial\Omega_\alpha^+$ and $\partial\Omega_\alpha^-$ of the RVE, respectively.

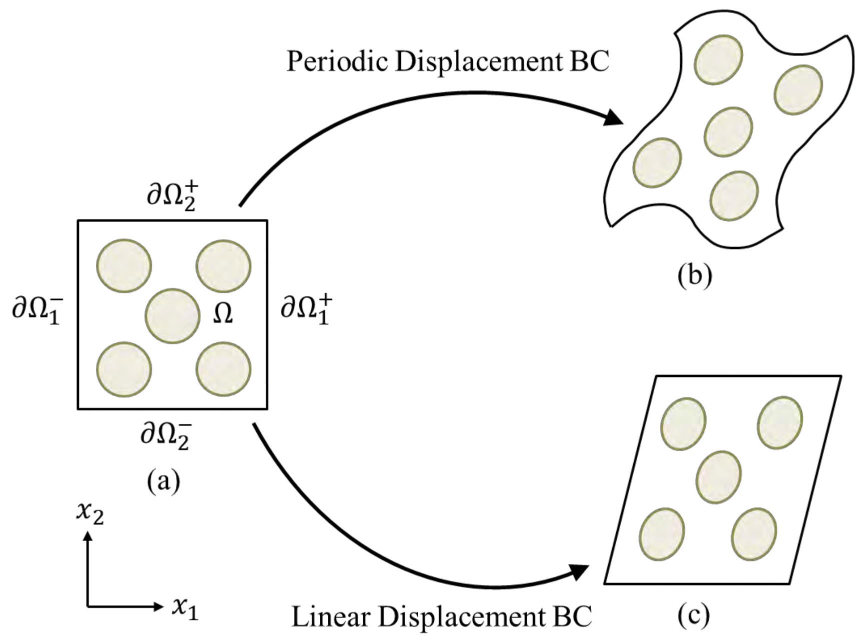


Figure 2. Schematic of a 2D RVE and different displacement boundary conditions. (a) The initial configuration of RVE occupying the domain Ω, where $\partial\Omega_\alpha^+$ and $\partial\Omega_\alpha^-$ denote a pair of opposite external boundaries, and $\alpha = 1, 2$ for this 2D model. (b) The current configuration of RVE subjected to periodic displacement boundary conditions. (c) The current configuration of RVE subjected to linear displacement boundary conditions.

In the RVE domain Ω, the microscopic stress $\boldsymbol{\sigma}(\boldsymbol{X})$ at any point $\boldsymbol{X} \in \Omega$ satisfies the following equilibrium equation:

$$\nabla_{\boldsymbol{X}} \cdot \boldsymbol{\sigma}(\boldsymbol{X}) = \boldsymbol{0}, \tag{1}$$

where $\nabla_{\boldsymbol{X}}$ denotes the gradient operator with respect to the micro-scale. For the microscopic material constituents, constitutive laws can be introduced to define the relationship between the incremental stress $\Delta\boldsymbol{\sigma}(\boldsymbol{X})$ and strain $\Delta\boldsymbol{\varepsilon}(\boldsymbol{X})$ as follows:

$$\Delta\boldsymbol{\sigma}(\boldsymbol{X}) = \boldsymbol{C}(\boldsymbol{X}) \colon \Delta\boldsymbol{\varepsilon}(\boldsymbol{X}), \tag{2}$$

where $\boldsymbol{C}(\boldsymbol{X})$ denotes the microscopic material stiffness tensor at $\boldsymbol{X} \in \Omega$, and the microscopic strain $\boldsymbol{\varepsilon}(\boldsymbol{X})$ can be further expressed as a function of the microscopic displacement field $\boldsymbol{w}(\boldsymbol{X})$ based on the kinematic compatibility condition.

Taking the volume average of the microscopic stress $\boldsymbol{\sigma}$ over the volume Ω, we obtain the homogenized macroscopic stress $\widetilde{\boldsymbol{\sigma}}$

$$\widetilde{\boldsymbol{\sigma}} = \frac{1}{\Omega}\int_{\Omega} \boldsymbol{\sigma}(\boldsymbol{X})\, \boldsymbol{d}\Omega, \tag{3}$$

Similarly, the homogenized macroscopic displacement gradient $\widetilde{\boldsymbol{H}}$ is linked to the microscopic displacement gradient $\boldsymbol{\nabla}_{\boldsymbol{X}}\boldsymbol{w}(\boldsymbol{X})$ as follows

$$\widetilde{\boldsymbol{H}} = \frac{1}{\Omega}\int_{\Omega} \boldsymbol{\nabla}_{\boldsymbol{X}}\boldsymbol{w}(\boldsymbol{X})\, \boldsymbol{d}\Omega, \tag{4}$$

where $\widetilde{\boldsymbol{H}}$ is defined as

$$\widetilde{\boldsymbol{H}} \equiv \nabla_{\widetilde{\boldsymbol{X}}}\widetilde{\boldsymbol{u}} = \widetilde{\boldsymbol{F}} - \boldsymbol{I}, \tag{5}$$

in which $\nabla_{\widetilde{\boldsymbol{X}}}$ denotes the gradient operator with respect to the macro-scale, $\widetilde{\boldsymbol{u}}$ denotes the macroscopic displacement field, $\widetilde{\boldsymbol{F}}$ denotes the macroscopic deformation gradient, and $\boldsymbol{I}$ denotes an identity tensor.

To describe the macroscopic constitutive behavior, we need to compute the homogenized stress $\widetilde{\boldsymbol{\sigma}}$ and displacement gradient $\widetilde{\boldsymbol{H}}$, which requires the calculation of microscopic stress and displacement fields by solving the equilibrium Eq. (1). To this end, properly defined boundary conditions must be imposed on the RVE boundary. Depending on the relationship between the macroscale deformation $\widetilde{\boldsymbol{H}}$ and the microscale displacement field $\boldsymbol{w}$, two types of displacement boundary conditions are commonly adopted.

The *linear displacement boundary condition (LDBC)* can be specified such that the microscale displacement on the RVE boundary is defined as

$$\boldsymbol{w}_{\alpha}(\boldsymbol{X}_{\alpha}) = \widetilde{\boldsymbol{H}} \cdot \boldsymbol{X}_{\alpha}, \tag{6}$$

where $\boldsymbol{X}_{\alpha} \in \partial\Omega_{\alpha}$ denotes an arbitrary microscale material point on the external boundary of RVE.

Alternatively, if the *Periodic Displacement Boundary Condition* (PDBC) is imposed, the microscale displacements $\boldsymbol{w}_{\alpha}^{+}$ and $\boldsymbol{w}_{\alpha}^{-}$ of the external boundary material points $\boldsymbol{X}_{\alpha}^{+}$ and $\boldsymbol{X}_{\alpha}^{-}$ are enforced to satisfy the following condition:

$$\boldsymbol{w}_{\alpha}^{+}(\boldsymbol{X}_{\alpha}^{+}) - \boldsymbol{w}_{\alpha}^{-}(\boldsymbol{X}_{\alpha}^{-}) = \widetilde{\boldsymbol{H}} \cdot (\boldsymbol{X}_{\alpha}^{+} - \boldsymbol{X}_{\alpha}^{-}), \tag{7}$$

where we have decomposed $\partial\Omega_{\alpha}$ into a pair of opposite external surfaces $\partial\Omega_{\alpha}^{+}$ and $\partial\Omega_{\alpha}^{-}$ of the RVE, respectively, i.e., $\partial\Omega_{\alpha} = \partial\Omega_{\alpha}^{+} \cup \partial\Omega_{\alpha}^{-}$, $\boldsymbol{X}_{\alpha}^{+} \in \partial\Omega_{\alpha}^{+}$, and $\boldsymbol{X}_{\alpha}^{-} \in \partial\Omega_{\alpha}^{-}$, as illustrated in Figure 2.

Remark 2.1. It is noteworthy to mention that RVEs with LDBC usually appear to be stiffer than RVEs with PDBC. When the size of RVE is large enough, however, the influence of different boundary conditions on the homogenized material properties becomes negligibly small.

Remark 2.2. Traditionally, enforcing PDBC requires the finite element mesh to be PDBC-matching, which means that the nodal distributions on the RVE's opposite sides must match well with each other. For instance, let us consider two opposite surfaces (e.g., $\partial\Omega_1^+$ and $\partial\Omega_1^-$) that are both perpendicular to the x_1 axis, if we draw a straight line that starts from any FEM node on surface $\partial\Omega_1^+$ and is parallel to the x_1 axis, then a node must be present at the intersection point of this line with surface $\partial\Omega_1^-$. The same condition applies if we draw a line from any node on $\partial\Omega_1^-$ to the surface $\partial\Omega_1^+$.

Remark 2.3. In our implementation in LS-DYNA [2], both PDBC-matching and non-matching meshes can be used to impose the periodic displacement boundary condition. For some cases, the non-matching mesh is attractive because the creation of PDBC-matching meshes could be tedious if very complex material micro-structures exist. Nevertheless, a PDBC-matching mesh is preferred for most cases, as it allows for the use of very efficient and accurate homogenization algorithms.

Remark 2.4. Enforcing PDBC/LDBC to an RVE model used to be a non-trivial task for most commercial FEA software users, as manually specifying multiple constraint conditions is quite tedious and error-prone. To offer a more user-friendly experience, we have designed the keyword *RVE_ANALYSIS_FEM in LS-DYNA in such a way that the users only need to choose PDBC or LDBC, and the software will automatically set up all the constraint conditions for the RVE.

3. Application based on RVE Analysis

RVE analysis becomes increasingly useful for advanced material design and structural analysis. Although there are many different application areas, in this section we will only briefly discuss two RVE-based applications.

3.1. Parameter Identification for Constitutive Laws

Constitutive models often involve many parameters, especially for nonlinear anisotropic materials. Calibration of these model parameters requires many material data. However, measuring these data from a series of physical experiments is quite time-consuming and expensive. These physical experiments, however, can be easily replaced by more cost-saving numerical RVE analysis. By performing homogenization of properly designed material samples, RVE models can accurately predict the macro-scale material behaviors, which can then be adopted to identify the material parameters of the assumed constitutive model [3].

3.2. Multiscale Structural Simulation

To perform multiscale structural simulation, separate finite element (FE) models are needed to represent structures at different length scales. Taking the multiscale car crash simulation as an example, we can construct a macro-scale FE model to represent a car body made of composite materials, whereas a micro-scale FE model (i.e., RVE model) is built to represent the material microstructures. At each integration point of the macro-scale FE model, there is an associated FE-based RVE model, so this multiscale structural simulation method is often referred to as the ‘FE^2 method’. During a concurrent multiscale simulation, the stress/deformation state of the macro-scale FE model serves as the boundary condition for the RVE model, and the micro-scale RVE simulation yields the macroscopic material behaviors for the macro-scale FE model. This two-way macro-micro coupling occurs at each time step, and thus there is no need to specify any constitutive laws at the macroscopic level.

This concurrent simulation offers high-fidelity multiscale structural simulation, but the resulting computational costs can be very high for large-scale engineering problems. Therefore, a reduced-order multiscale model that achieves both high accuracy and efficiency is desired. To this end, we are currently developing a mechanistic machine learning-based multiscale simulation approach for composite structures. In this approach, we employ material data from high-fidelity RVE simulations to perform offline training of machine learning models named Deep Material Network (DMN) [4]. Replacing high-fidelity RVE with DMN in a concurrent multiscale structural simulation can offer a computational speed orders-of-magnitude faster than the conventional FE^2 method. Currently, this DMN-based multiscale simulation method is under development in LS-DYNA for injection-molded short-fiber-reinforced composite structures [5].

4. Workflow for RVE Analysis in LS-DYNA

4.1 RVE Analysis Procedure

To prepare the input deck for RVE analysis in LS-DYNA, the following steps and associated keywords should be considered:

(1) RVE Construction *(*SECTION_xxx, *PART, *ELEMENT_xxx, *NODE):*

a. define the microstructure geometry
b. generate finite element mesh

(2) Microscopic Material Model *(*MAT_xxx):*

a. define the constitutive model for each material constituent (*Eq. (2)*)
b. provide the corresponding material constants

(3) Boundary Condition *(*RVE_ANALYSIS_FEM, *DEFINE_CURVE, *CONTROL_TERMINATION):*

a. choose LDBC (*Eq. (6)*) or PDBC (*Eq. (7)*)

b. define the loading history with displacement gradient $\widetilde{\boldsymbol{H}}$ (*Eq. (5)*)

(4) Output Control *(*DATABASE_RVE, *DATABASE_BINARY_D3PLOT):*

a. define the output parameters for the d3plot file

b. define the output frequency for the rveout file

(5) Implicit Solver *(*CONTROL_IMPLICIT_GENERAL, *CONTROL_IMPLICIT_AUTO, *CONTROL_IMPLICIT_SOLUTION, *CONTROL_IMPLICIT_SOLVER, etc.):*

a. choose a direct or iterative solver

b. define the implicit solver parameters

After the input deck is prepared, the RVE analysis job can be submitted in the same way as other standard SMP/MPP LS-DYNA simulations.

When the simulation is finished, 'd3plot' files will be generated, which contain the microscopic stress/deformation fields of the RVE, and these data can be visualized directly in the LS-PrePost software. Meanwhile, an 'rveout' file containing time histories of homogenized macroscopic stress/deformation fields is also created, and this file can be opened by any text editors.

Remark 4.1. While *RVE_ANALYSIS_FEM and *DATABASE_RVE are the most important keywords for the RVE model definition, a successful RVE analysis also requires the proper definition of other keywords (including keywords that are not listed in this article) to ensure the efficiency and accuracy of FEM simulation.

Remark 4.2. From the numerical point of view, a good mesh quality is required to obtain accurate numerical results. As in any finite element analysis, the finite element size should be small enough to ensure that numerically converged results are obtained.

Remark 4.3. From the micromechanics point of view, the size of the RVE should be large enough to ensure that accurate homogenization results are obtained.

4.2 RVE Example

As an example, let us consider a 3D RVE model for spherical particle-reinforced composites. Firstly, we follow the procedure described in Section 4.1 to prepare the input deck for RVE analysis.

In Step (1), we construct the RVE model based on the particle geometry and volume fraction. Figure 3 shows the PDBC-matching finite element mesh consisted of particle phase and matrix as two parts

(*PART) perfectly bonded together. The mesh information (*ELEMENT_SOLID, *NODE) is stored in the file 'mesh3d_particle_composite.k'.

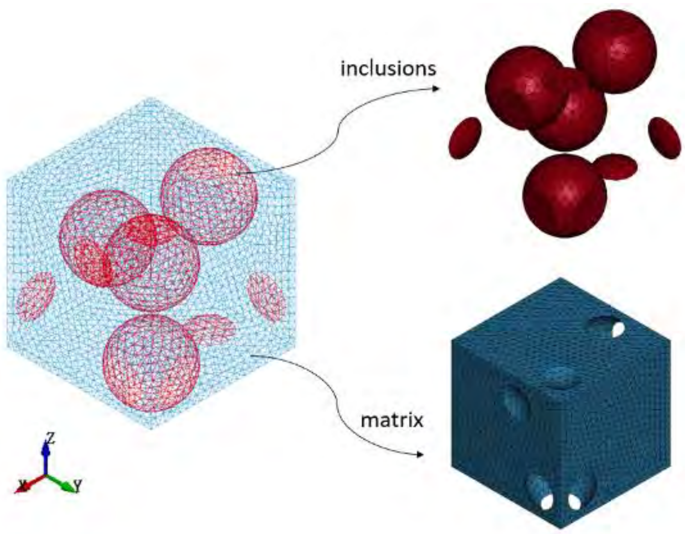


Figure 3. The RVE model for spherical particle-reinforced composites, for which the10-node tetrahedron solid finite elements are employed

In Step (2), we choose a nonlinear hyperelastic constitutive model with the keyword *MAT_MOONEY-RIVLIN_RUBBER and assign different material constants for the particle phase and the matrix phase, respectively, while different constitutive models can be assigned to each microscopic constituent in general applications.

In Step (3), we define the keyword *RVE_ANALYSIS_FEM as follows:

```
*RVE_ANALYSIS_FEM
mesh3d_particle_composite.k
$---+----1----+----2----+----3----+----4----+----5----+----6-
$     inpt      oupt      lcid      idof        bc    imatch
         0         1         1         3         0         1
$      H11       H22       H33       H12       H23       H13
       0.5
```

where the RVE mesh file name 'mesh3d_particle_composite.k' is given; inpt=0 is used to indicate that the RVE boundary conditions are imposed automatically by LS-DYNA; oupt=1 means that the RVE homogenization results will be written to the 'rveout' file; idof=3 is used for this 3D model; bc=0 means that PDBC will be imposed; imatch=1 is defined because a PDBC-matching mesh is used; and for the macroscopic displacement gradient $\widetilde{\boldsymbol{H}}$, its 11 component is defined to be 0.5, while all the other components of $\widetilde{\boldsymbol{H}}$ are set free by simply leaving the entries in the input card blank. In addition, lcid=1 in *RVE_ANALYSIS_FEM indicates that the RVE boundary condition associated with the imposed $\widetilde{\boldsymbol{H}}$ will be applied incrementally based on the load curve with lcid=1 defined in *DEFINE_CURVE:

```
*DEFINE_CURVE
1
0.0,0.0
1.0,1.0
```

In Step (4), we define the output frequency for d3plot and rveout files as follows:

```
*DATABASE_BINARY_D3PLOT
$  dt/cycl
       0.1

*DATABASE_RVE
$        dt
       0.1
```

In Step (5), we choose to use the direct implicit solver for this example.

```
*CONTROL_TERMINATION
$   endtim     endcyc     dtmin
       1.0          0       0.0
*CONTROL_ACCURACY
$      osu        inn
         1          1
*CONTROL_IMPLICIT_GENERAL
         1, 0.1,,,1
*CONTROL_IMPLICIT_SOLUTION
  , , ,1e-6, 1e-8, , ,1e-30
*CONTROL_IMPLICIT_SOLVER
                   6,0
```

An iterative implicit solver can be adopted to achieve better efficiency than the direct solver, but sometimes it may require some trial-and-error to find the optimal solver parameters.

Except for the mesh information (*ELEMENT_SOLID, *NODE) stored in 'mesh3d_particle_composite.k', we will store all the other keywords in a separate input file named 'input_rve3d.k', and then we can use the following command to run MPP LS-DYNA simulation:

```
mpirun -np 10 lsdyna_mpp_dp i= input_rve3d.k
```

where we have assumed that the executable file is named 'lsdyna_mpp_dp'.

After the finite element simulation is finished, we can open the d3plot file in LS-PrePost to check the microscopic stress and deformation, as shown in Figure 4.

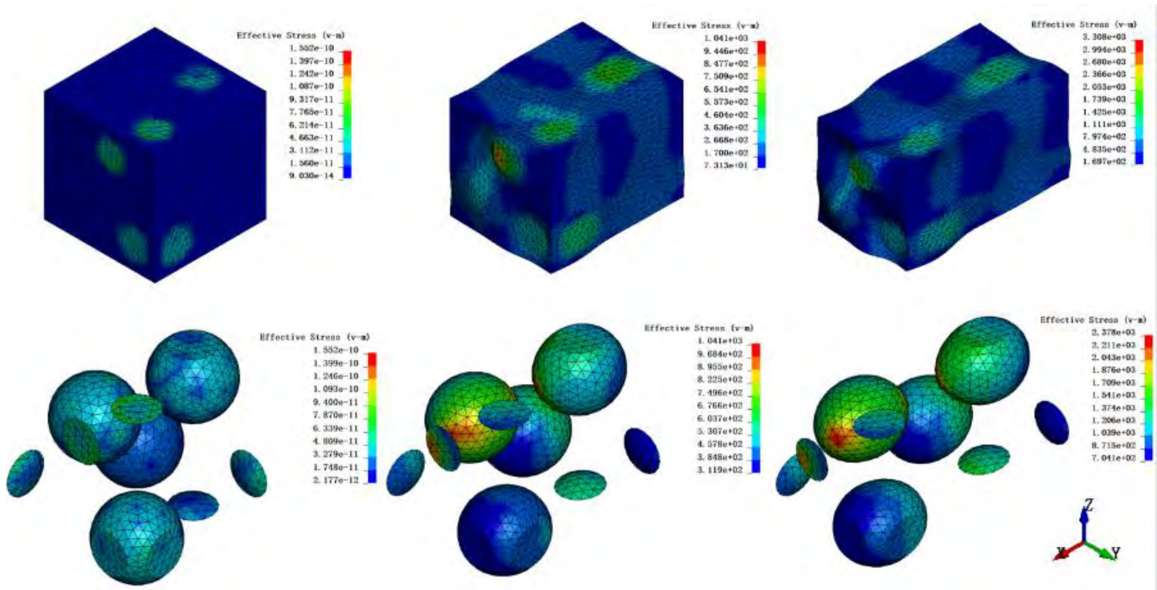


Figure 4. RVE simulation results for particle-reinforced composites. The distribution of von Mises stress in the composites and the particle phase are plotted at different loading steps.

In addition, the homogenized macroscopic stress/strain results are available in the rveout file. For instance, the deformation gradient, the Green strain, and the 1st Piola-Kirchhoff (PK1) stress at the last output step is given in rveout as follows:

```
deformation gradient F, Green strain E, PK1 stress P

 F11=  0.1500000000E+01   F22=  0.8164978140E+00   F33=  0.8219332698E+00   F12= -0.9944214154E-03   F23= -0.2224868009E-02   F31= -0.1304440114E-02
 E11=  0.6250013452E+00   E22= -0.1666626904E+00   E33= -0.1622095242E+00   E12= -0.1150336414E-02   E23= -0.1821997870E-02   E31= -0.1513305221E-02
 P11=  0.4641744224E+03   P22=  0.0000000000E+00   P33=  0.0000000000E+00   P12=  0.0000000000E+00   P23=  0.0000000000E+00   P31=  0.0000000000E+00
```

Remark 4.4. The complete input deck for this example can be downloaded from the website: https://www.lstc-cmmg.org/ex-rve

5. Conclusion

An RVE module has been developed in the multiphysics simulation software LS-DYNA. Given the material microstructural information (geometry and constitutive properties for base materials), this new feature enables LS-DYNA to perform high-fidelity RVE analysis in an efficiency manner, as it includes the following functions:

(1) Creation of periodic displacement boundary conditions (or linear displacement boundary conditions) automatically;

(2) Nonlinear implicit finite element analysis of RVE under any loading conditions;

(3) Calculation of the detailed distribution and evolution of microscopic stress/strain fields in the RVE;

(4) Homogenization of the RVE's microscopic material responses to yield the macroscopic effective material responses.

In summary, this function allows virtual testing of numerically re-constructed material samples, which is of great importance to design and analysis of advanced materials such as fiber-reinforced composites, particulate composites, laminar composites, polycrystalline aggregates, single-phase or multiphase porous media, etc., as well as structures made of such composite materials.

Acknowledgments

The authors would like to acknowledge Dr. Zeliang Liu for the helpful discussions and contributions on the LS-DYNA RVE analysis when he was working at Livermore Software Technology.

6. Reference

[1] Nemat-Nasser, S., Hori, M. (2013). Micromechanics: overall properties of heterogeneous materials. Elsevier.

[2] LS-DYNA Keyword User's Manual, Livermore Software Technology, ANSYS.

[3] Terada, K., Kato, J., Hirayama, N., Inugai, T., Yamamoto, K. (2013). A method of two-scale analysis with micro-macro decoupling scheme: application to hyperelastic composite materials. Computational Mechanics, 52(5), 1199-1219.

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

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

FIVE REASONS WHY YOU MUST VISIT FORD AT THE CHICAGO AUTO SHOW

JUL 14, 2021 | DEARBORN

- Ford to showcase latest vehicles, including Bronco, Bronco Sport and Mustang Mach-E at the Chicago Auto Show; visitors can climb hills with Ford Bronco and experience what nearly instantaneous torque feels like in Mustang Mach-E through unique outdoor drive events
- All-electric 2022 Ford F-150 Lightning is the hero of a new Built to Electrify experience highlighting how easy it actually is for customers to switch to electric – along with the new features and capabilities that are possible only through electrification
- All-new 2022 Ford Maverick compact pickup makes its public debut, while a full-size custom Rocket League F-150 pickup brings the multiplayer video game version to life

DEARBORN, Mich., July 14, 2021 – Ford is arriving to Chicago Auto Show in full force with a fresh new lineup of must-see vehicles, including an all-electric truck, an all-new compact pickup, and iconic nameplates in Bronco and Mustang Mach-E. This week's Chicago Auto Show, the first major American auto show in nearly 18 months, sees a packed display floor with Ford showing off several exciting new vehicles and thrilling hands-on experiences.

"New vehicle customers are looking for something more than parked cars before deciding what to purchase," said Raj Register, Ford head of global brand experiences. "We've created hand-on experiences that educate and let consumers discover what it's like to ride off-road in Bronco and Bronco Sport and feel the all-electric performance of Mustang Mach-E. We'll also have F-150 Lightning so shoppers can see and learn about new features like the Mega Power Frunk and available Ford Intelligent Backup Power."

Five reasons why you can't miss Ford at the 2021 Chicago Auto Show are:

1. F-150 Lightning makes an electrifying debut

The all-electric 2022 F-150 Lightning, revealed in May, is making its public introduction this week. Targeted to have the most torque ever in a F-1501, the all-new F-150 Lightning (available Spring 2022) will ride high into Chicago with impressive technology, Built Ford Tough capability and new features never seen before on an F-150, like a 14-cubic-foot-plus mega power frunk and available Ford Intelligent Back-Up Power, providing power for up to three days in an outage with the extended range battery2.

F-150 Lightning will be located at the Ford display in the west hall of McCormick Place.

2. Meet Maverick, Ford's all-new compact pickup

2022 Ford Maverick (available Fall 2021) also makes its public debut – the first standard full-hybrid pickup in America with a targeted EPA-estimated city fuel economy rating of 40 mpg3, all starting at an MSRP of $19,9954. The all-new Maverick reintroduces the compact pickup truck to the masses, bringing smart technology and flexible cargo solutions for a new segment of customers. This truck doesn't compromise one thing in terms of Built Ford Tough DNA, offering 1,500 pounds of standard payload capacity5 and up to 4,000 pounds of max towing capacity6 – enough for an average 21-foot boat.

Maverick will be on display in the west hall of McCormick Place as well.

3. Treat your wild side with the Built Wild Bronco mountain experience

If you get tired of looking at cars and want to take a ride in one, the Built Wild Bronco mountain experience is calling you.

Making its world debut is a 30,000-square-foot Built Wild experience that invites the public to come out and learn about the modularity, connectivity and hundreds of accessory offerings for the all-new Bronco family of vehicles, including Bronco two-door, four-door and Bronco Sport. The sixth-generation cult classic returns after more than 25 years and here you can experience it where it was meant to be experienced – outdoors.

Visitors can take on a 38-degree steep Bronco mountain with professional drivers. You'll experience the thrill of climbing while learning more about Bronco tech like G.O.A.T. Modes (Goes Over Any Type of Terrain) and feature offerings that make off-roading even more exciting and intelligent, like removable/stowable doors and waterproof interiors with floor drains for easy cleaning.

You'll find the Built Wild Bronco mountain experience right outside McCormick Place.

4. Charge up your motors for the Built to Electrify experience

Also making its world debut in an 11,000-square-foot outdoor display is Built to Electrify, where visitors can experience the full-hybrid Maverick plus the range of zero-emission Ford vehicles for retail and commercial customers – F-150 Lightning, Mustang Mach-E, and 2022 E-Transit (available late 2021).

F-150 Lightning will showcase new features and capabilities that are possible only through electrification. And for the very first time, consumers can get rides in the all-electric Mustang Mach-E, an icon recently crowned winner of Car and Driver's 2021 Electric Vehicle of the Year award. This pony goes 0-60 mph in 4.8 seconds with the extended range battery and eAWD7, and it has an EPA-estimated range of 270 miles8. Don't miss the frunk in both F-150 Lightning and Mustang Mach-E, which comes complete with a drain, so it does double duty as a cooler at tailgates.

The Built to Electrify display is located right outside McCormick Place.

5. Visit a video game truck brought to life

F-150 Rocket League Edition concept is making its real-life debut at the 2021 Chicago Auto Show, the first real-life version of any Rocket League vehicle.

The F-150 design team collaborated with designers at Rocket League developer Psyonix to ensure the in-game vehicle maintains the real-world styling cues and Built Ford Tough DNA of the iconic pickup. This includes the signature C-clamp headlight design, rounded wheel arches, drop-down windows and "F-150" stamped in the tailgate. One-of-a-kind features will make kids and adults alike smile with abandon, like thunder flairs, 37-inch tires, taillamp rocket vents, truck bed rocket booster and a stance wider than F-150 Raptor.

F-150 Rocket League Edition can be found center stage of the Ford display in the west hall of McCormick Place.

Media day for the Chicago Auto Show, the nation's largest, longest-running auto show, is Wednesday. Public days are Thursday through Monday. To learn more about Ford vehicles, visit ford.com.

1. Based on manufacturer testing using computer engineering simulations. Calculated via peak performance of the electric motor(s) at peak battery power. Your results may vary.

2. When home is properly equipped and home transfer switch disconnects home from the grid. Based on 30 kWh use per day using the F-150 Lightning with the extended-range battery. Your results may vary depending on energy usage. Rationing power assumes limiting the number of devices and turning the truck off when not needed.
3. Actual mileage will vary. Final EPA-estimated ratings available later in the 2021 calendar year.
4. MSRP for base vehicle. Excludes destination/delivery fee plus government fees and taxes, any finance charges, any dealer processing charge, any electronic filing charge, and any emission testing charge. Optional equipment not included.
5. Max payload varies and is based on accessories and vehicle configuration. See label on door jamb for carrying capacity of a specific vehicle.
6. Available 4K towing with available 2.0L EcoBoost engine and 4K Tow Package. Max towing varies based on cargo, vehicle configuration, accessories and number of passengers. EPA-estimated fuel economy, payload and towing are independent attributes and may not be achieved simultaneously.
7. Ford test data based on typical industry methodology using 1-ft rollout. Your results may vary.
8. Based on full charge. Actual range varies with conditions such as external environment, vehicle use, vehicle maintenance, lithium ion battery age and state of health.

About Ford Motor Company

Ford Motor Company (NYSE: F) is a global company based in Dearborn, Michigan, that is committed to helping build a better world, where every person is free to move and pursue their dreams. The company's Ford+ plan for growth and value creation combines existing strengths, new capabilities and always-on relationships with customers to enrich experiences for and deepen the loyalty of those customers. Ford designs, manufactures, markets and services a full line of connected, increasingly electrified passenger and commercial vehicles: Ford trucks, utility vehicles, vans and cars, and Lincoln luxury vehicles. The company is pursuing leadership positions in electrification, connected vehicle services and mobility solutions, including self-driving technology, and provides financial services through Ford Motor Credit Company. Ford employs about 186,000 people worldwide. More information about the company, its products and Ford Motor Credit Company is available at corporate.ford.com.

Read from website

ANSYS LS-DYNA

https://www.ansys.com/products/structures/ansys-ls-dyna

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

Social Media

FACEBOOK

BETA CAE Systems CADFEM
ESI Group

TWITTER

BETA CAE Systems ESI Group
ETA CADFEM

LINKEDIN

BETA CAE Systems CADFEM
DYNAmore Nordic ETA
ESI Group

YOUTUBE

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

GOOGLE+

BETA CAE Systems