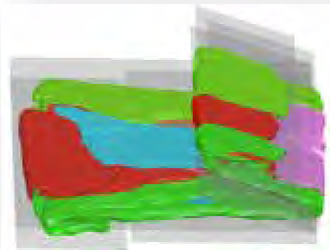
fea INFORMATION
Engineering Solutions

Volume 6, Issue 08, August 2017

www.feainformation.com www.feaiej.com www.feapublications.com

ETA Inventium



BETA CAE SYSTEMS

ESI Group

Rescale Blog

LSTC
Recent Developments
In LS-DYNA
(Editor: Yanhua Zhao)

Conversion between FLD and Stress Triaxial Limit Curve
X. Zhu, L. Zhang, Y. Xiao

A non-orthogonal material model of woven composites in the preforming process -
W. Zhang, H. Ren, B. Liang, D. Zeng, X.Su, J. Dahl, M. Mirdamadi, Q. Zhao, J. Cao

LSTC Livermore Software Technology Corp.
DFK
DYNA MORE

FEA Information Inc.

A publishing company founded April 2000 – published monthly since October 2000.
The publication's focus is engineering technical solutions/information.
FEA Information Inc. publishes:

FEA Information Engineering Solutions

www.feapublications.com
Contact: mv@feainformation.com

FEA Information Engineering Journal

www.feaiej.com
Contact: yanhua@feainformation.com

FEA Information China Engineering Solutions

Simplified and Traditional Chinese
To receive this publication contact yanhua@feainformation.com

FEA Information
Platinum Participants

logo courtesy - Lancemore

FEA Information Platinum Participants

logo courtesy - Lancemore

BETA SIMULATION SOLUTIONS

MSC Materials Sciences Corporation

eta

esi get it right®

JSOL

恒士达科技 Hengstar Tech.

Oasys The Software House of Arup

rescale

Finite Element Analysis Predictive Engineering

FEA Information News Sections

Automotive News **Editor - Dilip Bhalsod**

Aerospace News **Editor - Marnie Azadian**

China FEA News –Events – Participants **Editor – Yahua Zhao**

Solutions

Participants	**Distribution & Consulting**	**Cloud/On Demand/Subscription**
ATD – Barrier - THUMS		

Classes – Social Media - Conferences – Events

LSTC Information & Apps

Announcements

Software/version new releases from:

- BETA CAE Systems
- ETA
- ESI

15th LS-DYNA Conf .Abstract Time is getting closer
Deadline: November 15th, 2017

Please add to our new section on educational institutions that have had participants publishing papers on LS-DYNA

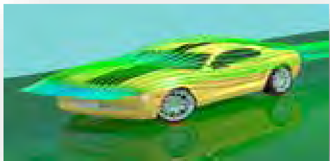
Two new feature papers from New Features - Editor of this area is Yanhua Zhao yanhua@feainformation.com - if you need a pdf copy contact Yanhua

Conversion between FLD and Stress Triaxial Limit Curve
X. Zhu, L. Zhang, Y. Xiao

A non-orthogonal material model of woven composites in the preforming process
W. Zhang, H. Ren, B. Liang, D. Zeng, X.Su, J. Dahl, M. Mirdamadi, Q. Zhao, J. Cao

Any suggestions please contact Marsha Victory mv@feainformation.com

15th International LS-DYNA® Users Conference & Users Meeting



June 10-12, 2018

Edward Hotel &
Convention Center
Dearborn. MI. USA

Welcome:

The conference will host a forum for engineers, professors, students, consultants, industry leaders, and interested parties to exchange their ideas, and listen to the latest in industry and academic presentations..

The presenter (1) One Presenter of the accepted paper will receive a complimentary (no fee) conference registration, when they register using the "LSTC Conference" group registration code at the Edward Hotel.

Conference Dates:

Sunday	06/10/2018	Registration	Exhibition Area	Reception
Monday	06/11/2018	Registration	Exhibition Area	Banquet
Tuesday	06/12/2018	Registration	Exhibition Area	Closing
Wednesday/Thursday	06/13-14/2018	Training Classes		

Information:

Abstracts & papers papers@lstc.com
Participation, Registration conference@lstc.com

Abstract Submission on line:
Deadline: November 15th, 2017
On line being processed by DYNAmore GmbH
www.dynamore.de/paper2018

Paper Submission: Deadline: February 14, 2018 FIRM

Notification and templates will be provided by DYNAmore
For any questions please write papers@lstc.com

Abstracts: www.dynamore.de/paper2018
Registration/Classes: www.ls-dynaconferences.com

Conference Call For Papers

- Acoustics
- Aerospace
- Automotive
  - Crashworthiness
  - Durability
  - NVH
- Ballistics and Penetration
- Biomechanics
- Civil Engineering
- Electromagnetics
- Fluid Dynamics
  - Compressible
  - ALE (Lagrangian, Eulerian)
  - CESE
  - Incompressible
- Granular Flow
- Heat Transfer
- HPC & Cloud Services
- Impact and Drop Testing
- Manufacturing Processes
- Material Parameter Identification
- Metal Forming
- Modeling Techniques
- Nuclear Power
- Occupant Safety
- Optimization
- Particle Method
  - Airbag Particle Method
  - Discrete Elements
  - Element Free Galerkin
  - Peradynamics
  - Smooth Particle Hydrodynamics
  - Smooth Particle Galerkin
- PrePost Processing
- Seismic Engineering
- Ship Building

BETA CAE Systems releases v18.0.0 of its software suite

(Excerpts) complete Article is at: www.beta-cae.com/news/20170728_announcement_suite_v18.0.0.htm

BETA CAE Systems, is proud for bringing simulation to a next level, with the new v18.0.0 of its software suite. Beyond the groundbreaking enhancements to ANSA, EPILYSIS and META, the new product, KOMVOS-SDM Console, is now integrated to the BETA suite.

KOMVOS, the new addition to the legacy BETA Suite, is an innovative Simulation Data Management platform for the interactive browsing, visualization and handling of all data related to CAE analysis, from PDM extractions to simulation runs, key results and reports.

Further beyond, with a plethora of revolutionary tools and groundbreaking solutions, v18.0.0 unquestionably addresses all challenges involved in the contemporary CAE industry, successfully combating all bottlenecks introduced by modeling complexity in any application area, while offers a significant boost to the operations of the CAE modeling process as a whole.

The new version 18.0.0 of ANSA adds value to the multi-disciplinary concept of our pre-processor, introducing a broad range of new features and enhancements to existing ones, reinforcing overall process consistency, accelerating user performance, and providing a considerable increase in productivity.

EPILYSIS, following its initial objective to bridge the gap between pre- and post- processing, comes with extended functionality in diverse areas, while the significant acceleration in the overall process execution is definitely one of its main assets in v18.0.0.

The new version of META extends the capabilities of post-processing with the introduction of pioneering tools and cutting-edge approaches, coupled with multi-purpose solutions for any process on demand, high levels of automation, as well as an impressive performance.

The highlights of our new software solutions are listed below. Do not miss:

The game-changing functionality for meshing and mid surface extraction.

BETA CAE Systems releases v18.0.0 of its software suite

The extended capabilities of META via the implementation of Virtual Reality in post-processing, captivating the perception and cognition of any given FEA workflow from a closer and more realistic perspective.
The abundant developments that took place in the graphics engine and visualization capabilities of META, addressing the need for photo-realistic visualization and post-processing of simulation results for divergent engineering tasks.

The promising potential for Web Collaboration that ANSA and META “Remote Viewer” offers, promoting efficient task management and progressed co-operation between team members and clients.
The most anticipated, dedicated Inverse Forming tool.

Our dedicated solutions on Marine and Offshore design applications, hosted under a brand new menu with accompanied exhaustive functionality.

Introducing Virtual Reality

Extending the capabilities of META, v18.0.0 provides a unique experience in processing FEA results from a closer and more realistic perspective via the implementation of Virtual Reality in post-processing.

Via the support of the HTC VIVE VR headset, v18.0.0 offers a virtual walk-around a real-size simulation model, physical focus on areas of interest and, above all, integration of VR capabilities in any given FEA workflow.

Visualization and Collaboration capabilities

Use your mobile device to take control of ANSA and META on your PC, even from a different time zone. ANSA and META "Remote Viewer" addresses this challenge in v18.0.0, with a port and an IP address as the sole prerequisites.

An impressive variety of graphics improvements in META adds value to the post-processing of results: Rendering Materials, Smooth Light, Spheres, Line Integral Convolution (LIC), Antialiasing, together with significant performance improvements, are some representative examples in this field.

Updates in CAD data translation

The new “translation” library (CT_2017_SP0), which has been incorporated in ANSA v18.0.0, enables the reading and translation of NX11, Creo 4.0 F000, SolidWorks 2017 and Inventor 2017 files. Furthermore, NX11 files can be translated based on the NX-Open toolkit.

BETA CAE Systems releases v18.0.0 of its software suite

Model & Data Management

Since the most important case during the model build up is a comprehensive model organization, BETA CAE Systems has come up with new data types that facilitate the set up for a simulation run.
Specifically, we discretize the CAE data into 5 distinct types: Parts & Connections, Subsystems, Simulation Models, Loadcases and Simulation Runs.

All these data types are efficiently handled via a constantly evolving environment, Model Browser, that offers the ability to the CAE analyst to work either as a single unit on a project or as a member of a team.
These new data types come to fully support Data Management, not only in ANSA, but also in SPDRM implementations.

Process Automation

Two main pillars are the outstanding innovations in this area.

Great Improvements in performance and functionality, such as various core improvements to speed up execution time of functions, like accessing Element Nodes, Collecting entities and improving the overall performance of an average script - leading to the development of tools in script indistinguishable from the native functionality.

New advanced functionality, for example the ability to save and retrieve custom data in the ANSA DB, new compare API, new API functionality, user draw modes and, last but not least, BETA GUI available in stand-alone python/jython module.

Meshing

ANSA v18.0.0 introduces new concepts and tools that radically change the rules of the meshing game.

Automatic mid-surfacing of complex casting parts is supported by a brand new algorithm with significantly improved results. This powerful upgrade, combined with the new Align Manager fixing tool, provides impressive process acceleration.

Feature Manager, on the other hand, hosts all feature-related functionality under one single "hyper-tool", facilitating the workflow process even more.

Furthermore, FE Perimeters introduction and handling has radically altered the meshing workflow concept. The existence of FE perimeters allows the FE mesh to be handled exactly like geometry and this is obvious in already available functions, previously devoted only to geometry. This results in a significant speed up of the whole improvement process (Num +/-, Cut, Join etc.).

New algorithms for the Extrude tool along with the redesign of the user interface make the generation of mapped/sweepable volumes trivial and faster.

Also the new tool Solid Builder offers advanced capabilities to create new elements with a single selection step, saving clicks and time.

BETA CAE Systems releases v18.0.0 of its software suite

Solutions for NVH

Process and Loadcase setup has never been easier with the implementation of Loadcase Assistant, while loadcasing is now streamlined even more with the introduction of new entity types, such as the Assembly & Loadcase Points (A/LC Points).

Moreover, easy and efficient identification of the Root Causes of poor performance for NVH Analysis can take place through a vast variety of tools and features, including Bush Sensitivity, Transfer Path Analysis, Mode Contribution, Energy Map, Grid Participations and System Modes Participations.

New 3D Plots, such as MAC3D, Nyquist and Sand dune provide an even greater means of communication of the results to the analyst, while MAC, AutoMAC and Driving Point Residues (DPR) drastically extend the capabilities of Modal/FRF Correlation Tool even more.

These come along with dedicated features for Powertrain NVH, such as the calculation of Equivalent Radiated Power (ERP) results from Modal Response, as a combination of Eigenmodes, Loads and Panels Definition. Not to be missed handy features, such as the brand new animation bar in META, providing view via Frequency and not via State ID, with scrollable capabilities as well.

Towards the performance acceleration, EPILYSIS comes with impressive advancements in the NVH field: A significant performance boost in SOL103 with AMLS method and extended Contact Simulation with the implementation of Glued Contacts, assisting in this way model flexibility by automatically connecting two bodies with dissimilar meshes.

(excerpts) For complete listing please visit the website

Solutions for Crash & Safety

v18.0.0 comes with several upgrades in the Safety field. A new mode for dummy-seat depenetration using EPILYSIS, for the deformation of the springs that are underneath the seat’s cushion, has been added. A tool for low speed tests is now available. Finally, Marrionette tool offers a pre-simulation set up for LS-DYNA Positioning Loadcase through a wizard.

LinkedIn	www.esi-group.com/linkedin
Facebook	www.esi-group.com/facebook
Twitter	www.esi-group.com/twitter
YouTube	www.esi-group.com/youtube

ESI Releases SYSTUS 2017 Dedicated to the Energy Sector

3 Aug 2017 Paris, France

www.esi-group.com

Software for the Testing of Thermomechanical and Multi-physical Behaviors of Components

Paris, France – August 3, 2017 – ESI Group, leading innovator in Virtual Prototyping software and services for manufacturing industries, announces the latest version of SYSTUS, proven software solution for thermomechanical design and regulatory analyses in the Nuclear sector. ESI SYSTUS 2017 brings significant new capabilities to the software, including an innovative Differential Method (RCCM-MD). The version also brings enhancements to investigate fracture mechanics using the innovative X-FEM method, and addresses topology optimization for many applications, including in the automotive and aeronautic sectors.

SYSTUS benefits from 40 years of R&D in Finite Elements Analysis (FEA). The software was originally developed by Framatome (now AREVA NP) to support the design of their nuclear power plants and to perform regulatory analyses — all virtually. Today, SYSTUS has become an industry-proven solution for an extensive range of regulatory analyses mandated by international standards for nuclear engineering. It specifically addresses the requirements of the ASME code[1], standard issued by the American Society of Mechanical Engineers, and the French standard, published by AFCEN and known as RCC-M (Règles de Conception et de Construction des Matériels Mécaniques des Ilots Nucléaires2 REP3).

SYSTUS 2017 brings new enhancements for conducting regulatory analyses, including new analytic options, and a better legibility of results. It introduces a Differential Method (RCCM-MD) for second category analyses that can help industrial manufacturers and their suppliers overcome major issues related to load changes over time and variations in the directions of principal stresses, as these interfere with the computation of the amplitude of the equivalent stresses Sn and the usage factor U.

In the field of fracture mechanics, SYSTUS allows industrial manufacturers and their suppliers to perform brittle and ductile fracture analysis. Working with AREVA for over 6 years, the ESI SYSTUS team has developed an innovative eXtended Finite Elements method (X-FEM) that enables the analysis of defects without the need to model the cracks explicitly — hence reducing complexity and saving time. With ESI's X-FEM method, the representation of defects is carried out using the Level-Set Method and specific enrichment functions.

ESI Releases SYSTUS 2017 Dedicated to the Energy Sector

SYSTUS 2017 includes enhancements for fracture mechanics with the X-FEM method, including new and simplified commands to improve software ergonomics. The version delivers strong efficiency improvements and offers results very close to those obtained with Finite Element Method (FEM) analysis with meshed cracks. Furthermore, with SYSTUS 2017, the X-FEM method now takes into account plasticity, includes meshing with quadratic elements, and enables the computation of the Energy rate using the Theta method.

In the field of fracture mechanics, SYSTUS allows industrial manufacturers and their suppliers to perform brittle and ductile fracture analysis. Working with AREVA for over 6 years, the ESI SYSTUS team has developed an innovative eXtended Finite Elements method (X-FEM) that enables the analysis of defects without the need to model the cracks explicitly — hence reducing complexity and saving time. With ESI's X-FEM method, the representation of defects is carried out using the Level-Set Method and specific enrichment functions.
SYSTUS 2017 includes enhancements for fracture mechanics with the X-FEM method, including new and simplified commands to improve software ergonomics. The version delivers strong efficiency improvements and offers results very close to those obtained with Finite Element Method (FEM) analysis with meshed cracks. Furthermore, with SYSTUS 2017, the X-FEM method now takes into account plasticity, includes meshing with quadratic elements, and enables the computation of the Energy rate using the Theta method.

1 ASME code : Section III, division 1, article NB 3200
2 RCC-M = Règles de Conception et de Construction des Matériels Mécaniques des Ilots Nucléaires. In English: Design and construction rules for mechanical components of PWR nuclear islands
3 REP = Réacteur à Eau Pressurisée. In English: Pressurized Water Reactor.
See RCC-M : Paragraph B3200

For more information about ESI SYSTUS, please visit: www.esi-group.com/systus
Join ESI's customer portal myESI to get continuously updated product information, tips & tricks, view the online training schedule and access selected software downloads: myesi.esi-group.com

For more ESI news, visit:
www.esi-group.com/press
ESI Group – Media Relations
Delphine Avomo Evouna
+33 1 41 73 58 46

http://blog.rescale.com

Rescale Blog:

Cloud: The Next Disruption in HPC

Gabriel Broner - August 11, 2017

In 1991, I joined Cray and had the opportunity to work on the machines Seymour Cray designed. I was working on the operating system and would often have to work alone on it at night, but the excitement of working on such unique systems kept me going. The Cray 1, XMP, YMP, represented a family of machines where a differentiated architecture and design allowed you to solve problems that you just couldn't solve with a regular computer.

When I joined, Cray was considering building a new type of parallel machines we called MPPs (massively parallel processing). I worked on the design and implementation of the operating system for the Cray T3E, a system with 2048 individual nodes, with standard CPU chips, memory, and a proprietary high-speed interconnect. Ahead of its time, Cray was building what we today call HPC clusters. Besides it being a fantastic engineering project, it was the beginning of a disruption: going from proprietary Cray architectures to clusters of nodes with commodity parts.

I lived through a second disruption around year 2000. I was leading software development at SGI, and we were planning to move from MIPS processors to Intel. At a time when Linux was still considered a hobbyist operating system, we at SGI took a risk by also moving to Linux during the migration, despite SGI's IRIX being touted as a great operating system at the time. Initially customers were quite resistant to the change because Linux was viewed as offering less control than their proprietary OS. Over time, customers warmed up to the idea, other vendors followed, and Linux became the standard operating system for HPC, enabling more applications to be readily available.

As years have passed, HPC has transitioned from unique and proprietary designs, to clusters of many dual-CPU Intel nodes. Vendors' products are now differentiated more by packaging, density, and cooling than the uniqueness of the architecture.

In parallel, cloud computing has gained momentum in the larger IT industry. Intel is now selling more processors to run in the cloud than in company-owned facilities, and cloud is starting to drive innovation and efficiencies at a rate faster than on premises.

High performance computing has evolved on-premise. You buy a computer for a few million dollars, and you are able to run simulations to reduce your innovation time and time to market for your products. The auto manufacturer depicted in Figure 1 represents the new dilemma faced in buying such an in-house HPC system. With the workload this company has, what size system should they buy? If they buy a system that accommodates the peak workload, they may have to spend around $20M, but the system will be only 20% utilized. If they buy a $4M system, the system will be highly utilized, but large jobs cannot be run, and jobs will wait in a queue—potentially for days—before they run, delaying innovation and time to market.

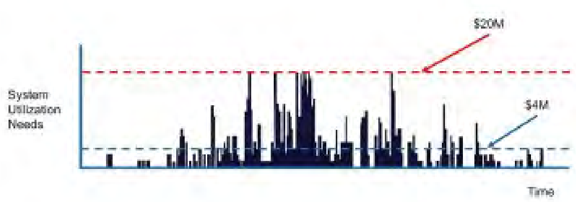


Figure 1 – The Challenge of an auto manufacturer selecting the next HPC system

This auto part manufacturer decided that neither option was acceptable to them, and instead they decided to go with Rescale and run HPC in the cloud. They now pay between $50K and $100K per month for the IT they need when they need it and they have instant access to the perfectly-sized system without waiting, significantly improving design throughput and time to market. With the Rescale cloud platform, they get the best of both worlds—they spend monthly what they would have spent in depreciation of a $4M system, while they get the service of a $20M system.

All customers with high performance computing requirements face the same challenge the auto manufacturer faced. The problem becomes even more pronounced with the new variety of processor architectures that includes CPU, GPU, TPU, KNL, FPGA, and multiple interconnect technologies because you either have to partition your system or forego the latest architectures. In contrast, the cloud does not force you to choose and instead allows you to run each workload on the best possible architecture for your problem type. Applications are readily accessible on the cloud, allowing customers to pay per usage. The availability of hardware and applications allows new customers to take advantage of HPC, which was only available to large enterprises before cloud HPC.

Like the previous disruptions of clusters vs. monolithic systems or Linux vs. proprietary operating systems, cloud changes the status quo, takes us out of our comfort zone, and gives us a sense of lack of control. But the effect of price, the flexibility to dynamically change your system size and choose the best architecture for the job, the availability of applications, the ability to select system cost based on the needs of a particular workload, and the ability to provision and run immediately, will prove very attractive for HPC users. It may be time to start thinking about HPC in the cloud in your organization!

The nature of cloud disruption is unique. It's not all or nothing, and you can dip your toe in the water. If you follow the traditional processes and buy another on-premise system, you will miss the advantages of cloud. Cloud gives us an opportunity to test the benefits of the future without committing to the next multi-million-dollar purchase. If you spend $100K you can start immediately, testing HPC in the cloud, accessing the latest architectures available. If the next HPC system in 3-5 years will be in the cloud or will be a hybrid system, testing it now, learning from it, and iterating will reduce risk and will enable a much smoother transition. So, in addition to thinking about cloud, I encourage you to test the future starting next week!

Gabriel Broner has been in the HPC industry for 25 years. He held roles of operating systems architect at Cray, VP & GM of HPC at SGI/HPE, head of innovation at Ericsson, GM at Microsoft. Gabriel joined Rescale as VP & GM of HPC in July 2017.

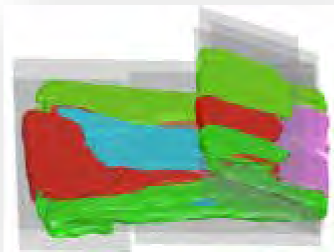
Additional Blogs for August

- **Rescale Partners with Equinix to Enable Fast and Secure Hardware & Data Transfer to the Cloud Big Compute Environment**
  - August 9, 2017
- **Rescale Parters with HPC Systems to Deliver Cloud HPC Platform to Scientific Computing in Japan**
  - August 7, 2017
- **Rescale Announces Southeast Asia Expansion with the Appointment of Zac Leow as Director, Southeast Asia**
  - August 1, 2017

ETA - Inventium 2017 R1 Released

Software Features New DYNAFOLD Feature for Airbag Folding Simulation

www.eta.com



8/9/2017—Engineering Technology Associates, Inc. (ETA) announced the release of Inventium 2017 R1 today. The release includes PreSys, ETA's finite element modeling software, and VPG, mechanical system analysis plugins.

Available immediately through ETA and its worldwide distribution network, substantial updates to the software have been implemented in the release for finite element modeling, meshing, geometry, solver compatibility, Post-Processing and a new 'Geometry Cleanup' function was added to significantly improve mesh quality. Additionally, this release also includes DYNAFOLD, a very exciting new feature for Airbag Folding, in VPG/Safety.

The DYNAFOLD feature is used to simulate the folding process of an airbag, providing various folding methods and folding tools for the user. It allows the user to generate folded airbag models using LS-DYNA solver calculations and it simulates the real folding process very precisely. It can effectively shorten research and development time and reduce cost for the manufacturer.

It provides seven fold types commonly used by the manufacturers. The user can define the fold methods and these fold methods can be combined arbitrarily. The folding methods are thin folds, roll folds, concertina folds, zigzag folds, scrunch folds, compress folds and trunk folds.

In addition, DYNAFOLD supports user-defined folding methods. The user can use five tools, which are provided in the software to create his/her own folding tool. Users can also read in a tool model with the 'Import Tool' function and then define their movements.

Inventium 2017 R1 is the latest release of the Inventium software that includes PreSys and the VPG Modules (Structure, Safety, Drop Test and Fluid Structure Interaction). A core solution for finite element analysis engineers, PreSys is a cost-effective software tool, which also interfaces with popular CAD software products such as CATIA, NX, ProEngineer, Solidworks and AutoCAD. It allows product development engineers and simulation specialists to access design data, quickly create complex finite element models and create insightful simulations.

Building upon its initial release in June of 2010 and subsequent releases in 2012 and 2014 and 2016, PreSys now offers added functionality and broader potential with its 2017 R1 Release. The software tool is now available on the Windows platform for 64-bit machines, directly through ETA and its distributor network. To find out more, please visit www.eta.com.

Inventium 2017 R1 Released

DYNAFORM 5.9.3 Available

Offering Automatic Springback Compensation & New License Control DYNAFORM 5.9.3 offers major improvements in blank/trim line development and Automatic Springback Compensation iterations.

This version combines the interface for blank development and trim line development. Therefore, it is now possible to develop the blank outline and trim line simultaneously. The new Auto SCP feature is used for automatic springback compensation iterations. The program automatically checks the springback amount and calls the solver to make a compensation adjustment for the tooling mesh. The compensated tool then replaces the original tool and multi-stage jobs are rerun until the ideal solution is found.

ACP OpDesign Update

ETA has collaborated with BETA CAE Systems to develop a software tool, which mirrors ETA's patented Accelerated Concept to Product (ACP) Process and will serve as a gateway for optimization and design.

The product was demonstrated at the 7th Annual BETA CAE Before Reality Conference in Thessaloniki, Greece in June. The live demo showed the ACP Opdesign modules ACP Concept and ACP 3G (Low Fidelity).

The product was demonstrated at the 7th Annual BETA CAE Before Reality Conference in Thessaloniki, Greece in June. The live demo showed the ACP Opdesign modules ACP Concept and ACP 3G (Low Fidelity).

If you are interested, please contac vwalvekar@eta.com

LS-DYNA: Implicit Quick-Start for Explicit Simulation Engineers

A series of informal articles about one engineer's usage of LS-DYNA to solve a variety of non-crash simulation problems.

By: George Laird, PhD, PE
Principal Mechanical Engineer, Predictive Engineering

Most FEA work in the world is dominated by linear elastic stress and vibration analysis (implicit). The complexity varies tremendously within this realm and can be every bit as challenging as a highly nonlinear transient model (explicit). In the linear world, stress values are very sensitive to small changes in strain, and often take on even greater importance, since their values are used to verify the design margin of a structure or its fatigue life. Since the mission statements and analysis requirements between implicit and explicit analyses are different, one has to shift gears to move from one to the other. It is the focus of this short note to point out how a journeyman explicit simulation engineer can quickly and efficiently create implicit analyses from linear to nonlinear.

Where do I really start?

Not with this note. If you are seriously interested in implicit one should immediately start by reading Appendix P in the latest version of the Keyword Manual Vol. 1 and then proceed to DYNASupport.com and read the DYNAmore paper titled: "Some Guidelines for Implicit Analysis Using LS-DYNA." Then, for examples and real-world scaling data on using implicit
LS-DYNA, see Laird and Pathy "A Roadmap to Linear and Nonlinear Implicit Analysis with LS-DYNA" which can be found at DYNALook.com. However, if you are looking for a quick overview, please keep reading and we'll try to boil it down as simply as possible.

High-Level Concepts

First off, we recommend using the latest release of MPP double-precision and sometimes, since implicit is changing so fast, the latest development version. It is on the cutting edge but today's implicit cannot be compared with yesterday's version. It truly changes that fast. Why MPP and double-precision? MPP scales better than SMP while double-precision is a necessity for clean convergence (i.e., nonlinear implicit). If you are hesitant about taking the plunge to MPP and are using Windows, we have a helpful readme note at www.predictiveengineering.com.

Core Basics for Static Implicit

It just boils down to a few *CONTROL cards and carefully picking your element formulations. The following table provides our recommendations in alphabetical order that will ensure a robust nonlinear (material and geometric (large deformation and contact)) implicit stress analysis. If your focus is on linear implicit analysis, take a look at our paper: "A Roadmap to Linear…" since it covers all the tips and tricks and examples from this paper can be downloaded at DYNAExamples.com.

The alphabetical overview of what you need for nonlinear implicit analysis.

Contact

There really is only two recommended contact formulations for implicit. The advantage of _MORTAR contact is that it is very robust and the general formulation will also handle beam-on-beam contact. The only option that is recommended is that to set IGNORE=1 to handle small penetrations. Mortar is not restricted to implicit and works well in explicit, albeit a bit slow. As for _TIED contact, we are really trying to narrow down the field to the most basic and most universal. For implicit, it is wise to avoid the use of springs or the penalty method whenever possible. Additionally, sometimes things don't line up well and _OFFSET is needed. To address both of these requirements, one can use _CONSTRAINED_OFFSET with _NODES for tying solids to whatever and _SHELL_EDGE for tying beams and shells to whatever. Since rigid bodies are sometimes in the mix, if the option IPBACK=1 is added, the _TIED formulation is switched to a penalty method if a rigid body is detected. There is no harm that we know of using _OFFSET for a non-offset condition and likewise keeping the IPBACK=1 option turned-on even if no rigid bodies are present. The concept is to keep it simple and robust with minimum tweaking of basic Keyword Commands.

*CONTACT Keyword	Options
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR	IGNORE=1
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR	IGNORE=1
*CONTACT_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET	IPBACK=1
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET	IPBACK=1

Control

This is my favorite section for setting up an implicit analysis since there are so many little pitfalls in the settings. In some cases, the cards have a bit of overlap with explicit settings. For example, the _ACCURACY card also includes the general explicit recommendation to set INN=4, so we carry this over to the implicit simulation for good measure. The setting IACC=1 does several little things and one should just read the manual (RTM). If one is using _SINGLE_SURFACE contact with shells, then it makes sense to also set SSTHK=1 (*CONTROL_CONTACT) to capture the true shell thicknesses

One may notice that I don't include *CONTROL_ENERGY in the setup. Since implicit isn't worried about hourglass and if contact is used, SLNTEN is automatically calculated; hence, one less Keyword card to mess with.

The _IMPLICIT_AUTO setting is pretty much self-explanatory while the _IMPLICT_DYNAMICS setting is guaranteed to raise questions since we have stated that the overall *CONTROL settings are for a static implicit analysis and now we throw in_IMPLICIT_DYNAMICS? How can one resolve this obvious dilemma of turning a static analysis into a dynamic analysis? Well, it requires a bit of an explanation since this card represents one of the most important static implicit settings to obtain quick convergence in contact problems and often in large deformation problems (e.g., buckling). It has been our experience that at the start of a complex implicit analysis, there are several parts of the structure that require contact to be restrained (e.g., bolt preload). During initiation of the analysis, these structures are free to move and will generate warning messages from the solver since it is struggling to find convergence. A common work around by many nonlinear implicit codes is to apply a bit of numerical glue between the contacting surfaces during the first few iterations to stabilize the model (i.e., restrain floating parts). As the solution progresses, the stickiness between the contacts is reduced. This technique has its pros and cons but mostly cons since if friction is used, it can cause contacting faces to lock-up unexpectedly and change the load path. A better way is to avoid the use of such techniques and let the model behave as it would in the real world; that is, everything is dynamic in the real world. When a load is applied to a structure, no matter how slow, it is still dynamic. One will note that the option settings for _IMPLICIT_DYNAMICS is non-standard. By setting GAMMA=0.60 and BETA=0.38, the dynamic response is heavily damped. This hinders the structure's dynamic response during load application and leads to a more static response of the structure as it moves and deforms under load. One concern in using this technique is the introduction of dynamic or inertia effects into a simulation that is designed to be static or free of dynamic effects. As mentioned, in the real world, all load applications are dynamic albeit some are faster than others. The idea of a slow, damped dynamic load application can be assessed by plotting the kinetic energy (KE) against the internal energy (IE). What we strive for is to maintain the KE below the IE during the simulation and should gradually decrease toward the end of the simulation. At the very end, it should be only a few percent of the IE. In this manner, we have a quantitative assurance that the simulation is behaving as a static analysis. In our experience, this approach leads to faster convergence for not only initially unstable contact analyses but also for material failure where element erosion can make the structure temporarily unstable, progressive failure of composites and buckling of structures. In short, we use _IMPLCIT_DYNAMICS (with GAMMA=0.60 and BETA=0.38) in every static, nonlinear implicit analysis model. A last little comment on using _IMPLICIT_DYNAMICS, your simulation is now aware of time and one can use BIRTH_DEATH options and if you want to and also just bury the dynamic behavior toward the end of the simulation (see the Keyword Card).

After that rather lengthy explanation of one Keyword card, everything else is rather mundane. For example, we have _IMPLICIT_GENERAL with a DTO=0.01 since we like to end our static implicit analyses at ENDTIME=1.0. The idea is to start easy and then let _AUTO ramp up the load as needed. As for the solver, we rarely ever touch the settings on _IMPLICIT_SOLUTION except to tighten up the convergence tolerance to ABSTOL=1e-20 and have it print out diagnostics with NLPRINT=2 (we'll talk a bit about convergence failure later on under Troubleshooting). Since it is implicit and we are using fully integrated element formulations, we want to extrapolate the stresses (Superconvergent Patch Recovery) from the integration points out to the nodes. This is done using _OUTPUT with SOLSIG=2 or =3. Unfortunately this is only done for solid elements at this time. We are hoping that one day LSTC will add the same capability for shell elements. Currently, if one wants to view extrapolated shell stresses in LSPP, it requires the use of extrapolate. The last command automatically switches the element formulation for triangular elements from ELFORM=16 to ELFORM=17.

*CONTROL Keyword	Options
*CONTROL_ACCURACY	INN=4, IACC=1
*CONTROL_CONTACT	SSTHK=1
*CONTROL_IMPLICIT_AUTO	IAUTO=1
*CONTROL_IMPLICIT_DYNAMICS	IMASS=1, GAMMA=0.60 & BETA=0.38
*CONTROL_IMPLICIT_GENERAL	IMFLAG=1, DTO=0.01
*CONTROL_IMPLICIT_SOLUTION	ABSTOL=1E-20, NLPRINT=2
*CONTROL_OUTPUT	SOLSIG=2 or =3
*CONTROL_SHELL	ESORT=2

Database

There is only one specialized command for implicit and it really isn't specialized to implicit. It has to do with dumping out all the stresses from the full integration point set. For example, with MAXINT=-3 setting one gets all integration point stresses at the outermost NIP layers and at the midplane (if the recommended NIP=5 is used). Likewise, if NINTSLD=8, all integration point data is written out whether it is a brick or a tetrahedral. As for beams, that is a user choice as to what number of integration points to kick out using BEAMIP.

*DATABASE Keyword	Options
*DATABASE_EXTENT_BINARY	MAXINT=-3, NINTSLD=8

Section

For isoparametric elements (which according to Ed Wilson (see www.edwilson.org) “The introduction of the isoparametric element formulation by Bruce Irons in 1968 was the single most significant contribution to the field of finite element analysis during the past 40 years”), one simply switches the element formulation to a fully integrated formulation. For shells, it is recommended to use the more advanced formulation ELFORM=-16 and for solids, the ELFORM=-1 does a nice job at reasonable numerical cost (see Tobias Erhart’s excellent work “Review of Solid Element Formulations in LS-DYNA”). Otherwise there is nothing special about setting up the definition. As with all isoparametric elements, one should not be too loose and fast with element quality. Good element quality always pays dividends in convergence and in pleasing stress results or in other words, if it looks good it is good. If one is interested in light discussion of the topic, take a look at “See Analysis Data’s True Colors” by Laird and Waterman.

*SECTION Keyword	Options
*SECTION_SHELL	ELFORM=-16, NIP=5
*SECTION_SOLID	ELFORM=-1

Why Implicit?

For most of us, being a good simulation engineer is being skeptical of anything that looks too easy; that is, we just know that we are going to find challenges somewhere, somehow. Our experience has followed this dictum of struggling to obtain convergence, uncovering new, undocumented features and being perplexed more often than we like in the pursuit of a quality static nonlinear implicit solution. The model shown in Figure 1 was no exception and convergence was eventually obtained by not rushing the process and paying attention to mesh quality, contact behavior and using the absolute latest version of LS-DYNA MPP Double-Precision.

The model follows all the standard keywords as outlined above. It sounds so easy but what would trip us up is the usual silly modeling omissions where joints were not properly setup and would create mechanisms and greatly hinder convergence. With hindsight, one would always create pilot models of complicated sections and debug these sections first, but that would be too logical, we would rather suffer.

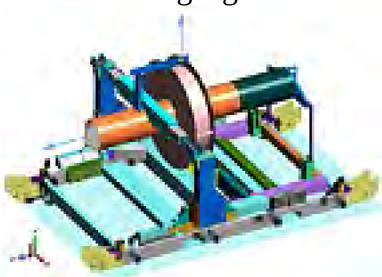
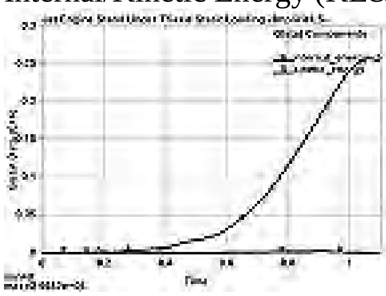
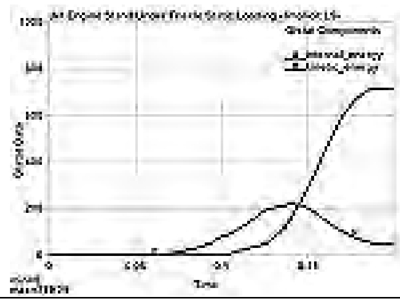
Manufacturing Jig-200k nodes/190k elements	Stress Results at End of Simulation
	
Internal/Kinetic Energy (KE&IE)	At Start--KE>IE Until Structure Stabilizes
	

Figure 1: Manufacturing jig that is subjected to static triaxial loading

Some additional notes about the model is that the kinetic energy (KE) does exceed the internal energy (IE) at the start of the simulation. This can be expected since the structure is moving under the load (high KE) and the strains are minimal (low IE). Once the structure stabilizes, the IE rapidly shoots up. At the end of the simulation the load is held constant from 1.0 to 1.1 and the analysis settles to equilibrium.

Figure 2: US Army's SCAT gun used to improve munition design

One of the more useful implicit applications is for model preload prior to explicit. This is something we have done many times with composite structures to test firing of instrumented test munitions (see Figure 2). This is done as one analysis routine within LS-DYNA and not a start/stop/re-start routine but a seamless run from implicit to explicit.

Troubleshooting

Although one hopes to read about some magic approach that will solve all your convergence issues, it doesn't exist as far as we know. Figure 3 shows an example of an analysis running along doing the usual convergence dance. The report of |du|/|u| tells us about how the structure is physically moving and deforming. When this shoots up, it means that the structure is un-constrained, has buckled or is just loose. The other measure of Ei/Eo is the strain energy. If this shoots up, then one typically has a large plasticity issue where finding equilibrium is difficult.

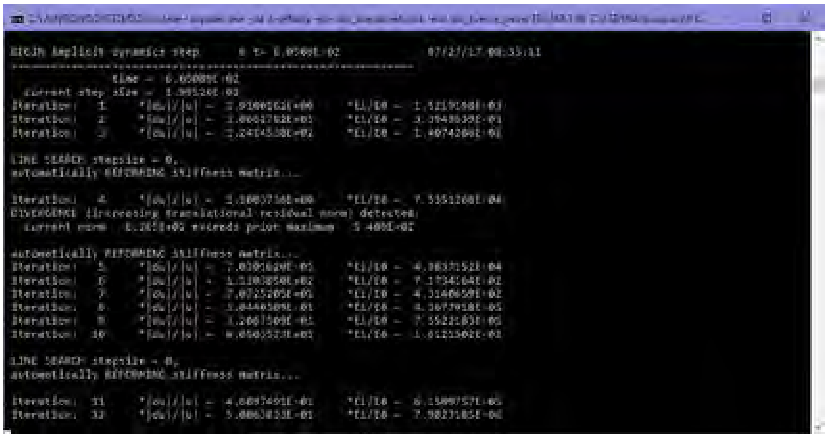


Figure 3:

LS-DYNA MPP Double-Precision analysis run with NLPRINT=2

Although one can recommend changing the number of equilibrium iterations prior to matrix reformulation or changing the line search method or adjusting the convergence tolerances, our experience with these practices has been dismal. What has worked for us is to tear apart the model and analyze sub-sections to determine why convergence is not occurring. What we like about the divide and conquer approach is that small models run quickly and lead to insights about why convergence is not occurring that can then be leveraged forward to future implicit simulations.

Another effective strategy that we have used to diagnose convergence problems is to contour the residual forces that are being calculated within an equilibrium iteration. This is done by setting D3ITCTL=1 in *CONTROL_IMPLICIT_SOLUTION and then RESPLT=1 in *DATABASE_EXTENT_BINARY.

For more details, please read the most excellent Appendix P in the Keyword Manual Vol. 1.

Wrap-Up - Lest we not forget, the title of this blog is Quick-Start and it is far from comprehensive. The idea is just to open the door toward using the implicit solver and remove some of the fear, uncertainty and doubt (FUD) about getting started. It should also be mentioned that the LSTC support team does an outstanding job in supporting the implicit solver and you will not be left dangling.

Thank you for your interest and we would welcome any feedback that you might have on this article

Materials Sciences Corporation MAT161/162

www.materials-sciences.com

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

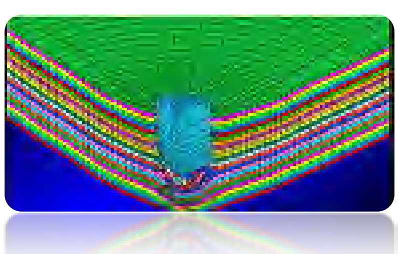
> Materials Sciences Corporation has provided engineering services to the composites industry since 1970. During this time, we have participated in numerous programs that demonstrate our ability to: perform advanced composite design, analysis and testing; provide overall program management; work in a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

Center for Composite Materials - University of Delaware - www.ccm.udel.edu/software/mat162/

MAT162 is a material model for use in LS-DYNA that may be used to simulate the onset and progression of damage in unidirectional and orthotropic fabric composite continua due to 3D stress fields. This failure model can be used to effectively simulate fiber dominated failures, matrix damage, and includes a stress-based delamination failure criterion. This approach to predicting interlaminar failure is advantageous in cases when locations of delamination sites (i.e., interlaminar crack initiation surfaces) cannot be anticipated.

Examples are located at www.ccm.udel.edu/software/mat162/examples /

Example 1:
Sphere Impact on a Composite Laminate
Example 2:
Sphere Impact on a Perfectly Clamped Composite Plate
Example 3:
Sphere Impact on Elliptical Carbon/Epoxy Tube



High Velocity Impact of Square Plate using MAT161/162

www.youtube.com/watch?v=NgjncjfLKGw

Shanghai Hengstar - LS-DYNA Sales, Consulting - Training

Shanghai Hengstar Technology with its head office in Shanghai was founded in 2009 by Dr. Hongsheng Lu, who worked as a senior scientist at LSTC headquarter in Livermore, USA. Besides software sales, Shanghai Hengstar provides engineering services, consulting and training that combine analysis and simulation using Finite Element Methods such as LS-DYNA.

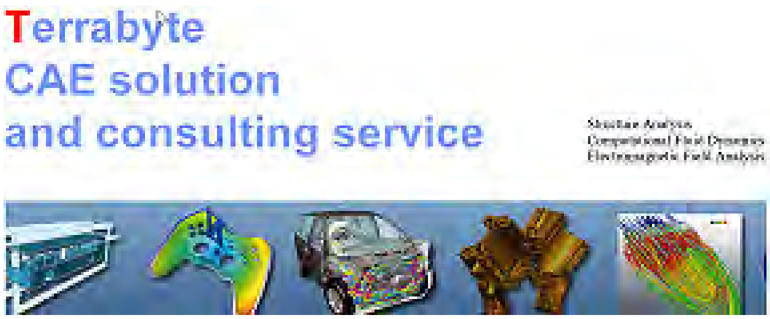
WEBINAR - Composite Material in LS-DYNA (Shanghai) - Instructor Dr. Tabiei was held Feb.

Upcoming Training Classes - Sign up Now.

10/31 Crashworthiness Simulation with LS-DYNA - Webinar - Shanghai
Instructors: Paul Du Bois (Sr. Consultant) Suri Bala (LSTC)

11/24 Material Failure and GISSMO in LS-DYNA - Shanghai
Instructor: Mr. Yanfei Fan (Hengstar)

www.terrabyte.co.jp/english/index.htm



Among the many products we provide

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. LS-DYNA is optimized for shared and distributed memory Unix, Linux, and Windows based platforms. The code's origins lie in highly nonlinear, transient dynamic FE analysis using explicit time integration.

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.

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.

CADFEM GmbH (Elite Channel Partner)

www.cadfem.de

Founded in 1985, CADFEM has established itself as a leading provider of CAE simulation software, consulting and support, employing more than 200 employees worldwide. Since its founding, CADFEM has overseen the complete range of products of the world's leading CAE provider ANSYS in Germany, Austria and Switzerland. CADFEM is the ANSYS Elite Channel Partner in Central Europe with offices in the Czech Republic, UK, Ireland, Poland, Russia, China, India and the U.S.A. In addition to software sales, CADFEM provides all services related to support, consulting, customization and seminars.

But because software alone cannot guarantee the success of a simulation, CADFEM is a systems vendor and now an engineering services provider with CADFEM Engineering Simulation Cloud. Their clients receive everything that is essential to their simulation success from a single source: software and IT solutions, advice, support, engineering, and transfer of expertise.

www.eta.com

www.flotrend.com.tw

恒士达科技
Hengstar Tech.
www.hengstar.com

www.autocae.cn

www.oasys-software.com/dyna

www.dynawe.com

www.agilesim.com.tw

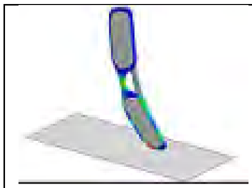
www.pan-i.com

http://dalianfukun.com

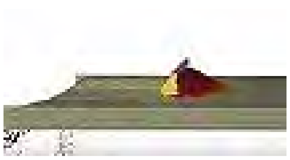
FEA Information China - For Sign Up or to offer Articles Contact:
Editors: Yanhua Zhao - Yanhua@feainformation.com

YouTube Choices for the Month & Gallery

Author: Marsha Victory mv@feainformation.com

	**Drop test analysis for Full Rim Eyeglass was performed using LS-DYNA(a Multi-physics solver).** www.youtube.com/watch?v=k-iXWD5vSG8
	**Vehicular Safety research developed at Grupo de Mecânica dos Sólidos e Impacto em Estruturas (GMSIE) - Escola Politécnica da Universidade de São Paulo** www.youtube.com/watch?v=A2Av9nygEoI

Previously showcased

	**LS-DYNA SPH : Earth-moving simulation** https://www.youtube.com/watch?v=GvShD3nnM2c **[1] An accelerated, convergent, and stable nodal integration in Galerkin meshfree methods for linear and nonlinear mechanics** **[2] A quasi-linear reproducing kernel particle method**
	**LS-DYNA Steel Profile Stamping Analysis - Video Tutorial** Ahmet OKUDAN - Published on Jun 10, 2017 www.youtube.com/watch?v=F5b5Uiw9aFs
	**LS-DYNA CFD : Flow through a funnel** LS-DYNA Multiphysics Published on Jun 2, 2017 www.youtube.com/watch?v=1ijUDN2yK4k
	**LS-DYNA - Thermal Stress Analysis** Sercan Devrim - Published on Jun 13, 2017 www.youtube.com/watch?v=apzGJdDEGGw

Educational Institutions - LS-DYNA

LS-DYNA®
Conferences

Published Conference Papers from participants of the following educational institutions.

Papers are located at www.dynalook.com

- Aalborg University
- Bergen University College
- Brown University
- Chongqing University
- Cité Universitaire
- Clemson University
- Cranfield University
- De Montfort University
- Gazi University
- Gdańsk University of Technology
- George Mason University
- Glasgow Caledonian University
- Helsinki University of Technology
- Hunan University
- Kazan State University, Russia
- Kettering University
- King Mongkut's University of North Bangkok
- King Saud University
- Konkuk University
- Konkuk University
- Konkuk University
- Konkuk University
- Kumamoto University
- Laval University
- Linköping University
- Linköping University
- Luleå University of Technology
- Marquette University
- Michigan State University
- Moscow State University
- National Research Nuclear University
- North Carolina A & T State University
- Norwegian University of Science and Technology
- Nottingham Trent University
- Ohio State University
- Ohio University
- Padova University
- Queen Mary University of London
- Russia Saint Petersburg State Polytechnical University, Russia
- RWTH Aachen/University of Agder
- Ryerson University
- Second University of Naples
- Shanghai Jiaotong University
- South-Ural State University
- St.Petersburg State Polytechnical University
- Strasbourg University

Educational Institutions - LS-DYNA

- Technical University Munich
- Technical University of Koszalin
- Technical University, Riga
- Texas A&M University
- The George Washington University
- The Johns Hopkins University Baltimore
- The Ohio State University
- The University of Bolton, UK
- The University of British Columbia
- The University of British Columbia
- The University of Mississippi
- The University of Texas at Austin
- TOBB University of Economics and Technology
- Universite de Lille
- University Edinburgh
- University of Akron
- University of Alabama at Birmingham
- University of Applied Sciences, Aalen
- University of Bristol
- University of California at Berkeley
- University of California, San Diego
- University of Central Florida, Orlando
- University of Cincinnati
- University of Kassel
- University of Liverpool
- University of Liverpool
- University of Maribor
- University of Maribor, Slovenia
- University of Massachusetts Lowell
- University of Missouri
- University of Naples "Federico II
- University of Nebraska-Lincoln
- University of Nizhniy Novgorod
- University of Pittsburgh
- University of Pretoria
- University of Sheffield
- University of Surrey
- University of Toronto
- University of Turin
- University of Warwick
- University of Waterloo
- University of Windsor
- University of Wolverhampton
- Washington State University
- Wayne State University

Website Month Showcase – Sites to Visit for Information

www.fujitsu.com/jp	**www.infinite.nl**
www.dynamax-inc.com	www.dynasplus.com
www.beta-cae.com	**www.dynamore.de**
www.materials-sciences.com	**www.dalianfukun.com** **www.dell.com**
www.eta.com	www.lancemore.jp/index_en.html

AUTOMOTIVE NEWS & EVENTS

Editor: Dilip Bhalsod

The purpose of this section is to provide a place, for our automotive readers, to share news and events relative to their company and/or products.

The criteria for submitting information is as follows:

- It has to be public information
- Published on the Internet
- Be automotive informational, or human interest.
- We do not accept financial quarterly information

We would welcome the opportunity to share information about your company with our readership.

You may send Title to your information and the accompanying URL to agiac99@aol.com - Subject Line please use " Automotive News"

Submissions should be received by the 15th of each month, of the month you want your article placed

Submission publications is at the sole discretion of FEA Information Inc.

The following are copyright© to their respective companies.

New Mustang GT Owners Can Now Hush the V8 Growl to be Good Neighbors

New Mustang GT Owners Can Now Hush the V8 Growl to be Good Neighbors; Brush with the Law Gave Engineer the Idea

- Industry-first Quiet Start – also called "Good Neighbor Mode" by Mustang engineers – allows drivers to schedule the time of day when their Mustang GT's V8 engine will roar and when to keep it quiet to show courtesy to neighbors
- New Quiet Exhaust mode limits volume of 2018 Mustang GT's 5.0-liter V8 engine using active valve performance exhaust system that closes valves to restrict the amount of noise made by the car
- According to a recent poll by Ranker.com, loud engine revving ranks among the most annoying noises neighbors make, alongside other common nuisances including lawn mowers, power tools, barking dogs and band practice

DEARBORN, Mich., Aug. 1, 2017 – Someone called the cops on Steve von Foerster. The former head of vehicle engineering for Ford Motor Company wasn't thrilled, but he understood why.

On an otherwise peaceful morning in his suburban Detroit neighborhood, von Foerster had just backed a Shelby GT350 Mustang out of his driveway. As the car's V8 engine thundered, an annoyed neighbor set aside their coffee cup and dialed 911.

Von Foerster had left before the officers arrived, and he didn't end up with a ticket. Nor did he get angry. What he got was an idea for the new Mustang.

"I love the sound of the V8, but it can be loud, and you can't annoy people like that in your neighborhood," said von Foerster, who now leads Ford's user experience team in product development. "It sounds so cool, but I thought, 'There has to be a way to give people more control over the engine's sound.'"

The experience fueled a discussion between von Foerster and fellow Mustang program team members that led to development of Quiet Exhaust mode and industry-first Quiet Start, known as "Good Neighbor Mode" among Mustang engineers. The new features on properly equipped 2018 Mustang GT vehicles allow drivers to keep engine sound at a minimum if they wish and to program quiet start-up times in advance.

New Mustang GT Owners Can Now Hush the V8 Growl to be Good Neighbors

While some sports cars offer active exhaust systems with on/off functionality, Mustang's Quiet Start is the first to allow scheduling of times. Using steering wheel-mounted thumb controls, drivers toggle through a menu in the instrument cluster to select when they want to fire up their Mustang GT without sharing the event with neighbors. For example, between 8 p.m. and 7 a.m., drivers can keep the peace by scheduling their car to start, minus the roar.

Both new Quiet Exhaust mode and Quiet Start features will be part of the available active valve performance exhaust system on the new Mustang GT, along with different exhaust volumes for Normal, Sport and Track modes, giving customers more choice than ever before.

"Active valve performance exhaust gives Mustang owners the best of both worlds – that classic Mustang sound, and the ability to not wake up your neighbors when you leave the house early in the morning or arrive home late at night," said Matt Flis, Ford exhaust development engineer.

Neighborhoods across the country should be thrilled. According to a recent poll by Ranker.com, loud engine revving ranks among the most annoying noises neighbors make, alongside other common nuisances including power tools, barking dogs and band practice. Only early-morning lawn mowing is more robustly despised. With new Quiet Exhaust mode and Quiet Start, Ford is doing its part to keep the peace.

"When sounds get up into the upper-70-decibel range, that's typically about when they start to bother people," said Flis. "With quiet start activated, the decibel level of the new Mustang GT drops by about 10 decibels, to a much more comfortable 72 decibels – about the level of a household dishwasher."

On Mustang GT equipped with the available all-digital 12-inch instrument cluster, the exhaust mode menu appears within the pony menu. With the standard 4-inch cluster, exhaust mode is found within the settings menu.

Active valve performance exhaust is one of many changes Mustang fans will see when cars reach showrooms later this year. In addition to its restyled exterior design and refined interior, Mustang will offer an all-digital 12-inch instrument cluster, a 10-speed automatic transmission, MagneRide™ damping system and SYNC® Connect with FordPass™.

With its new Drag Strip mode, Mustang GT now reaches 60 mph in under four seconds – faster than a $94,000 Porsche 911 Carrera – setting a new standard as the quickest Mustang GT ever.

AEROSPACE NEWS & EVENTS

Editor: Marnie Azadian

The purpose of this section is to provide a place, for our aerospace readers, to share news and events relative to their company and/or products.

The criteria for submitting information is as follows:

- It has to be public information
- An internet URL
- Be technical, informational, or human interest.
- We do not accept financial quarterly information

We would welcome the opportunity to share information about your company with our readership.

You may send Title to your information and the accompanying URL to agiac99@aol.com - Subject Line please use "Aerospace News"

Submissions should be received by the 15th of each month, of the month you want your article placed. For example: We would need the title of the news or event by December 15th, 2015 to be featured in the December 2015 FEA newsletter.

Submission publications is at the sole discretion of FEA Information Inc.

The following are copyright© to their respective companies.

F-35 Developmental Testers Surge Toward IOC

(Source: US Air Force 412th Test Wing; issued August 17, 2017)

An F-35 fires an AIM-120 missile over the Pacific Ocean range near NAWS Point Mugu during a recent Weapons Delivery Accuracy testing surge.
(Lockheed Martin photo)

EDWARDS AFB, Calif. --- Edwards Air Force Base is famous for aviation firsts, but the F-35 Developmental Test team recently achieved the opposite, an aviation last.

The Block 3F version of the F-35A and F-35C are one step closer to initial operating capability with the last weapons delivery accuracy (WDA) surge completed in early August.

The term surge is used because the F-35 weapons testing required a great deal of assets and personnel dedicated for a short period of time, according to Torrey Given, a weapons integration engineer with the 461st Flight Test Squadron.

"This allowed us to accomplish multiple events over the course of a handful of days," he said.

Given, who has been with the F-35 project since the beginning, from flight science all the way through the WDA testing, said this round of testing could be called a mini surge, as there were only a handful of events that needed to be completed.

"This was kind of a cleanup, or a closeout, of (System Development and Demonstration). It's the closeout of JSF developmental test for Block 3F, which is a big deal because it's for Air Force IOC, and Navy IOC," he said.

"We were able to accomplish some complex air-to-air demonstrations with the (Advanced Medium-Range Air-to-Air Missile) in order to show the full capability of the aircraft," Given said.

The AMRAAM missiles were equipped with live motors and guidance systems, but the warheads were exchanged for telemetry units, and the tests were done over the open water range at Naval Air Weapons Station Point Mugu in Ventura County, California.

Given said most everything we do at Edwards Air Force Base is an aviation first, in some way, shape or form.

"I like to think of these as the last for Developmental Test," he said. "This is like our graduation exercise before we hand the aircraft off to the operational test organizations so they can go prove it's ready for combat. That's very significant for us." He said the critical piece is the wrap-up of System Development and Demonstration, which has been a long time coming.

"For some of us, this has been roughly eight years working on F-35 to get to where it is operationally viable," Given said. There is a single WDA event remaining for 3F, which marks the completion for all three variants and will pave the way to the declaration of IOC for all F-35s.

BETA CAE Systems. www.beta-cae.com

BETA CAE Systems - ANSA

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

BETA CAE Systems μETA

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

Solutions for:

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

DatapointLabs **www.datapointlabs.com**

Testing over 1000 materials per year for a wide range of physical properties, DatapointLabs is a center of excellence providing global support to industries engaged in new product development and R&D.

The company meets the material property needs of CAE/FEA analysts, with a specialized product line, TestPaks®, which allow CAE analysts to easily order material testing for the calibration of over 100 different material models.

DatapointLabs maintains a world-class testing facility with expertise in physical properties of plastics, rubber, food, ceramics, and metals.

Core competencies include mechanical, thermal and flow properties of materials with a focus on precision properties for use in product development and R&D.

Engineering Design Data including material model calibrations for CAE Research Support Services, your personal expert testing laboratory Lab Facilities gives you a glimpse of our extensive test facilities Test Catalog gets you instant quotes for over 200 physical properties.

ETA – Engineering Technology Associates **www.eta.com**

etainfo@eta.com

Inventium Suite™

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

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

PreSys

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

VPG

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

DYNAFORM

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

Latest Release is ESI Visual-Environment 12.0

ESI Group **www.esi-group.com**

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

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

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

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

Latest Release is ESI Visual-Environment 12.0

ESI Group **www.esi-group.com**

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

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

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

JSOL

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

HYCRASH

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

JSTAMP/NV

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

JMAG

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

Livermore Software Technology Corp. www.lstc.com

LS-DYNA

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

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

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

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

LSTC Dummy Models:

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

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

Material Sciences Corporation **info@materials-sciences.com**

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

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

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

MSC/LS-DYNA Composite Software and Database -

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

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

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

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

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

Oasys PRIMER

Key benefits:

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

www.oasys-software.com/dyna

Oasys D3PLOT

Key benefits:

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

Oasys
The Software House of Arup

Oasys T/HIS

Key benefits:

- Graphical post-processor created specifically for LS-DYNA®
- Automatically reads all LS-DYNA® results
- Wide range of functions and injury criteria
- Easy handling of data from multiple models
- Scripting capabilities for fast post-processing

Oasys REPORTER

Key benefits:

- Automatic report generation tool created specifically for LS-DYNA®
- Automatically post-process and summarize multiple analyses
- Built-in report templates for easy automatic post-processing of many standard impact tests

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

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

Our History

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

恒士达科技 Hengstar Tech.

Shanghai Hengstar

www.hengstar.com

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

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

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

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

Consulting

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

Lenovo **www.lenovo.com**

Lenovo is a USD39 billion personal and enterprise technology company, serving customers in more than 160 countries.

Dedicated to building exceptionally engineered PCs, mobile Internet devices and servers spanning entry through supercomputers, Lenovo has built its business on product innovation, a highly efficient global supply chain and strong strategic execution. The company develops, manufactures and markets reliable, high-quality, secure and easy-to-use technology products and services.

Lenovo acquired IBM’s x86 server business in 2014. With this acquisition, Lenovo added award-winning System x enterprise server portfolio along with HPC and CAE expertise.

Canada **Metal Forming Analysis Corp MFAC** galb@mfac.com

www.mfac.com

LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
LSTC Dummy Models	LSTC Barrier Models	eta/VPG	
eta/DYNAFORM	INVENTIUM/PreSys		

Mexico **COMPLX** Armando Toledo

www.complx.com.mx / armando.toledo@complx.com.mx

LS-DYNA LS-OPT LS-PrePost

LS-TAsc Barrier/Dummy Models

United States **DYNAMAX** sales@dynamax-inc.com

www.dynamax-inc.com

LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
LSTC Dummy Models		LSTC Barrier Models	

United States **Livermore Software Technology Corp** sales@lstc.com

LSTC www.lstc.com

LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
LSTC Dummy Models	LSTC Barrier Models	TOYOTA THUMS	

United States	**ESI Group N.A** **info@esi-group.com**			
	www.esi-group.com			
	PAM-STAMP			
	QuikCAST	SYSWELD	PAM-COMPOSITES	CEM One
	VA One	CFD-ACE+	ProCAST	
		Weld Planner	Visual-Environment	IC.IDO

United States	**Engineering Technology Associates – ETA** etainfo@eta.com www.eta.com			
	INVENTIUM/PreSy	NISA	VPG	LS-DYNA
	LS-OPT	DYNAform		

United States	**Predictive Engineering** george.laird@predictiveengineering.com www.predictiveengineering.com			
	FEMAP	NX Nastran	LS-DYNA	LS-OPT
	LS-PrePost	LS-TaSC	LSTC Dummy Models	
			LSTC Barrier Models	

France	**DynaS+**		v.lapoujade@dynasplus.com	
	www.dynasplus.com		Oasys Suite	
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	DYNAFORM	VPG	MEDINA	
	LSTC Dummy Models		LSTC Barrier Models	

France	**DYNAmore France SAS**		sales@dynamore.eu	
	www.dynamore.eu			
	LS-DYNA, LS-PrePost	LS-OPT	Primer	DYNAFORM
	DSDM Products		LSTC Dummy Models	FEMZIP
	LSTC Barrier Models		DIGIMAT	

Germany	**CADFEM GmbH**		lsdyna@cadfem.de	
	www.cadfem.de			
	ANSYS	LS-DYNA	optiSLang	
	ESAComp	AnyBody		
	ANSYS/LS-DYNA			

Germany	**DYNAmore GmbH**		uli.franz@dynamore.de	
	www.dynamore.de			
	PRIMER	LS-DYNA	FTSS	VisualDoc
	LS-OPT	LS-PrePost	LS-TaSC	DYNAFORM
	Primer	FEMZIP	GENESIS	Oasys Suite
	TOYOTA THUMS		LSTC Dummy & Barrier Models	

The Netherlands	**Infinite Simulation Systems B.V**		j.mathijssen@infinite.nl	
	www.infinite.nl			
	ANSYS Products	CivilFem	CFX	Fluent
	LS-DYNA	LS-PrePost	LS-OPT	LS-TaSC

Russia	**STRELA**		info@dynarussia.com	
	LS-DYNA	LS-TaSC	LS-OPT	LS-PrePost
	LSTC Dummy Models		LSTC Barrier Models	

Spain	**DYNAmore France SAS**		sales@dynamore.eu	
	www.dynamore.eu			
	LS-DYNA, LS-OPT	LS-PrePost	Primer	DYNAFORM
	DSDM Products		LSTC Dummy Models	FEMZIP
	LSTC Barrier Models		DIGIMAT	
Sweden	**DYNAmore Nordic**		marcus.redhe@dynamore.se	
	www.dynamore.se		Oasys Suite	
	ANSA	µETA	LS-DYNA	LS-OPT
	LS-PrePost	LS-TaSC	FastFORM	DYNAform
	FormingSuite		LSTC Dummy Models	
			LSTC Barrier Models	
Switzerland	**DYNAmoreSwiss GmbH**		info@dynamore.ch	
	www.dynamore.ch			
	LS-DYNA		LS-OPT	LS-PrePost
	LS-TaSC		LSTC Dummy Models &	Barrier Models
UK	**Ove Arup & Partners**	dyna.sales@arup.com		
	www.oasys-software.com/dyna		TOYOTA THUMS	
	LS-DYNA		LS-OPT	LS-PrePost
	LS-TaSC		PRIMER	D3PLOT T/HIS
	REPORTER	SHELL	FEMZIP	HYCRASH
	DIGIMAT	Simpleware	LSTC Dummy Models	
			LSTC Barrier Models	

China	**ETA – China**		lma@eta.com.cn	
	www.eta.com/cn			
	Inventium	VPG	DYNAFORM	NISA
	LS-DYNA	LS-OPT	LSTC Dummy Models	LS-PrePost
			LSTC Barrier Models	LS-TaSC

China	**Oasys Ltd. China**		de-long.ge@arup.com	
	www.oasys-software.com/dyna			
	PRIMER D3PLOT	HYCRASH	T/HIS REPORTER	SHELL
	LS-DYNA	LS-OPT	LSTC Dummy Models	LS-PrePost
	DIGIMAT	FEMZIP	LSTC Barrier Models	LS-TaSC

China	**Shanghai Hengstar Technology**		info@hengstar.com	
	www.hengstar.com			
	LS-DYNA	LS-TaSC	LSTC Barrier Models	D3VIEW
	LS-PrePOST	LS-OPT	LSTC Dummy Models	
	Genesis	VisualDoc		ELSDYNA
	Visual-Crahs DYNA	Visual-Proeces		DynaX & MadyX
	Enki Bonnet	Visual Environement		

India	**Oasys Ltd. India**		lavendra.singh@arup.com	
	www.oasys-software.com/dyna			
	PRIMER D3PLOT	T/HIS		
		LS-OPT	LSTC Dummy Models	LS-PrePost
		LS-DYNA	LSTC Barrier Models	LS-TaSC

India	**CADFEM Eng. Svce**		info@cadfem.in	
	www.cadfem.in			
	ANSYS	VPS	ESAComp	optiSLang
	LS-DYNA	LS-OPT	LS-PrePost	

India	**Kaizenat Technologies Pvt. Ltd**		support@kaizenat.com	
	http://kaizenat.com /			
	LS-DYNA	LS-OPT	LSTC Dummy Models	LS-PrePost
	Complete LS-DYNA suite of products		LSTC Barrier Models	LS-TaSC

Japan	**CTC**		LS-dyna@ctc-g.co.jp	
	www.engineering-eye.com			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	CmWAVE	
Japan	**JSOL**			
	www.jsol.co.jp/english/cae		Oasys Suite	
	JSTAMP	HYCRASH	JMAG	
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	TOYOTA THUMS	
Japan	**FUJITSU**			
	http://www.fujitsu.com/jp/solutions/business-technology/tc/sol/			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	CLOUD Services	
Japan	**LANCEMORE**	info@lancemore.jp		
	www.lancemore.jp/index_en.html			
	Consulting			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models		
Japan	**Terrabyte**	**English:**		
	www.terrabyte.co.jp	www.terrabyte.co.jp/english/index.htm		
	Consulting			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	AnyBody	

Korea	**THEME**	wschung7@gmail.com		
	www.lsdyna.co.kr		Oasys Suite	
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	eta/VPG	Planets
	eta/DYNAFORM	FormingSuite	Simblow	TrueGRID
	JSTAMP/NV	Scan IP	Scan FE	Scan CAD
	FEMZIP			

Korea	**KOSTECH**	young@kostech.co.kr		
	www.kostech.co.kr			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM
	eta/DYNAFORM	DIGIMAT	Simuform	Simpack
	AxStream	TrueGrid	FEMZIP	

Taiwan	**AgileSim Technology Corp.**			
	www.agilesim.com.tw			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM

Taiwan	**Flotrend**			
	www.flotrend.com.tw			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM

Taiwan	**SiMWARE Inc..**			
	www.simware.com.tw			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM

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

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

JSOL Corporation is cooperating with chosen cloud computing services

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

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

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

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

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

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

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

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

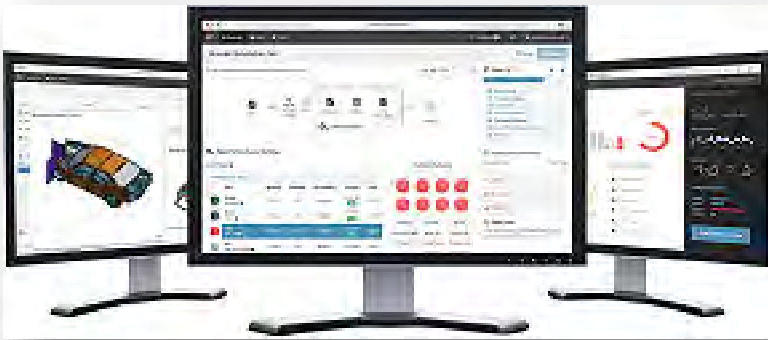
Platform Computation Cloud
CreDist.Inc.

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

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

Rescale Cloud Simulation Platform

www.rescale.com



Rescale: Cloud Simulation Platform

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

Using the power of cloud technology combined with LS-DYNA allows you to:
· Accelerate complex simulations and fully explore the design space
· Optimize the analysis process with hourly software and hardware resources
· Leverage agile IT resources to provide flexibility and scalability

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

Rescale has strategic/technology partnerships with infrastructure and software providers to offer the following:
· Largest global hardware footprint – GPUs, Xeon Phi, InfiniBand
· Customizable configurations to meet every simulation demand
· Worldwide resource access provides industry-leading tools to every team
· Pay-per-use business model means you only pay for the resources you use
· True on-demand resources – no more queues

ScaleX Enterprise: Transform IT, Empower Engineers, Unleash Innovation
The ScaleX Enterprise simulation platform provides scalability and flexibility to companies while offering enterprise IT and management teams the opportunity to expand and empower their organizations.

Rescale Cloud Simulation Platform

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

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

Industry-Leading Security

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

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

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

LSTC - DYNAmore GmbH JSOL Corporation

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

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

ESI Cloud Based Virtual Engineering Solutions

www.esi-group.com

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

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

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

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

Virtual Performance Solution:

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

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

The benefits of VPS hybrid on ESI Cloud include:

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

ESI Cloud Based Virtual Engineering Solutions

www.esi-group.com

VPS On Demand

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

Key solution capabilities:

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

Result visualization

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

Key result visualization capabilities:

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

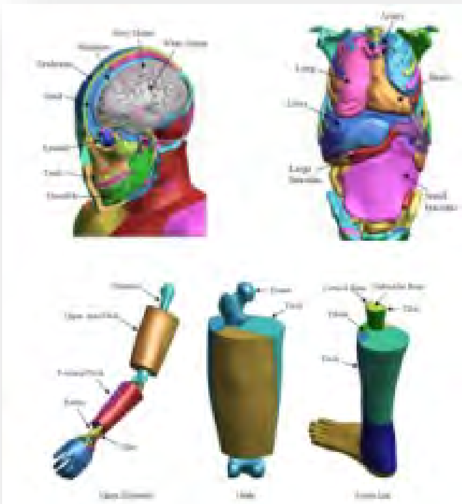
Collaboration

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

Key collaboration capabilities:

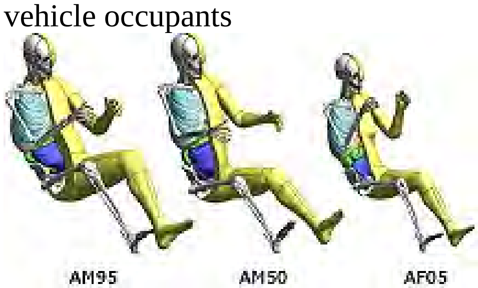
- Data, workflow or project asynchronous collaboration
- Multi-user, browser based collaboration for CAD, geometry, mesh and results models
- Real-time design review with notes, annotations and images archiving and retrieval
- Email invite to non ESI Cloud users for real time collaboration

TOYOTA - Total Human Model for Safety – THUMS

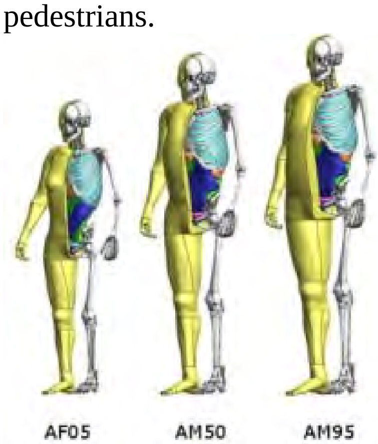


The Total Human Model for Safety, or THUMS®, is a joint development of Toyota Motor Corporation and Toyota Central R&D Labs. Unlike dummy models, which are simplified representation of humans, THUMS represents actual humans in detail, including the outer shape, but also bones, muscles, ligaments, tendons,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

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



and as standing model to represent pedestrians.



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

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

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

For information please contact: THUMS@lstc.com

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

LSTC – Dummy Models

LSTC Crash Test Dummies (ATD)

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

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

e-mail to: atds@lstc.com

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

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

Models In Development

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

Planned Models

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

LSTC – Barrier Models

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

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

•ODB modeled with shell elements
•ODB modeled with solid elements
•ODB modeled with a combination of shell and solid elements

•MDB according to FMVSS 214 modeled with shell elements
•MDB according to FMVSS 214 modeled with solid elements

•MDB according to ECE R-95 modeled with shell elements
•AE-MDB modeled with shell elements

•IIHS MDB modeled with shell elements
•IIHS MDB modeled with solid elements
•RCAR bumper barrier

•RMDB modeled with shell and solid elements

e-mail to: atds@lstc.com.

Conference – Events – Users Meetings

Keep up to date on upcoming

Conferences

Meetings

Events

if you have a new event to be listed please send to agiac99@aol.com

Conference/Events/User Forums

Sep 26-27	**ESI Forum in North America 2017** Birmingham, MI , US	www.esi-group.com
Oct. 11-13	**The 7th GACM Colloquium on Computational Mechanics (GACM 2017**)	www.gacm2017.uni-stuttgart.de
Oct. 23-25	**3rd China LS-DYNA User's conference** Shanghai, China	http://www.lsdyna.cn
Oct 31-Nov 1	**LS-DYNA&JSTAMP Forum 2017** Tokyo, Japan	http://ls-dyna.jsol.co.jp/en
Nov 15-16	**PUCA FORUM 2017** Tokyo, Japan	www.esi-group.com

7th GACM Colloquium on Computational Mechanics 2017

The 7th GACM Colloquium on Computational Mechanics (GACM 2017) will take place on October 11-13 2017 in Stuttgart, Germany. The colloquium is hosted by the Institute of Structural Mechanics and the Institute of Applied Mechanics of the University of Stuttgart in cooperation with DYNAmore GmbH.

The GACM Colloquium on Computational Mechanics is intended for young scientists who are engaged in academic and industrial research on Computational Mechanics and Computer Methods in Applied Sciences. It provides a platform to present and discuss results from recent research efforts and industrial applications.

Thematically arranged sessions and organized mini-symposia as well as social events will provide an environment for lively discussions in an informal atmosphere.

Young scientists from Europe and all other continents are welcome to this colloquium. Presentations will be given in English.

www.gacm2017.uni-stuttgart.de

LS-DYNA & JSTAMP Forum 2017 in Tokyo, Japan

http://ls-dyna.jsol.co.jp/en/event/uf2017/

Welcome to the LS-DYNA & JSTAMP Forum 2017

JSOL Corporation holds an annual LS-DYNA & JSTAMP Forum to provide our users a wide range of information including the latest simulation technologies and case studies and also to offer the opportunity for information exchange among our users.

This year the venue of the LS-DYNA & JSTAMP Forum 2017 moves from Nagoya to Tokyo. It will be held at Tokyo Conference Center Shinagawa, from Tuesday 31 October to Wednesday 1 November 2017. Our engineers will showcase the latest simulation technologies and poster sessions will be held. We welcome any inquiry, consultation and discussion about your day-to-day work.

We encourage our users to take advantage of this once a year opportunity. We look forward to your attendance in the event.

JSOL Corporation - Engineering Technology Division

LS-DYNA&JSTAMP Forum 2017 - Organizer: JSOL Corporation

Dates: Tuesday 31 October to Wednesday 1 November 2017.
Venue: Tokyo Conference Center Shinagawa (Tokyo, Japan)

ESI Group User Forums

www.esi-group.com

ESI User Forums

26 Sep 2017 - 27 Sep 2017
ESI Forum in North America 2017
Birmingham, Michigan, United States

26 Sep 2017 - 27 Sep 2017
VPS User Conference 2017
Czech Republic

17 Oct 2017 - 19 Oct 2017
5th OpenFOAM User Conference 2017
Frankfurt/Main, Germany

7 Nov 2017 - 9 Nov 2017
ESI Forum in Germany 2017
Weimar, Germany

15 Nov 2017 - 16 Nov 2017
PUCA FORUM 2017
Hilton Hotel Tokyo, Japan

22 Nov 2017
ESI Forum in India 2017
Pune, India

2017 3rd China LS-DYNA User's conference Oct. 23rd-25th, Shanghai, China

Yanhua Zhao & China Conference Team - chinaconf@lstc.com

The 3rd China LS-DYNA conference will echo the success of the well-participated 1st and 2nd China User's Conference, in 2013 and 2015.

Accompanied by the rapid growth of CAE applications in China, LS-DYNA is highly recognized as one of the most widely used finite element analysis software by Chinese users.

China is gaining momentum and recognition in Finite Element Analysis. In the past years, the continuing expansion of application areas has been gaining more users in automotive, die and mold, aerospace and aeronautics industries in China.

In China LS-DYNA is fast becoming the software of choice, by all engineers, students, professors and consulting companies. It is recognized that LS-DYNA, LS-PrePost, LS-OPT and the LSTC ATD and Barrier Models, developed by LSTC, are setting standards for the finite element simulation industry. At the conference LSTC software new features will be introduced and helpful techniques will be shared.

The conference will be attended by experienced users from different industries, LSTC technical support engineers and software developers. Additionally, it will be attended by academic researchers, hardware vendors and software vendors.

With the popularity and attendance of the 1st and 2nd conference and demand from users it has been decided that the conference will be held regularly. One of the goals is to serve as a convenient platform for people in this field to exchange their ideas, share their findings and explore new software functions.

Hosts: Livermore Software Technology Corp. & Dalian Fukun Technology Development Corp.
Date: Oct. 23rd -25th, 2017
Location: InterContinental Shanghai Pudong, Shanghai, China
Website: http://www.lsdyna.cn
Contact: chinaconf@lstc.com

Training and Social Media Section

Training

Classes

Webinars

On Site – On Line

We will be adding to this section monthly – if you have a new event to be listed please send to Anthony agiac99@aol.com

Participant's Training Classes

Webinars

Info Days

Class Directory

Participant Class Directory

Arup (corporate)	www.oasys-software.com/dyna/en/training
BETA CAE Systems (corporate)	www.beta-cae.com/training.htm
DYNAmore (corporate)	www.dynamore.de/en/training/seminars
ESI-Group (corporate)	https://myesi.esi-group.com/trainings/schedules
ETA (corporate)	www.eta.com/support2/training-calendar
KOSTECH	www.kostech.co.kr/
LSTC - (corporate)	www.lstc.com/training
LS-DYNA OnLine - (Al Tabiei)	www.LSDYNA-ONLINE.COM

ARUP Visit the website for complete listings/changes/locations

www.oasys-software.com/dyna/en/training

Arup offers a wide range of training for new and existing users of the Oasys LS-DYNA Environment software who are seeking to improve their understanding and application of these powerful analysis tools. New users will benefit from our introductory courses and can quickly become effective in other areas of application through the range of courses on offer. The courses will also provide existing users with knowledge of how to use the latest features in Oasys and LS-DYNA.

BETA CAE SYSTEMS

Visit the website for complete listings/changes/locations

www.beta-cae.com/training.htm

Basic and advanced training courses can be scheduled upon request. A variety of standard or tailored training schedules, per product or per discipline, are being offered to meet customers needs.

A number of recommended training courses offered are described below. The list is not exhaustive and more courses can be designed according to your needs.
Please, contact ansa@beta-cae.com for further details.

Recommended Training Courses (Complete information on website)

- SPDRM
- ANSA / μETA Basics
- ANSA / μETA for CFD
- ANSA / μETA for Crash & Safety simulation
- ANSA / μETA for Durability simulation
- ANSA / μETA for NVH analyses
- Multi-Body Dynamics
- Laminated Composites
- Morphing and Optimization
- Automation
- Additional special sessions

DYNAmore Training Dates 2017

Author: Christian Frech christian.frech@dynamore.de

DYNAmore Visit the website for complete overview and registration

www.dynamore.de/seminars

Seminar dates offered by DYNAmore – September 2017

Download full seminar brochure (pdf): www.dynamore.de/seminars2017

Selection of trainings from September

Introduction	
Introduction to LS-PrePost	11 September (Tr)
Introduction to LS-DYNA	12-14 September (Tr)
Crash/Short-Term Dynamics	
Contact Definitions in LS-DYNA	15 September
Passive Safety	
Introduction to Passive Safety Simulation with LS-DYNA	28-29 September
Implicit	
Implicit Analysis with LS-DYNA	25-26 September
Particle Methods	
Smoothed Particle Hydrodynamics (SPH) in LS-DYNA	21-22 September
Discrete Element Method (DEM) in LS-DYNA	27 September
Multiphysics/Biomechanics	
ALE and Fluid-Structure Interaction in LS-DYNA	19-20 September
Optimization	
LS-OPT – Optimization and Robustness	12-14 (G)/19-21 (V) September

Information days (free of charge)

Possibilities with LS-DYNA/Implicit	18 September
Optimization/DOE/Robustness	25 September

Support/Webinar series (free of charge) – Registration via www.dynamore.de

Support day: LS-DYNA	15 September

If not otherwise stated, the event location is Stuttgart, Germany. Other event locations are:
G = Gothenburg, Sweden; L = Linköping, Sweden V = Versailles, France; T = Turin, Italy, Sb = Salzburg, Austria, Tr = Traboch, Austria

We hope that our offer will meet your needs and are looking forward to welcoming you at one of the events.

DYNAmore Information Day Welding and Heat Treatment w/LS-DYNA

Author: Christian Frech christian.frech@dynamore.de

Information day Welding and Heat Treatment with LS-DYNA,

17 October, Aachen

Due to the increasing importance of simulations with welding processes and other heat treatments, numerous extensions have been implemented in LS-DYNA. It is now possible to calculate the complete process chain in several stages.

LS-DYNA furthermore offers special heat source functions for shells and solids with energy input control and special welding contacts such that all welding processes can be captured.

This information day aims at simulation engineers who want to obtain an overview of the available tools in LS-DYNA, DynaWeld and SimWeld that can used for model building as well as simulation of welding and heat treatment processes.

Registration:
www.dynamore.de/info-welding-e

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

Among the many classes held during the year are the following:

September

11-12 LS-DYNA ALE/Euler
18-19 Intro LS-OPT – Funcunality & Standard
20 LS-DYNA Discrete Element Method
25-17 Intro to LS-DYNA Explicit

October

09 Intro to LS-DYNA Implicit
10 LS-DYNA Implicit - Advanced
16-17 LS-DYNA CFD Incompressible - ICFD
18 LS-DYNA CFD Compressible - CESE
19-20 LS-DYNA Electromagnetism - EM

https://myesi.esi-group.com/trainings/schedules

VPS Explicit - PAM-CRASH I - Grundlagen (Basics)
5 Sep 2017 to 7 Sep 2017

Please visit the website for complete information on all the classes and locations

https://myesi.esi-group.com/trainings/schedules

KOrea Simulation TECHnology Co., Ltd. Training

www.kostech.co.kr/

Anna Choi, Assistant Manager - choian@kostech.co.kr
KOrea Simulation TECHnology Co.,Ltd [Kostech]
Rm. 804 Nam-Jung City Plaza 1th, 760 Janghang-dong
Ilsandong-gu, Goyang-si, Gyeonggi-do, 410-380, Korea

Contact Anna Choi for Classes

LSTC 2017 Training

For Pricing Please visit www.lstc.com

September			
12-13	MI	Airbag Modeling	A. Nair
13	CA	Material Characteristics for Metals, Plastics, and Polymers - Test Data to Material Model	S. Bala
14-15	CA	Contact	S. Bala
October			
10-13	MI	Optimization and Probalistic Analysis using LS-OPT	A. Basudhar
16	MI	Intro to LS-PrePost	P. Ho / Q. Yan
17-20	MI	Intro to LS-DYNA	A. Nair
17-18	CA	NVH and Frequency Domain Analysis	Y. Huang
November			
6	CA	Intro to LS-PrePost	P. Ho / Q. Yan
7-10	CA	Intro to LS-DYNA	A. Nair
13-14	CA	LS-DYNA Advanced	S. Bala
Nov 30- Dec 1	CA	Advanced Metal Forming	L. Zhang / X.Zhu
December			
11	MI	Intro to LS-PrePost	P. Ho / Q. Yan
12-15	MI	Intro to LS-DYNA	A. Nair

LS-DYNA On Line

Visit the website for complete listings/changes/locations

www.LSDYNA-ONLINE.COM

For Information contact: courses@lsdyna-online.com or 513-331-9139

Composite Materials In LS-DYNA

This course will allow first time LS-DYNA users to use composite materials. The most important elements to start using all the composite material models in LS-DYNA will be presented in the 8 hours.

Foam & Viscoelastic Materials in LS-DYNA

Objective of the course: Learn about several foam material models in LS-DYNA to solve engineering problems. Detailed descriptions are given of the data required to use such material in analysis. Examples are used to illustrate the points made in the lectures

Plasticity, Plastics, and Viscoplastics Materials in LS-DYNA

Objective of the course: Learn about several plasticity based material models in LS-DYNA to solve engineering problems. Detailed descriptions are given of the data required to use such material in analysis. Examples are used to illustrate the points made in the lectures.

Rubber Materials in LS-DYNA

Objective of the course: Learn about several rubber material models in LS-DYNA to solve engineering problems. Detailed descriptions are given of the data required to use such material in analysis. Examples are used to illustrate the points made in the lectures.

FACEBOOK

BETA CAE Systems   CADFEM

ESI Group   Lenovo

TWITTER

BETA CAE Systems   ESI Group

ETA   CADFEM   Lenovo

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
Lenovo	

GOOGLE+

BETA CAE Systems	

LS-DYNA Resource Links

LS-DYNA Multiphysics YouTube Facundo Del Pin

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

FAQ LSTC Jim Day

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/tiki/tiki-index.php

Applications - Information for LS-DYNA

	LS-DYNA®, LS-OPT®, LS-PrePost, LS-TASC®, LSTC ATD and Barrier Models · 12 – 6 - 3 months/1 or 2 core license available · Students, Engineers. · NON-COMMERICAL USE For Information contact: sales@lstc.com
	**LS-Run – A standalone application -** a new graphical control center to start LS-DYNA simulations with either SMP or MPP - LS-Run has a parametric LS-DYNA command line builder making it easy to create the command and change the most common arguments such as ”memory”, ”ncpu” and the solver executable. For information contact: nik@dynamore.de
	A mobile & web application which is built to help LS-DYNA Users to get instant answers for technical query from global experts. For information contact: ramesh@kaizenat.com

LSTC Recent Developments, Features, Updates, News, Presentations

Editor: Yanhua Zhao

Conversion between FLD and Stress Triaxial Limit Curve
X. Zhu, L. Zhang, Y. Xiao
A non-orthogonal material model of woven composites in the preforming process -
W. Zhang, H. Ren, B. Liang, D. Zeng, X.Su, J. Dahl, M. Mirdamadi, Q. Zhao, J. Cao

Previously Presented: For a copy write to yanhua@feainformation.com

July
Best Fit GUI for Metal Forming in LS-PrePost® 4.5
Q. Yan, X. Zhu, P. Ho, L. Zhang, Y. Xiao - LSTC
Modeling and Numerical Simulation of Afterburning of Thermobaric Explosives In a Closed Chamber
KS Im, G. Cook, Jr., and ZC Zhang - LSTC
June
Improvement of Sandwich Structure Part Adaptivity in LS-DYNA
Xinhai Zhu, Houfu Fan, Li Zhang and Yuzhong Xiao - LSTC
New Inflator Models in LS-DYNA®
Kyoung-Su Im, Zeng-Chan Zhang, and Grant O. Cook, Jr. - LSTC
May
Improvement of Mesh Fusion in LS-DYNA
Houfu Fan, Xinhai Zhu, Li Zhang and Yuzhong Xiao - LSTC
Representative Volume Element (RVE) analysis using LS-DYNA
C.T. Wu, W. Hu LSTC
April
New features of 3D adaptivity in LS-DYNA -
W. Hu LSTC
New Feature: Defining Hardening Curve in LS-DYNA® -
X. Zhu, L. Zhang, Y. Xiao
March
Improvements toOne-Step Simulation in LS-DYNA
Xinhai Zhu, Houfu Fan, Li Zhang,
February
LS-DYNA Smooth Particle Galerkin (SPG) Method
C.T. Wu, Y. Guo, W. Hu - LSTC
January
Lancing features in LS-DYNA
Quanqing Yan, Li Zhang, Yuzhong Xiao, Xinhai Zhu, Philip Ho - LSTC

LSTC Recent Developments, Features, Updates, News, Presentations

2016

December

Thermal Coupling Method Between SPH Particles and Solid Elements
in LS-DYNA
Jingxiao Xu, Jason Wang, LSTC

November

Introduction to second order Lagrangian elements in LS-DYNA
Hailong Teng - Livermore Software Technology Corp.

October

*An Introduction to *CONSTRAINED_BEAM_IN_SOLID*
Hao Chen - Livermore Software Technology Corp

September:

Introduction to the new framework for User Subroutine Development of LS-DYNA
Zhidong Han and Brian Wainscott
*New Features in *ELEMENT_LANCING*
Xinhai Zhu, Li Zhang, Yuzhong Xiao

August :

Equivalent Radiated Power calculation with LS-DYNA
Yun Huang, Zhe Cui - Livermore Software Technology Corporation

July:

Recent Developments for Laminates and TSHELL Forming
Xinhai Zhu, Li Zhang, Yuzhong Xiao - LSTC

Conversion between FLD and Stress Triaxial Limit Curve

Xinhai Zhu, Li Zhang, and Yuzhong Xiao
LSTC

INTRODUCTION/MAIN FEATURES

Increasingly, as more Advanced High Strength Steels (AHSS) are being used, stamping engineers need to worry about material failure such as shear fracture during forming, in addition to the traditional necking failure. Two keywords are created in response to the users' requirement to account for the material failure modes from necking to fracture. They are:
*DEFINE_CURVE_FLD_FROM_TRIAXIAL_LIMIT, and
*DEFINE_CURVE_TRIAXIAL_LIMIT_FROM_FLD.

The FLD conversion from stress triaxial limit curve creates the corresponding necking failure limit curve when only a triaxial limit curve exists. The stress triaxial limit curve conversion from FLD curve generates the corresponding fracture limit curve when only a FLD limit curve is available.

Such conversions can be used in material models such as *MAT_037_NLP_FAILURE, *MAT_ADD_EROSION, or *MAT_260B, or in keywords such as *CONTROL_FORMING_ ONESTEP.

The conversion assumes plane stress and Von-Mises yield criterion. The converted FLD or stress triaxial curve can be found in the ".o" file (a scratch file from batch queue run).

EXAMPLES:

An example of the keyword *DEFINE_CURVE_FLD_FROM_TRIAXIAL_LIMIT is listed below. Note the abscissas represent stress triaxialities, typically ranging from -1/3 to 2/3; the ordinates represent equivalent plastic strains to fracture. The stress triaxial curve is referenced from the paper by *Li, Yaning et al, "Prediction of shear-induced fracture in sheet metal forming," Journals of Material Processing Technology, Volume 210, issue 14, (2010).*

```
*DEFINE_CURVE_FLD_FROM_TRIAXIAL_LIMIT
909
-.3284545, 2.485632
-.3193636, 2.327586
-.3102727, 2.198276
-.3011818, 2.04023
-.2875454, 1.882184
-.2739091, 1.70977
-.2602727, 1.522989
-.2466363, 1.37931
-.2239091, 1.235632
-.2011818, 1.106322
-.1648182, .9626437
-.133, .8477012
-8.754542E-02, .7471265
```

```
-4.663633E-02, .6896552
-1.481815E-02, .6465518
3.36367E-03, .632184
3.972731E-02, .6178162
8.518185E-02, .6034483
.1397273, .6034483
.1897273, .6465518
.2351819, .7183908
.2715455, .7758621
.3033637, .862069
.3306364, .9770116
.3488182, .9195403
.3715455, .8189656
.3988182, .7040231
.4351819, .5890805
.4715455, .5028736
.517, .4310345
.5533637, .4166667
.5760909, .4166667
.617, .4310345
.6397273, .4885058
.6533636, .5603449
.6670001, .8045977
*end
```

Figure 1 illustrates this conversion from stress triaxial curve to FLD curve. Note the red curve is the input triaxial curve and green curve is the output FLD curve from LS-DYNA. The LS-DYNA calculated FLD curve exactly matches that from the paper.

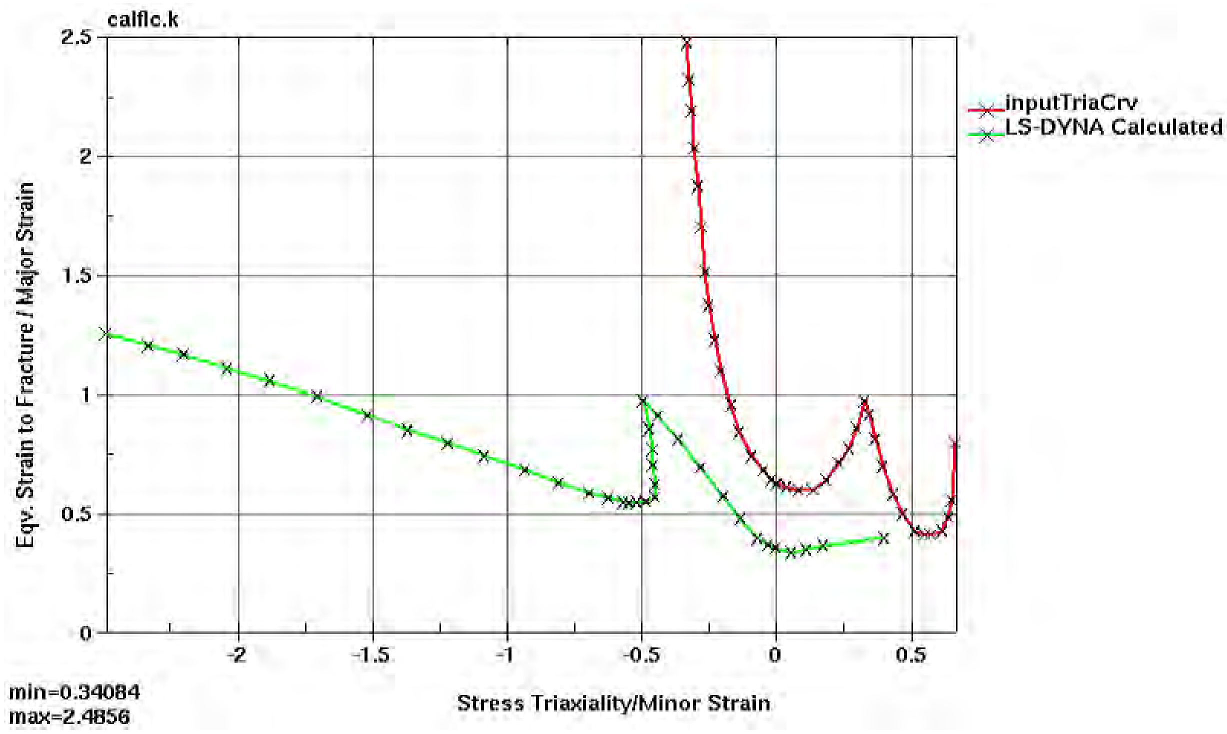


Figure 1 Conversion from Stress Triaxial Curve to FLD Curve

Another example of the keyword *DEFINE_ CURVE_TRIAXIAL_LIMIT_FROM_FLD is listed below. Note the abscissas represent minor true strains of a FLD curve; the ordinates represent major true strains of a FLD curve.

```
*DEFINE_CURVE_TRIAXIAL_LIMITFROM_FLD
909
-2.485543, 1.260929
-2.326915, 1.211872
-2.196562, 1.173440
-2.037161, 1.115458
-1.876514, 1.064689
-1.701195, 0.9987057
-1.511570, 0.9169926
-1.364909, 0.8546170
-1.215432, 0.8004163
-1.080362, 0.7465187
-0.9267878, 0.6888055
-0.8039308, 0.6348159
-0.6904781, 0.5923799
-0.6199200, 0.5716710
-0.5669760, 0.5526081
-0.5458882, 0.5490728
-0.5156967, 0.5524927
-0.4797515, 0.5568793
-0.4477362, 0.5742416
-0.4447934, 0.6287722
-0.4554417, 0.7088588
-0.4556653, 0.7716601
-0.4686948, 0.8609383
-0.4924086, 0.9770012
-0.4380930, 0.9192049
-0.3606666, 0.8170972
-0.2779830, 0.6991526
-0.1942268, 0.5787448
-0.1300228, 0.4857036
-6.8558961E-02, 0.4028141
-2.8324760E-02, 0.3741716
-1.5549607E-03, 0.3616189
5.8079619E-02, 0.3408428
0.1153265, 0.3534369
0.1781329, 0.3710328
0.4022586, 0.4023391
*end
```

Figure 2 shows the conversion from the FLD curve to stress triaxial limit curve. Note the red curve is the input FLD curve and green curve is the output stress triaxial limit curve from LS-DYNA. Again, the LS-DYNA calculated stress triaxial limit curve exactly matches that from the paper.

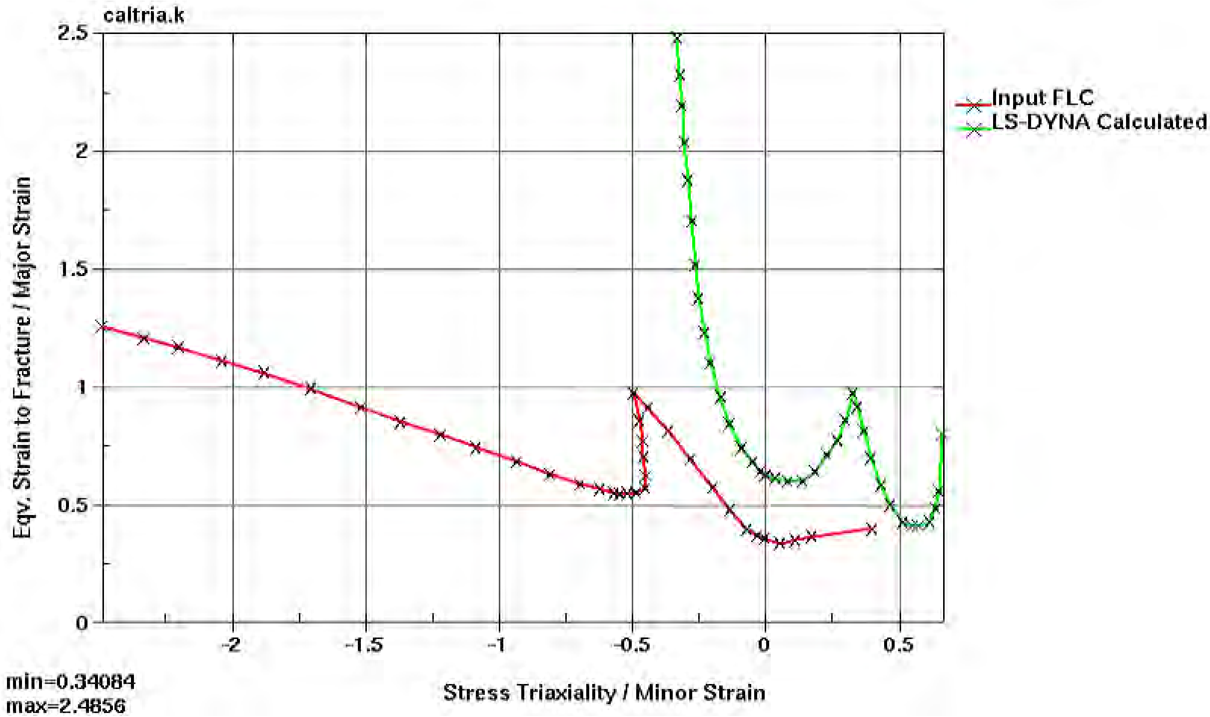


Figure 2 Conversion from FLD Curve to Stress Triaxial Limit Curve

REFERENCE:

1) LS-DYNA User's Manual (draft).
2) Li, Yaning et al, "Prediction of shear-induced fracture in sheet metal forming," *Journals of Material Processing Technology*, Volume 210, issue 14, (2010).

A non-orthogonal material model of woven composites in the preforming process

[a]Weizhao Zhang[#], [a]Huaqing Ren[#], [a]Biao Liang, [b]Danielle Zeng, [b]Xuming Su, [b]Jeffrey Dahl, [c]Mansour Mirdamadi, [d]Qiangsheng Zhao, [a]Jian Cao (1)

[a]*Department of Mechanical Engineering, Northwestern University, Evanston, IL, USA*

[b]*Ford Motor Company, Dearborn, MI, USA*

[c]*Dow Chemical Company, Midland, MI, USA*

[d]*Livermore Software Technology Corporation, Livermore, CA, USA*

[#]*Equal contribution*

1. Introduction

Woven carbon fiber reinforced plastics (CFRPs) have received growing attentions from transportation industry because of their high performance to weight ratio [1, 2]. Due to its good geometric conformability, woven CFRP is most suitable for complex part geometries. A highly-automated process chain consisting of preforming and curing process developed recently to manufacture the CFRP parts at low cost and high speed. Materials used in the first preforming step is the stacked flat layers of prepregs, which are woven CFRPs impregnated with uncured thermoset resin in desired fiber orientations. These layers are heated above the resin melting temperature to fully soften the prepreg and formed into the part shape on a press. The formed part is then cured to harden the resin for the permanent shape [3].

There exists ample design freedom in woven CFRP products in terms of parameters or options in material design and preforming processes. The large consumption of the test material and the extensive experimental trial out period could lead to high developing cost and long product development cycle. Numerical methods that can simulate the preforming process should be developed in order to solve this problem [4].

A non-orthogonal model for the woven CFRP preforming process is developed in this work, which has been incorporated into the LS-DYNA® software as MAT_293 (MAT_COMPRF) through the joint effort of this academic and industry team. Following in this paper is the detailed illustration of the fundamentals of this model and its experimental validation conducted at an industrial lab. Additionally, the measurement of interaction between prepreg layers is also characterized.

2. Analysis of the material deformation mechanism

Woven CFRPs are highly anisotropic in mechanical properties. The prepreg has large tensile modulus along the warp and weft yarn directions because of the stiff carbon fibers, but small intra-ply shear modulus, especially at the preforming temperature when the resin is molten as the shear resistance is mostly provided by the resin and the friction between the fiber yarns. During the preforming, the most dominant deformation mode is the intra-ply shear. To capture this mechanism, we propose to fully decouple the tension and shear deformation and the decoupling must hold well under large shear deformation.

Stress analysis for the woven CFRP with the modified non-orthogonal model is shown in Fig. 1. σ_{f1}, and σ_{f2} are the stress components caused by yarn stretch, and they are along the warp and weft yarn directions, respectively. σ_{m1} and σ_{m2} are the stress components caused by the yarn rotation. These stress components will be transformed into the local corotational coordinate, summed up as σ_{XX}, σ_{XY}, and σ_{YY}, and will be the stress outputs reported from the material model to the FEM software.

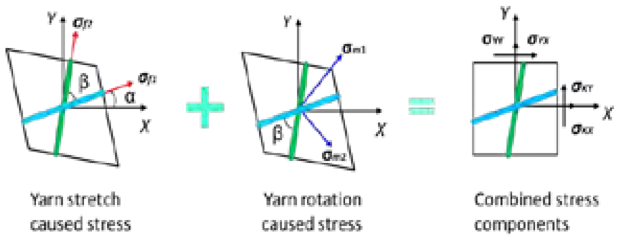


Fig. 1. Stress analysis of the woven CFRP with the modified non-orthogonal model.

The deformation gradient tensor $\boldsymbol{F}$ is utilized in this model to trace the yarn directions and stretch ratios during the preforming via $\boldsymbol{g}=\boldsymbol{F}\bullet\boldsymbol{G}$, where $\boldsymbol{g}$ and $\boldsymbol{G}$ are the final and initial orientations of the local fibers respectively. It can be used to calculate α, which indicates the relative rotation between the local warp direction and the *X*-direction in the local corotational coordinate, and yarn angle β, which indicates the amount of shear deformation in the material.

The model was implemented into the FEM software LS-DYNA® as MAT_293 (MAT_COMPRF). MAT_293 enables users to directly input experimental data to define the stress-strain curves, as well as the shear locking angle, which indicates whether the shear deformation reaches to the extent that the rotation resistance between warp and weft yarns is no longer small compared to the tensile modulus of the material. The flowchart of the model is shown in Fig. 2.

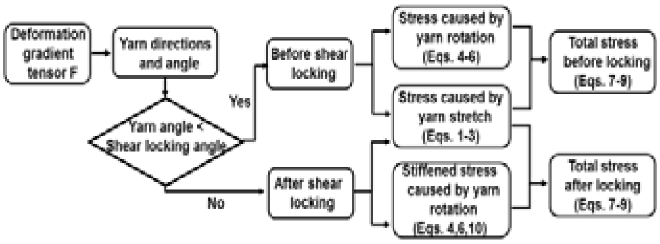
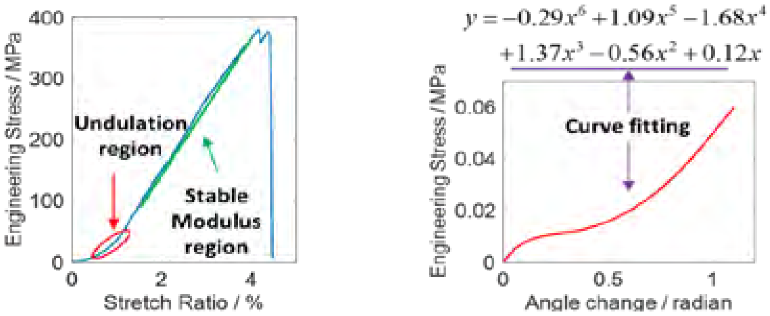


Fig. 2. Calculation flowchart of the LS-DYNA MAT_293.

In the material subroutine, the warp and weft directions for each element are calculated from the deformation gradient tensor. If the angle between the warp and weft yarns are smaller than the shear locking angle, then the small shear modulus condition will hold. If the angle between the warp and weft yarns reaches to the shear locking angle, the resistance for further shear deformation will greatly increase because the contacted fiber yarns stiffen the woven structure. In this situation, the shear resistance of the model will increase automatically to avoid further large shear deformation.

3. Material characterization

Material characterization is essential for the FEM model to predict the behavior of the woven CFRPs during the preforming process. It can be seen from Fig. 1 that the stresses caused by both yarn stretch and yarn rotation need to be calibrated for any specific woven material that is of interest. The calibration can be performed experimentally by the uniaxial tension and bias-extension tests [5]. The undulation strain and the stable tensile modulus along the yarn directions, as shown in Fig. 3(a), are obtained from the uniaxial tension test. In the FEM calculation, at every material point, once the stretch ratio along the yarns was obtained from the deformation gradient tensor, the resulting stress due to yarn stretch, i.e., "yarn stretch caused stress" σ_{f1} and σ_{f2}, can be obtained by referencing to the data in Fig. 3 (a). The shear behavior obtained via the bias-extension test, as shown in Fig. 3(b), is directly implemented as a polynomial function into the model to calculated the "yarn rotation caused stress" σ_{m1} and σ_{m2} given the angle change between the warp and weft yarns obtained from the deformation gradient tensor. The shear locking angle is also measured after the test and input to the model for the small/large shear moduli selection process shown in Fig. 2.

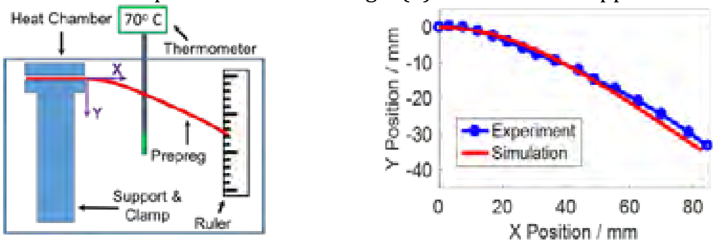


(a) (b)

Fig. 3. (a) Uniaxial tension test result for undulation and tensile modulus characterization and (b) bias-extension test result for shear deformation characterization

However, these two tests only provide the in-plane intra-ply properties of the material. During the preforming simulation, the bending behavior of the single layer and the interaction between the composite layers will also affect the in-plane strain distribution and the wrinkling initiation. Hence, characterizations of bending stiffness and inter-ply interaction are also necessary.

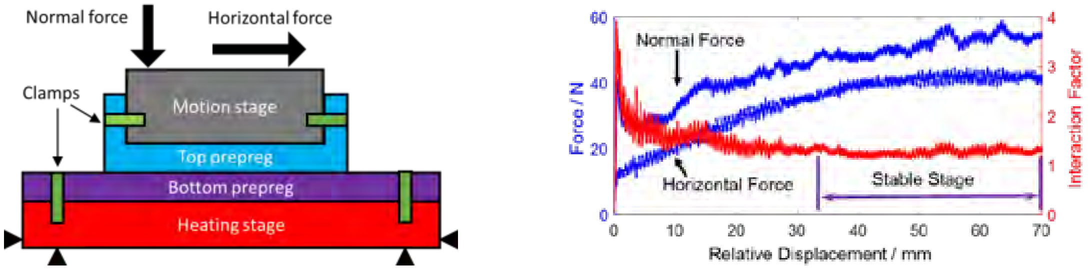
Bending stiffness of the composite is characterized with the cantilever beam system, as shown in Fig. 4(a). During the test, the single-ply prepreg will deform under gravity and the deflection is measured by a digital image analysis system. A bending test simulation model is utilized for parameter calibration. Material properties such as tensile modulus and composite density are inputs to the FEM model. Then the compressive modulus is adjusted until the same displacement in the Y direction of the end tip as that in the experiment is achieved. The final bending profile will be compared as shown in Fig. 4(b) to confirm the approach.



(a) (b)

Fig. 4. (a) Experimental setup, and (b) bending shape comparison at 70°C for bending stiffness characterization.

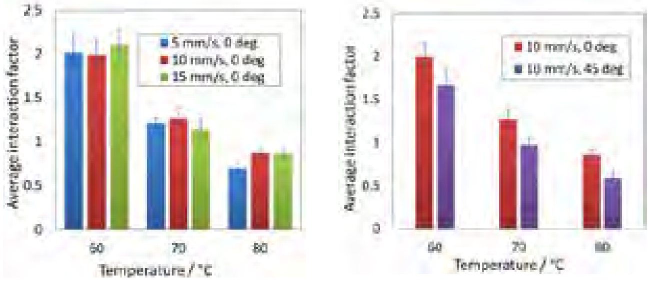
The setup for interaction characterization is demonstrated in Fig. 5(a). It moves two prepreg layers relatively to each other for the interaction characterization. A load cell records the normal and horizontal forces, whose ratio is defined as the interaction factor. One example of the histories of forces and the interaction factor in a test is shown in Fig. 5(b), while the stable value of the interaction factor is used in FEM. The preforming process operates at the temperature ranging from 60ºC to 80ºC, and the characterization results at different temperatures, sliding speeds, and fiber yarn orientations are shown in Fig. 6.



(a) **(b)**

Fig. 5. (a) Experimental setup for the interaction characterization, and (b) force and interaction factor history at 5 mm/s, 70ºC, 0 degree yarn angle difference

It can be seen in Fig. 6 that when the temperuture is fixed, the interaction factors at various slidng speeds and fiber orientations do not change significantly. For convenience, in one preforming simulation, assuming that the temperature distribution is uniform, the interaction factor will be treated as a constant.



(a) (b)

Fig. 6. Interaction factor at various temperatures subjected to different (a) relative motion speeds and (b) fiber yarn orientations.

4. Experimental validation

The double-dome test, as shown in Fig. 7, was conducted and simulated to demonstrate the capability of the material model for 3D shape forming regarding different yarn orientations and stacking sequences. The simulation model was established in LS-DYNA® using the explicit integration method. The sheet was modeled by reduced integrated shell elements. Each element is about 4 mm × 4 mm with five through-thickness integration points. The punch, binder and die were modeled by rigid shell elements.

Fig. 7. Experimental setup for the double dome test.

The simulation results in the upper-right quarter of Fig. 8 shows that the non-orthogonal material model established is capable of accurately predicting the physical experiments regarding the yarn angle distribution and blank draw-ins. For instance, the deviation of the maximum draw-in distance is about 7 mm (49 mm in experiment versus 42 mm in simulation). For comparison, an orthotropic material model (MAT_002) is utilized in another simulation whose result is shown in the upper-left quarter of Fig. 8 in the same scale as non-orthogonal model and experiment results. Since the orthotropic model cannot track the material property change during the yarns' rotation, the corresponding simulation has a maximum draw-in deviation of 24 mm, not capturing the overall process behavior.

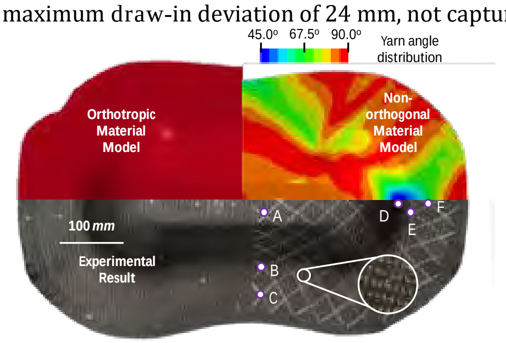


Fig. 8. Simulation and experimental results comparison of deformed geometry and yarn angle distribution for double dome preformed part of ±45º single layer woven prepreg.

In the non-orthogonal model, the yarn angle is defined as an output variable, while MAT_002 does not have the capability for direct visualization. For clarity, Table 1 compares the resulting shear angles at various locations obtained from the experiment and simulations. Again, it shows that the current model has improved the prediction accuracy.

Table 1 Resulting yarn angles from the single-layer case

Location	A	B	C	D	E	F
Experiment	80°	88°	71°	49°	56°	66°
Sim-orthotropic	70°	85°	86°	47°	59°	77°
Sim-present	81°	88°	73°	46°	60°	70°

The double-layer preforming test was conducted next. These two blanks were modeled by two layers of shell elements with different initial yarn orientations. The inter-ply interaction was simulated via the Coulomb friction model. The experimental and simulation results are shown in Fig. 9 from both top and bottom views, with both part geometry and wrinkle location marked. The simulation predicts the overall geometry and relative motion of two prepreg sheets well. The slight discrepancy might be caused by the constant interaction factor in the simulation, while in reality, it might be velocity- and pressure-dependent. Furthermore, the thickness distribution due to deformation will not be uniform, which may influence the frictional interaction and need to be captured by advanced shell elements or continuum elements.

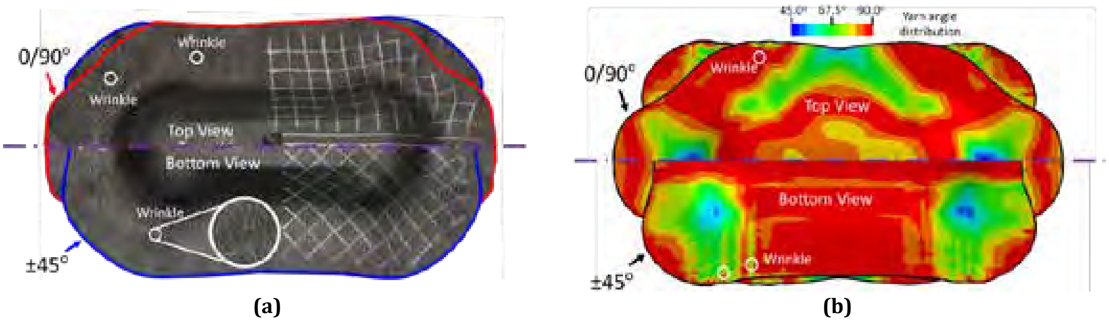


Fig. 9. Double dome preformed part of double-layer woven prepregs (a) experimental result, (b) simulation result.

5. Conclusions

An improved non-orthogonal material model that decouples the tension and shear behavior of the woven CFRPs under large shear deformation is proposed and implemented in the commercial finite element software LS-DYNA® for the composite preforming process. A systematic set of material characterization methods are developed to characterize both intra and inter-ply properties. With calibrated material properties, the material model can accurately predict the 3D deformation of the woven CFRP prepregs in preforming, demonstrated by the comparison between the simulation and experimental results from a double dome study.

Acknowledgements

This work was supported by the Office of Energy Efficiency and Renewable Energy (EERE), U.S. Department of Energy, under Award Number DE-EE0006867.

References

[1] Teti R. Machining of Composite Materials. CIRP Annals - Manufacturing Technology. 2002;51(2):611-34.

[2] Brinksmeier E, Janssen R. Drilling of Multi-Layer Composite Materials consisting of Carbon Fiber Reinforced Plastics (CFRP), Titanium and Aluminum Alloys. CIRP Annals - Manufacturing Technology. 2002;51(1):87-90.

[3] Yanagimoto J, Ikeuchi K. Sheet forming process of carbon fiber reinforced plastics for lightweight parts. CIRP Annals - Manufacturing Technology. 2012;61(1):247-50.

[4] Hamila N, Boisse P, Sabourin F, Brunet M. A semi-discrete shell finite element for textile composite reinforcement forming simulation. International Journal for Numerical Methods in Engineering. 2009;79(12):1443-66.

[5] Cao J, Akkerman R, Boisse P, Chen J, Cheng HS, de Graaf EF, Gorczyca JL, Harrison P, Hivet G, Launay J, Lee W, Liu L, Lomov SV, Long A, de Luycker E, Morestin F, Padvoiskis J, Peng XQ, Sherwood J, Stoilova T, Tao XM, Verpoest I, Willems A, Wiggers J, Yu TX, Zhu B. Characterization of mechanical behavior of woven fabrics: Experimental methods and benchmark results. Composites Part A: Applied Science and Manufacturing. 2008;39(6):1037-53.