FEA Information Engineering Solutions

Inside This Issue:

FEA Information Inc. is a publishing company founded April 2000, incorporated in the State of California July 2000, and first published October 2000. The initial publication, FEA Information News continues today as FEA Information Engineering Solutions. The publication's aim and scope is to continue publishing technical solutions and information, for the engineering community.

FEA Information Inc. Publishes:

FEA Information Engineering Solutions

FEA Information Engineering Journal

FEA Information China Engineering Solutions

FEA Information Engineering Solutions:

A monthly publication in pdf format sent via e-mail, additionally archived on the website FEA Publications. www.feapublications.com

FEA Information China Engineering Solutions

The first edition was published February 2012. It is published in Simplified and Traditional Chinese in pdf format. Published : February, April, June, August, October, December. The China Solutions is archived on the website FEA Publications. www.feapublications.com

To sign up for the Traditional, or Simplified edition write to yanhua@feainformation.com

FEA Information Engineering Journal: ISSN #2167-1273, first published February, 2012

Available on www.feaiej.com

To receive the journal, via e-mail, pdf format monthly contact:. editor@feaiej.com

Subject: Sign Up Journal

ISSN 2167-1273
Volume 1
Issue 3
April 2012

**FEA Information
Engineering Journal**

**FLUID STRUCTURE
INTERACTION**

.

Platinum Participants

www.beta-cae.gr

www.cray.com

www.datapointlabs.com

www.eta.com

www.esi-group.com

www.gns-mbh.com/

http://gridcore.se

www.hengstar.com /

www.jsol.co.jp/english/cae

www.lstc.com

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

TABLE OF CONTENTS

FEA Information Engineering Solutions

ANNOUNCEMENTS

For participation subscription including full page ads, articles, static listing, contact Anthony Giaccana, agiac99@aol.com

FEA Information Inc.

We are pleased to announce that we have received our ISSN from the Library of Congress for FEA Information Engineering Journal (FEAIEJ).

If you are interested in submitting for reprint, (authorization from the previous publisher is needed), or for original publication contact editor@feaiej.com.

FEAIEJ is an open access internet journal.   One time a year original papers, and those reprinted papers that we feel of technical excellence, will be published in a hardcopy journal.

We welcome unsolicited topics, ideas, and articles. Publishing is at the sole discretion of FEA Information Inc.

Technical Note:

On the conditioning of plastics prior to testing

Many commonly used plastics absorb water from the air with a consequential reduction in mechanical properties. The extent of absorption of water varies from material to material. It also increases with exposure time eventually reaching an equilibrium. In locales where the ambient humidity changes with the seasons, the moisture level in the plastic will fluctuate as water is absorbed or given off to maintain equilibrium. The mechanical properties will fluctuate correspondingly, weakening with increased moisture content.

With hydrophilic plastics such as nylon, the difference between the newly molded plastic, often called dry-as-molded (DAM) and that exposed to moisture is quite noticeable. While plastics like nylon, acetal and polyester are well known to be

hydrophilic, it is interesting to note that even hydrophobic plastics like olefins can show a decrease in modulus and strength after prolonged exposure to water. Clearly if the intent is to capture in-field performance, it is important to account for this behavior in FEA.


Figure. 1 Comparing stress-strain data of DAM and moisture conditioned nylon

Choosing the correct conditioning to apply to test specimens prior to testing is an important step. If the intent is to

6

simulate performance during or immediately after production, the DAM data is indicated. To simulate a more typical "in-field" performance, it is preferable to condition the test specimens so that they take up some moisture prior to testing. A common standards-based approach is to simply expose the test specimens at room temperature to a 50% relative humidity environment for 40 hours prior to testing. This ensures a standard moisture uptake which is based on the mass diffusion coefficient of the plastic. The test specimen typically does not reach an equilibrium moisture content but the amount of moisture uptake is the same for a given plastic.

For true "in-field" performance FEA, it may be desired to attain an equilibrated moisture level in the test specimens prior to testing. For this an understanding of the service environment is needed. For products that perform in a moisture stable environment, such as tropical or desert locales it is possible to declare the target moisture content. In locales where there is wide temporal variation in humidity level, this is not so easy and some sort of average or best case/worst case scenario must be defined. Test specimens are then subjected to moisture gain regimen to achieve the target moisture level prior to

testing. Different methods are available; the specimens can simply be left in the appropriate humidity environment until they equilibrate. This can take weeks or months. Alternatively, accelerated methods can be employed that use elevated temperature and high humidity to force moisture into the test specimens at a faster pace; the process can still take days to weeks. These methods are not standardized and their effectiveness for a particular product application must be proven by alternate testing. This adds considerable complexity and cost and the resulting practice is often specific to that particular application. Additionally, since plastics are visco-elastic in nature, the stress-relaxation effects due to long term elevated temperature exposures must also be kept in mind. In most cases, this would be beneficial; however where test specimens from finished parts are being conditioned, two unassociated factors of visco-elasticity and moisture level would be simultaneously modified as a consequence of accelerated environmental exposure.

Expansion of Conditioning Capabilities

DatapointLabs can perform environmental conditioning to a variety of protocols.

"Getting Started with LS-DYNA – Course Notes" by Morten Rikard Jensen,

Author – Morten Rikard Jensen

LSTC's "Getting Started with LS-DYNA®- Course Notes", will be available for purchase within two months. Pre-Order Reservation hold is available.

The book follows the basic getting started class to LS-DYNA® that users have been able to attend, as an on-site class at LSTC. Chapters will bring you through the slides, from the class, developed by Morten Rickard Jensen, Ph.D, of LSTC's support team, as well as one of the course instructors.

You will be able to view the slides of the two day class, inclusive of introductions to LS-DYNA® and LS-PrePost®

The course given by Dr. Jensen was developed specifically for first time users. Selected options are presented based on what is most often applied by users. In the printed book specific options were left out due to their complexity that would need class instruction.

The aim of the book is for the reader will be able to:

- Understand the flow of a LS-DYNA® simulation and get an overview of the process involved
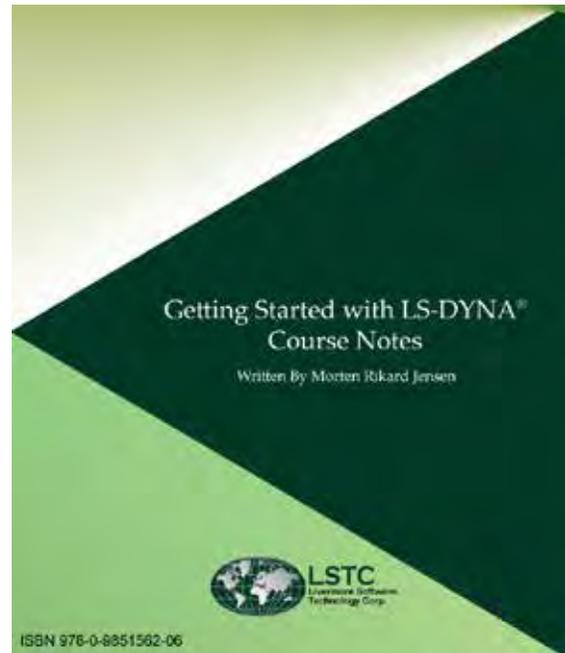
- Start LS-DYNA® using both Windows Manager and Linux Shell

- Build a small model in LS-PrePost®

- Add and edit keywords in LS-PrePost®

- Understand the structure of the Keyword User's Manual

Getting Started with LS-DYNA – Course Notes" by Morten Rikard Jensen

q Select a material model

q Specify contact, boundary conditions and loading etc.

q Post process results using LS-PrePost®

The book is divided into 18 different chapters covering main areas of using LS-DYNA®:

Acknowledgements and Preface
1. Introduction; LSTC and LS-DYNA®
2. The Keyword Input Deck and User's Manual
3. Overview of LS-PrePost® *EX 1 & 2*
4. Element Formulations *EX 3*
5. Choosing a Material Model *EX 4*
6. Applying Loads and Initial Conditions *EX 5*
7. Boundary Conditions *EX 6*
8. Specification of Contact *EX 7*
9. Control Cards
10. Output and Databases *EX 8*
11. Time Step and CPU time
12. Hourglass
13. Specifying Connections
14. Applying Damping
15. Assessment of FE Model
16. Summary
17. Additional Classes
18. References

Throughout the notes exercises are listed to emphasize specific options. The exercises sheets, the input files and the answer sheets are on the CD located in the back of the notes. Additional exercises and corresponding answer sheets can also be found on the CD.

To reserve a book and for pricing contact Julia Wolley, LSTC Documentation Manager jwolley@lstc.com.

"Getting Started with LS-DYNA® – Course Notes"
Author: Morten Rikard Jensen
LSTC publication ISBN: 978-0-9851562-0-6, 338 Pages

Linkedin LS-DYNA® Group By Morten Rikard Jensen

Author: Morten Rikard Jensen

LS-DYNA® group

Linked in

In 2008 the LS-DYNA® group was started on Linkedin, by Dr. M. R. Jensen of LSTC. The group has now grown to more than 1100 members from all over the world. The aim and scope of the group is to bring together members of the LS-DYNA® community. It has established a community to share information and knowledge about the LSTC products. The group members work various industries, where automotive engineering is the largest.

Additionally, it provides a forum for discussions on various topics related to using LS-DYNA®, new options, documentations, seminars etc. The group is not a general support forum. A general purpose forum for support is available on yahoo.

Jobs for LS-DYNA® engineers are frequently posted under the Job section and many connections have been made due to this feature.


It's easy to be a member, just sign up on Linkedin and search for LS-DYNA. We look forward to welcome you to the group.

Book Review – Formulas for Mechanical and Structural and Impact

Formulas for Mechanical and Structural Shock and Impact

Publication Date: October 15, 2009
by Gregory Szuladzinski
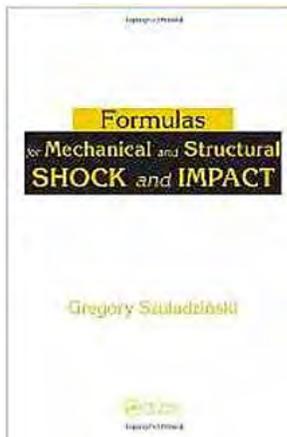
Preparations for physical testing or for large-scale calculations, during which a dynamic model is generated

Indirect verification of computer-generated results, to explain questionable results or guard against hidden errors

In dealing with extreme loads on structures, simple approximations of key variables can indicate if there is a threat of collapse. The ability to determine such variables early on strongly impacts the decisions about the engineering approach to adopt.

Formulas for Mechanical and Structural Shock and Impact is a self-contained and concise presentation of formulas and methodology you can use to determine dynamic response to shock loads, to help you decide on the optimal design. This book offers insight into how objects and structures respond to sudden, strong—and generally short—impulses. In our computer-oriented environment, in which structural programs are used for most large analytical tasks, engineers can still benefit from certain manual calculations and analytical methods to quickly assess the situation at hand.

Exploring a range of mechanical and civil engineering applications, the text enables engineers to manually calculate what happens to structures and objects when pushed, pulled, jerked, or blasted by providing ready access to formulas required for advanced problem solving.

It describes relatively simple methods of dealing with many design situations, in which simple spreadsheets or MathCad are sometimes employed. These scenarios may include:

Determination of preliminary figures on the anticipated dynamic response of a system that is in an early stage of design and for which a full-scale computation is not practical

Structural safety can be facilitated through the use of simple approximate solutions early in the design process, often eliminating the need for complicated and ore involved solutions later. This book is a valuable companion for modern engineers who need concise and relatively easy

methods of hand calculation to determine the essential variables. Without emphasizing any one particular type of structure, its scope is quite broad and applies to mechanical aspects of aeronautical, automotive, nuclear, and civil engineering, as well as those in general machine design. Stressing simplicity, the author presents the theoretical basis for manual calculations that will remain abundantly useful in the foreseeable future.

Recommended Reading – Reference Library
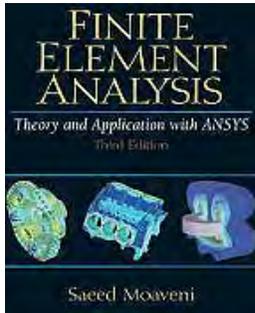
Finite Element Analysis Theory and Application with ANSYS (3rd Edition)

Saeed Moaveni

Practical Stress Analysis with Finite Element

Bryan J Mac Donald
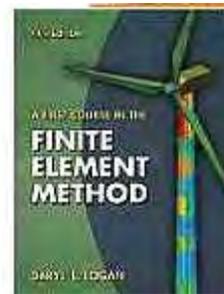
A First Course in the Finite Element Method

Daryl L. Logan
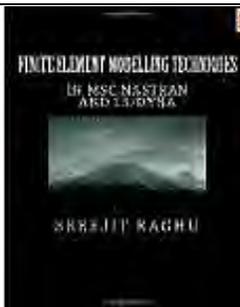
Finite Element Modelling Techniques in MSC.NASTRAN and LS/DYNA

Sreejit Raghu
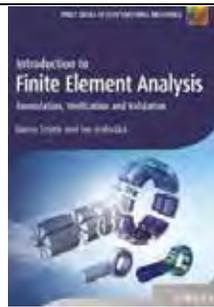
Finite Element Analysis/formulation & verification

B. A. Szabo
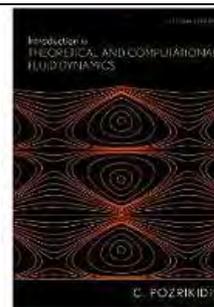
Introduction to Theoretical and Computational Fluid Dynamics

C. Pozrikidis

Recommended Reading – Reference Library
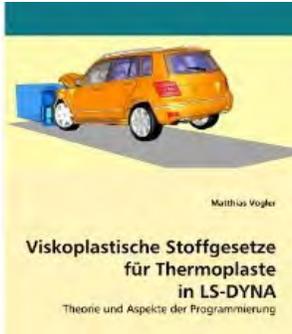
Viskoplastische Stoffgesetze für Thermoplaste in LS-DYNA: Theorie und Aspekte der Programmierung

Matthias Vogler
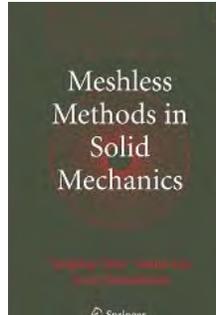
Meshless Methods in Solid Mechanics

Youping Chen
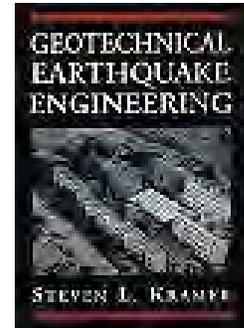
Geotechnical Earthquake Engineering

Steven Lawrence Kramer
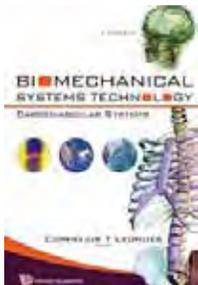
Biomechanical Systems Technology: Computational Methods

Cornelius T. Leondes

Numerical response of steel reinforced concrete slab subjected to blast and pressure loadings in LS-DYNA.

Vivek Re ddy

Recommended Reading – Reference Library
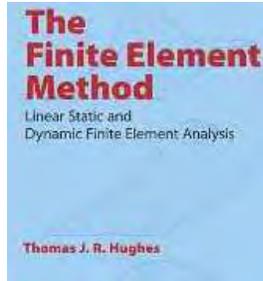
The Finite Element Method

Thomas J. R. Hughes
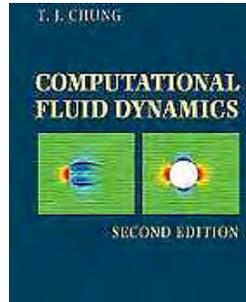
Computational Fluid Dynamics

T. J. Chung
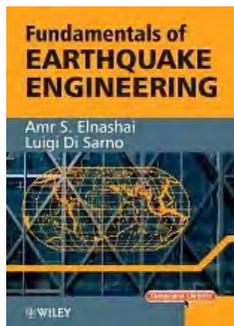
Fundamentals of Earthquake Engineering

Amr S. Elnashai
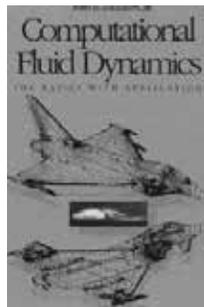
Computational Fluid Dynamics

John David Anderson
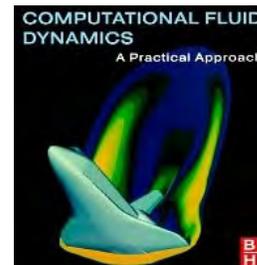
Computational Fluid Dynamics: A Practical Approach [Paperback]

Guan Heng Yeoh

BETA CAE Systems S.A.

BETA CAE Systems S.A.– ANSA

Is 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 S.A.– 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

The Cray XK6

The Cray XK6 supercomputer combines Cray's proven Gemini interconnect, AMD's leading multi-core scalar processors and NVIDIA's powerful many-core GPU processors to create a true, productive hybrid supercomputer

Cray XE6™ and Cray XE6m™ Supercomputers

The Cray XE6 scalable supercomputer is engineered to meet the demanding needs of capability-class HPC applications. The Cray XE6m is optimized to support scalable workloads in the midrange market.

Cray XMT™ System

The Cray XMT supercomputing system is a scalable massively multithreaded platform with a shared memory architecture for large-scale data analysis and data mining. The system is purpose-built for parallel applications that are dynamically changing, require random access to shared memory and typically do not run well on conventional systems.

Cray CX1000™ High(brid) Performance Computers

The Cray CX1000 series is a dense, power efficient and supremely powerful rack-mounted supercomputer featuring best-of-class technologies that can be mixed-and-matched in a single rack – creating a customized hybrid computing platform to meet a variety of scientific workloads.

Cray Sonexion 1300™ Storage System

The Cray Sonexion 1300 system is an integrated, high performance storage system that features next-generation modular technology to maximize the performance and capacity scaling capabilities of the Lustre file system.

Cray also offers custom and third-party storage and data management solutions

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.

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

The compary 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.

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.

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

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

Visual-Crash

Visual Crash for LS-DYNA helps engineers perform crash and safety simulations in the smoothest and fastest possible way by offering an intuitive windows-based graphical interface with customizable toolbars and complete session support. Being integrated in ESI Group's Open VTOS, an open collaborative multi-disciplinary engineering framework, Visual-Crash for DYNA allows users to focus and rely on high quality digital models from start to finish. Leveraging this state of the art environment, Visual Viewer, visualization and plotting solution, helps analyze LS-DYNA results within a single user interface.

vibro-acoustic software

With ESI's vibro-acoustic software you no longer have to account for noise and vibration right at the design stage - no more costly delays or panic driven test-based solutions. Our vibro-acoustic software has everything you need to diagnose potential noise and vibration problems up front in your development process. Manage risk by identifying possible problem areas that may need more detailed modeling or test based development, while you still have time to make an impact on the product!

VA One

VA One is a complete solution for simulating the response of vibro-acoustic systems across the full frequency range. VA One seamlessly combines Finite Elements (FE), Boundary Elements (BEM) and Statistical Energy Analysis (SEA) in ONE model. It is the only simulation code on the market today that contains the complete spectrum of vibro-acoustic analysis methods within ONE common environment.

GNS - Gesellschaft für Numerische Simulation mbH

www.gns-mbh.com

Animator4

A general finite element post-processor and holds a leading position in its field. Animator4 is used worldwide by almost all automotive companies, a great number of aerospace companies, and within the chemical industry.

Generator2.

A specialized pre-processor for crashworthiness applications and has become very successful in the field of passenger safety and pedestrian protection. It is mainly used as a positioning tool for finite element component models by a great number of automobile companies throughout the world.

Indeed

An easy-to-use, highly accurate virtual manufacturing software that specializes in the simulation of sheet metal forming processes. Indeed is part of the GNS software suite and works concurrently with all other GNS software products.

OpenForm

A pre- and post-processor independently of a particular finite element forming simulation package. The software is extremely easy to handle and can be used as was designed to enable those who are not finite element experts to carry out multi-stage forming simulations with even complex multi purpose finite element codes.

Gompute is owned, developed and operated by Gridcore AB in Sweden. Founded in 2002, Gridcore is active in three areas: Systems Integration, Research & Development and HPC as a service.

Gridcore has wide experience of different industries and applications, developed a stable product portfolio to simplify an engineer/scientist's use of computers, and has established a large network of partners and collaborations, where we together solve the most demanding computing tasks for our customers. Gridcore has offices in Gothenburg (Sweden), Stuttgart (Germany), Durham NC (USA) and sales operations in The Netherlands and Norway.

The Gridcore developed E-Gompute software for internal HPC resources gives end users (the engineers) an easy-to-use and complete environment when using HPC resources in their daily work, and enables collaboration, advanced application integrations, remote pre/post, accounting/billing of multiple teams, license tracking, and more, accelerating our customers usage of virtual prototyping.

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

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

Oasys LS-DYNA® Environment

The Oasys Suite of software, exclusively written for LS-DYNA®, is at the leading edge of the market and is used worldwide by many of the largest LS-DYNA® customers.

Oasys PRIMER

Oasys PRIMER is a model preparation tool that is fully compatible with the latest version of LS-DYNA®, eliminating the risk of data loss or corruption when a file is manipulated, no matter what operations are performed on it:

  Key benefits:

- Maintains data integrity
- Finds and fixes model errors (currently over 5000 checks)
- Specialist tools for dummy positioning, seatbelt fitting, mechanisms, interior head impact etc.
- Connection manager for spotwelds, bolts, adhesive etc.
- Intelligent editing, deletion and merging of data
- Customisable with macros and JavaScript.

Oasys D3PLOT

Oasys D3PLOT is a powerful 3D visualization package for post-processing LS-DYNA® analyses

  Key benefits:

- Fast, high quality graphics
- Easy, in-depth access to all LS-DYNA® results.
- User defined data components
- Customisable with JavaScript.

Oasys T/HIS

Oasys T/HIS is an X-Y graph plotting package for LS-DYNA®

  Key benefits:

1. Automatically reads all LS-DYNA® results.
2. Wide range of functions and injury criteria.
3. Easy handling of data from multiple models
4. Scriptable for automatic post-processing

Oasys REPORTER

Oasys REPORTER is an automatic report generation tool, for use with LS-DYNA®. which allows fast automatic report creation for analyses.

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, Hengstar Technology will continue to organize high level training courses and seminars in 2012.

The lectures/training are taught by senior engineers and experts mainly from LSTC, Carhs, OEMs, and other consulting groups.

On Site Training

Hengstar also provides customer customized training programs on-site at the company facility. Training is tailored for company needs using LS-DYNA or the additional software products by LSTC.

Distribution & Support

Hengstar Distributes and supports LS-DYNA, LS-OPT, LS-PrePost, LS-TaSC. Hongsheng Lu, previously was directly employed by LSTC before opening his distributorship in China for LSTC software. He travels to LSTC often to keep current on the latest software features and support to continue to grow Hengstar as a CAE consulting group.

Participant Products

ANSA	**TestPaks®**	**LS-PrePost**	**CRAY XE6m™**
Inventium Suite™	**CRAY XK6**	**HYCRASH**	**LS-TaSC™**
Animator 4	**TOYOTA THUMS**	**CRAY XE6™**	**PreSys**
CRAY CX1000™	**Visual Crash**	**Generator 2**	**VPG**
JMAG	**VA One**	**DYNAFORM**	**LSTC Dummy Models**
LS-DYNA	**CRAY Sonexion 1300™**	**Indeed**	**Vibro-Acoustic Software**
OpenForm		**Oasys Primer**	
JSTAMP/NV	**GOMPUTE Cloud**		**LSTC Barrier Models**
Oasys D3Plot	**μETA**		**Oasys T/HIS**

North America Distribution & Consulting			

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		

United States	**CAE Associates Inc.** www.caeai.com	info@caeai.com		
	ANSYS Products	CivilFem	Consulting ANSYS	
			Consulting LS-DYNA	

United States	**DYNAMAX** www.dynamax-inc.com	sales@dynamax-inc.com		
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	LSTC Dummy Models		LSTC Barrier Models	

	North America Distribution & Consulting		

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

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

United States	**Gompute** www.gompute.com	info@gompute.com	
	LS-DYNA Cloud Service	Additional software	
	Additional Services		

North America

Distribution & Consulting

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	**Predictive Engineering** www.predictiveengineering.com	george.laird@predictiveengineering.com

FEMAP   NX Nastran   LS-DYNA   LS-OPT

LS-PrePost   LS-TaSC   LSTC Dummy Models

                                        LSTC Barrier Models

France	**Alliance Svce. Plus - AS+**		v.lapoujade@asplus.fr	
	www.asplus.fr/ls-dyna			
	LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
	DYNAFORM	VPG	MEDINA	
	LSTC Dummy Models		LSTC Barrier Models	

France	**ALYOTECH**		nima.edjtemai@alyotech.fr	
	www.alyotech.fr			
	ANSYS	LS-DYNA	MOLDEX3D	FEMZIP
	Primer	PreSys	DYNAFORM	SKYGEN
	MERCUDA	MOCEM		

Germany	**CADFEM GmbH**		lsdyna@cadfem.de	
	www.cadfem.de			
	ANSYS	LS-DYNA	optiSLang	DIGIMAT
	ESAComp	AnyBody	VPS	
	FTI FormingSuite			

Germany	**DYNAmore GmbH**		uli.franz@dynamore.de	
	www.dynamore.de			
	ANSYS	LS-DYNA	optiSLang	DIGIMAT
	LS-OPT	LS-PrePost	LS-TaSC	DYNAFORM
	Primer	D-Spex	GENESIS	FEMZIP
	VisualDoc		LSTC Dummy & Barrier Models TOYOTA THUMS	

Germany	**GNS**		mbox@gns-mbh.com	
	www.gns-mbh.com			
	Animator	Generator	Indeed	OpenForm

Netherland	**Infinte**		j.mathijssen@infinite.nl	
	www.infinite.nl			
	ANSYS Products	CivilFem	CFX	Fluent
	LS-DYNA	LS-PrePost	LS-OPT	LS-TaSC

Italy	**EnginSoft SpA**	info@enginsoft.it	
	www.enginsoft.it		
	ANSYS	MAGMA	Flowmaster	FORGE
	CADfix	LS-DYNA	Dynaform	Sculptor
	ESAComp	AnyBody	FTI Software	
	AdvantEdge	Straus7	LMS Virtual.Lab	ModeFRONTIER

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

Sweden	**DYNAmore Nordic**	marcus.redhe@dynamore.se	
	www.dynamore.se		
	ANSA	μETA	LS-DYNA	LS-OPT
	LS-PrePost	LS-TaSC	FastFORM	DYNAform
	FormingSuite		LSTC Dummy Models	
			LSTC Barrier Models	

Sweden	**GRIDCORE**	info@gridcore.com	
	www.gridcore.se		
	LS-DYNA Cloud Service		Additional software	
			Additional Services	

Europe

Distribution – Consulting

Switzerland	**DYNAmoreSwiss GmbH**	info@dynamore.ch	
	www.dynamore.ch		
	LS-DYNA	LS-OPT	LS-PrePost
	LS-TaSC	LSTC Dummy Models	
		LSTC Barrier Models	

UK	**Ove Arup & Partners**		dyna.sales@arup.com	
	www.oasys-software.com/dyna			
	LS-DYNA		LS-OPT	LS-PrePost
	LS-TaSC	PRIMER	D3PLOT	T/HIS
	REPORTER	SHELL	FEMZIP	HYCRASH
	DIGIMAT	Simpleware	LSTC Dummy Models	
			LSTC Barrier Models	

Asia Pacific
Distribution – Consulting

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**   Stephen.zhao@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	
LS-DYNA Courses	LS-OPT	LSTC Dummy Models	LS-PrePost

Asia Pacific				
Distribution – Consulting				

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   **EASI Engineering**   rvenkate@easi.com

www.easi.com

ANSA

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

India   **CADFEM Eng. Svce**   info@cadfem.in

www.cadfem.in

ANSYS	VPS	optiSLang	ESAComp	DIGIMAT
LS-DYNA	LS-OPT	LSTC Dummy Models	LS-PrePost	
FTI FormingSuite	AnyBody	LSTC Barrier Models	LS-TaSC	

Asia Pacific
Distribution – Consulting

Japan   **ITOCHU**   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

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

Japan   **FUJITSU**

http://jp.fujitsu.com/solutions/hpc/app/lsdyna

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

<table>
<tr><td></td><td colspan="4" align="center">Asia Pacific

Distribution – Consulting</td></tr>
</table>

Korea	**THEME**	wschung@kornet.com		
	www.lsdyna.co.kr			
	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	

Asia Pacific
Distribution – Consulting

Taiwan   **Flotrend**   gary@flotrend.tw

www.flotrend.com.tw

LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM

Taiwan   **APIC**

www.apic.com.tw

LS-DYNA	LS-OPT	LS-PrePost	LS-TaSC
LSTC Dummy Models	LSTC Barrier Models	eta/VPG	FCM

Cloud Services including LS-DYNA		
Japan	Fujitsu	www.fujitsu.com
Germany	Gridcore	www.gridcore.se
Sweden	Gridcore	www.gridcore.se
United States	Gompute	www.gompute.com

The Complete Courses Offered Can Be Found At: www.cadfem.de
Please check the site for accuracy and changes. Among the many course offered:

Introduction to explicit structural mechanics with ANSYS-LS-DYNA and LSTC's LS-DYNA
    05/09/12
    08/29/12   09/05/12
    11/06/12   12/19/12

Material Modeling with LS-DYNA
    10/16/12

Simulation of composites with ANSYS Composites PrepPost and LS-DYNA
    05/08/12   08/21/12

Contact modeling with LS-DYNA
    05/22/12   11/06/12

Modeling joints with LS-DYNA
    10/12/12

Crash simulation with LS-DYNA
    09/25/12

Introduction to simulation with Diffpack
    11/06/12

Working efficiently with Diffpack in ANSYS Workbench
    11/07/12

Introduction to simulation of joint- and muscle-forces with AnyBody
    09/19/12

Efficient coupling of AnyBody with ANSYS Workbench
    09/21/12

Additional Courses are offered – please check the website for upcoming dates for: FTI Forming Suite - DIGIMAT DIFFPACK and others.

Individual Training: Take advantage of the expertise of our specialists and get to know how simulation processes in your company can be arranged in an optimal way.

The Complete Courses Offered Can Be Found At: www.dynamore.de/en

Intro LS-DYNA   Spot Welds
09/20/12   10/15/12 10/30/12   12/10/12   09/27/12

Contact Definitions   Dummy Modeling
    10/18/12   06/14/12

Element types   Airbag Modeling
    07/04/12   06/14/12

Plasticity   eta/DYNAFORM
    10/24/12   09/17/12

Users Interfaces   ALE
    11/19/12   10/11/12

Crash Analysis   Meshless Methods
    12/04/12   10/11/12

The Complete Courses Offered Can Be Found At: www.lstc.com

Please check the site for accuracy and changes.
Among the many course offering are the following:

Advanced Impact Using LS-DYNA MI
>   April 24-26, 2012

Introduction to LS-DYNA CA
>   May 1-4, 2012

Introduction to LS-PrePost (no charge) CA
>   April 30, 2012

The Complete Courses Offered Can Be Found At: www.dynamore.se

Please check the site for accuracy and changes.
Among the many course offering are the following:

LS-PrePost 3, introduction
         September 3

LS-PrePost 3, introduction
         November 26

LS-DYNA, introductory course   September 4

LS-DYNA, introductory course   November 27

         LS-OPT, optimization and robust
         design September 18

LS-DYNA, implicit analysis
         October 2

ANSA & Metapost, introductory course
         October 9

Contacts in LS-DYNA
         October 12

LS-DYNA, simulation of sheet metal forming processes
         October 16

LS-DYNA, advanced training class in impact analysis
         November 20

The complete Training Courses offered can be found at www.asplus.fr/ls-dyna
Please check the site for accuracy and changes.

LS-DYNA Introduction Explicit Solver
   10-12/09

LS-DYNA Introduction Implicit Solver
   24/09

LS-DYNA Unified Introduction Implicit &
Explicit Solver
   16-19/01, 18-21/06 & 12-15/11

LS-OPT & LS-TaSC Introduction
   24-25/10

Switch to LS-DYNA
   10-11/10

Switch from Ls-PrePost 2.X to 3.X
   4/04 & 26/09 & 28/11

LS-DYNA Advanced Implicit Solver
   25/09

LS-DYNA ALE / FSI
22-23/10

LS-DYNA SPH
   21-22/05 & 8-9/10

LS-PrePost 3.0 – Advanced meshing
capabilities
   5/04 & 27/09 & 29/11

LS-DYNA User Options
   23-24/05

LS-DYNA – Plasticity, Damage & Failure –
By Paul DU BOIS
   26-27/11
   (date may be changed in Q1)

LS-DYNA – Polymeric materials – By Paul
DU BOIS
   12-13/12

LS-DYNA – Geo-material modeling
   14-15/12

The Complete Courses Offered Can Be Found At: www.eta.com

Please check the site for accuracy and changes.
Among the many course offering are the following:

Introduction to DYNAFORM

        05/01-02/2012

Introduction to PreSys

        05/08/2012

Introduction to LS-DYNA

        05/15-16/2012

The Complete Courses Offered Can Be Found At: www.caeai.com

Please check the site for accuracy and changes. Among the many course offering are the following:

ANSYS Training, CFD and FEA Consultants Serving CT, NJ, NY, MA, NH , VT

May 14, 2012 1 day
ANSYS Workbench Meshing for CFD
Middlebury, CT

May 15, 2012 2 days
Introduction to CFX
Middlebury, CT

Jun 11, 2012 1 day
ANSYS DesignModeler
Middlebury, CT

Jun 16, 2012 2 days
Introduction to ANSYS Mechanical
(Workbench)
Middlebury, CT

PAUL DU BOIS & LEN SCHWER

Announce their annual LS-DYNA® training class schedule

May 2012 - Somerset Inn - Troy Michigan

These popular LS-DYNA training classes have been offered since 2006, in North America and Europe. The classes continuously evolve, reflecting the ongoing additions and improvements in LS-DYNA's simulation capabilities.

Paul du Bois and Len Schwer, Co-Instructors:

Penetration Modeling with LS-DYNA   May 1 - 2 (Tuesday – Wednesday)

Blast Modeling with LS-DYNA   May 3 - 4 (Thursday – Friday)

Modeling and Simulation with LS-DYNA   May 7 - 8 (Monday - Tuesday)

Len Schwer, Instructor:

Concrete & Geomaterial Modeling   May 9 - 10 (Wednesday - Thursday)

Class descriptions available on the LSTC web site: **www.lstc.com/training**

To request Registration Form, please send email to: **schwerdubois@yahoo.com**

Training Classes
China
Hengstar Technology

The Complete Courses Offered Can Be Found at www.hengstar.com

2012	2	3	4	5	6	7	8	9	10	11	12
An Introduction to LS-DYNA(High Level)											
Concrete & Geomaterial Modeling with LS-DYNA	■										
Pedestrian Safety and Bonnet Design with LS-DYNA			■	■							
Crashworthiness Theory and Technology			■	■							
LS-DYNA MPP, Airbag Simulation with LS-DYNA					■	■					
Introduction of LS-OPT which is Based on LS-DYNA						■	■				
Passive Safety and Restraint Systems Design								■	■		
Crashworthiness Simulation with LS-DYNA					■	■	■				
Passive Safety Simulation with LS-DYNA										■	■
Crashworthy Car Body Development - Design, Simulation and Optimization								■	■		

For course location visit www.alyotech.fr

LS-DYNA Introduction

   June   04-06

   Sept   10-12

   Oct   01-03

   Nov   12-14

   Dec   03-05

LS-DYNA Thermal

   Sept   13-14

LS-DYNA Implicit

   May   21-23

   Sept   17-19

   Nov   19-21

LS-PrePost – Meshing

   May   24

   Sept 27

   Nov   26

LS-PrePost – New Interface

   May   25

   Sept   28

   Nov   27

LS-OPT Introduction

   June   18-19

   Dec   10-11

LS-TaSC – Topology Optimization

   June   20

   Dec   12

Material Modeling & User Defined

Material in LS-DYNA

   July 10-11

Crash & Impact Modeling

   April 02-05

FSI & ALE in LS-DYNA

   March 15-16

LS-DYNA Composite

   July 12-13

Gompute User Group Meeting (7th Edition) May 08 & 09, 2012

May 08 & 09, 2012
www.simdi.se /

Location:
Scandic Crown Hotel
Gothenburg (Sweden)

About the Gompute User Meeting
The Gompute User Meeting, formerly called SIMDI (Simulation and Data intensive Fair), is an event that gathers different actors of the Simulation Industry

Primarly GUM11 gathers users of Gompute® software suite and the Gompute on Demand™ service but it is open to any other person interested in the application of numerical simulation on their daily work.

This year meeting is the sixth one all of them has been celebrated in Gothenburg Sweden.

Gompute on Demand delivers HPC on demand for technical and scientific applications, it is owned and operated by Gridcore AB, a Swedish System integrator specialized in technical and scientific computing.

You may get more information about gompute and Gridcore at www.gompute.com and www.gridcore.se .

GUM11's Agenda Manager
Iago Fernandez
Phone: +46 704 19 39 76
Mail: iago.fernadez@se.gompute.com

GUM11's Event Manager
Anne-Marie Ohlsson
 Phone: +46 31 18 21 60
Mail: amo@gridcore.se

June 03-05, 2012

Conference website: www.ls-dynaconferences.com

12^{th,} Int'l LS-DYNA Users Conference
   June 3rd, 4th & 5th
      Training 6th & 7th

Conference Plenary/Keynote Addresses
   June 4th

Professor Thomas J.R. Hughes
   The University of Texas

Professor David J. Benson
   University of California

Mr. Kenji Takada
   Honda Japan

Mr. Paul A. Du Bois
   Consulting

FEA Information Participants
 Sponsoring and/or exhibiting:

- ARUP
- CRAY
- BETA CAE Systems, S.A.
- DatapointLabs
- ESI North America
- ETA
- Gompute Inc.
- GNS mbH
- D3View
- FEA Information
- JSOL

Hotel - Venue - Reservations
   Hyatt Regency Dearborn
   Fairlane Town Center, Dearborn, MI 48126
Please inform reservations that you're
 attending the LS-DYNA Conference

For booth, sponsorships or any conference questions contact: info@lst.com

TRAINING OFFERED:

- Pre- Conference LS-PrePost
- Pre- Conference   User Material Dev. In LS-DYNA
- Advanced ALE Applications
- ALE/Eulerian & FSI
- Blast & Penetration Using LS-DYNA
- Heat Transfer with Warm   Forming and Hot Stamping Applications
- Introduction to LS-OPT
- NVH & Frequency Domain Analysis w/ LS-DYNA
- Occupant Restraint: Dummies/Barriers
- Polymeric Material With LS-DYNA
- Probabilistic Design Using LS-OPT & LS-DYNA
- SPH & Element-Free Galerkin

June 25th 2012

www.openfoamworkshop.org/2012/OFW7.html

7th OpenFOAM® Workshop

The 7th OpenFOAM® Workshop will be held in Darmstadt (next to Frankfurt, Germany) from June 25-28, 2012 - hosted by the Center of Smart Interfaces (CSI) and the Graduate School of Computational Engineering (GSCE) of the Technische Universität Darmstadt.

The OpenFOAM® Workshop intends to bring together OpenFOAM®'s developers and users, to promote collaborative activities, exchange information and share experiences in similar areas of interest. The 7th OpenFOAM® Workshop is

· a community forum and open discussion platform for OpenFOAM® users and developers,

· the largest OpenFOAM® community event of the world (expected 400 participants),
· since 2006: sustainable, community-driven and non-profit.

The 7th OpenFOAM® Workshop is organized using the Community Portal of the Extend Project.

Please visit for abstract submission, registration and progam schedule.

Your OpenFOAM®-Workshop Organizing Committee

committee@openfoamworkshop.org

October 09, 2012

www.dynamore.de/en/training/conferences/upcoming/ls-dyna-forum-2012/ls-dyna-update-forum-2011

German LS-DYNA Forum 2012

LS-DYNA Forum, 9 - 10 October 2012, Ulm, Germany

On the 9th and 10th October 2012, our 11th LS-DYNA Forum will be taking place at the Maritim Hotel in Ulm, Germany. We cordially invite you not only to attend the event but submit a paper. In your presentation, you can talk about your experiences with LS-DYNA or LS-OPT and you can discuss and exchange these experiences with other users.

User presentations will form the core of the event. General lectures given by renowned speakers are also planned as well as talks on the latest LS-DYNA und LS-OPT .

Comprehensive information all about LS-DYNA software can be obtained from the accompanying exhibition.

The Forum will be accompanied by seminars which will be held during the week of the conference on the subjects of CPM Airbag OoP, ALE and fluid-structure inter-action, meshless methods and on concrete and geomaterial modeling.

Your presentation: You are cordially invited to contribute towards the program plan by submitting a paper. Contributions from the various areas of application of LS-DYNA/LS-OPT are planned

To Submit your papers: Please send us the title, authors and a short summary (approx. 300 words).

Dates: Submission of proposed paper:
25th May 2012
Author notification: 11th June 2012
Submission of two-page summary for proceedings: 7th Sept. 2012

Location: Maritim Hotel Ulm
          Basteistraße 40, 89073 Ulm

Registration and contact
DYNAmore GmbH
Tel. +49 (0) 7 11 - 45 96 00 - 0
Fax +49 (0) 7 11 - 45 96 00 - 29
E-Mail: forum@dynamore.de
www.dynamore.de/forum12

October 24-26, 2012

Location: Kassel Germany

Environmental protection and economic aspects make electric mobility one of the great challenges of the coming years. Step-by-step it will replace traditional forms of mobility in everyday life. Therefore, a number of projects have been defined in so-called 'model regions' in order to better understand and optimize this process.

For a better understanding of electric mobility and its optimization, simulation specialist ANSYS has extended its portfolio with a set of simulation applications that can serve as models in the development and implementation of innovative drive concepts. Structural and fluid mechanics and electromagnetic simulation models of the individual components are modeled in a consistent environment both individually and interacting, considering the

drive as a complete multi-physical system – Engineering the System!

The ANSYS Conference & the 30th CADFEM Users' Meeting focus on the many simulation options in electric mobility and several other current application fields where structural mechanics, fluid mechanics and electro magnetics issues are important.

CADFEM GmbH and ANSYS Germany GmbH cordially invite you to join the conference

We look forward to your participation

The CADFEM & ANSYS Germany Team

Events
Guest Postings

Conference - Event

2012 SIMULIA Customer Conference

www.3ds.com/company/events/scc-2012/overview

Nastran Users Meeting

http://pages.mscsoftware.com/NastranUserMeeting2012-Home.html

AMD Fusion12 Developer Summit

www.amd.com

Seventh M.I.T. Conference

www.seventhmitconference.org

Part 2

Automotive Composites: Modeling Resin Transfer Molding (RTM) using LS-DYNA

Art Shapiro, LSTC, shapiro@lstc.com

Hao Chen, LSTC, hao@lstc.com

Resin Transfer Molding (RTM) is a closed mold process for making composite materials. The manufacturing process consists of:

1. Laying a fiber (e.g., glass, carbon) mat inside a mold cavity
2. Injecting a resin to fill the voids in the fiber mat
3. Creating an environment for the resin to cure

Part 1 (January issue of FEA Information News) in this series of reports presented an ALE methodology to model resin filling a mold cavity containing an orthotropic fabric. In this 2nd report, the methodology is extended to cover

1. Specifying an inlet pressure instead of an inlet velocity
2. Defining essential boundary conditions on the external surface of the ALE domain
3. Modeling heat transfer in the flowing resin and between the resin and mold

How to specify an inlet pressure

Inlet pressure is created by defining an ALE ambient element with a non-zero initial relative volume, $v_r = \rho_0/\rho$, on the EOS keyword. The Gruneisen equation of state is

$$P = \frac{\rho_0 C^2 \mu \left[1 + \left(1 - \gamma_0/2\right)\mu - \frac{a}{2}\mu^2\right]}{\left[1 - (S_1 - 1) - S_2\frac{\mu^2}{(\mu + 1)} - S_3\frac{\mu^3}{(\mu + 1)^2}\right]^2}$$

Set all *EOS_GRUNEISEN input parameters equal to zero (except for the sound speed C) to model an incompressible fluid near atmospheric pressure.

This then simplifies to $\qquad P = \rho_0 C^2 \mu$

Where, by definition $\qquad \mu = \rho/\rho_0 - 1$

Therefore $\qquad P = \rho_0 C^2 \mu = \rho_0 C^2(\rho/\rho_0 - 1) = \rho_0 C^2(1/v_r - 1)$

Solving for relative volume $\qquad v_r = 1 \Big/ \left(\frac{P}{\rho_0 C^2} + 1\right)$

Simple test problem → straight_pipe_pressure.k
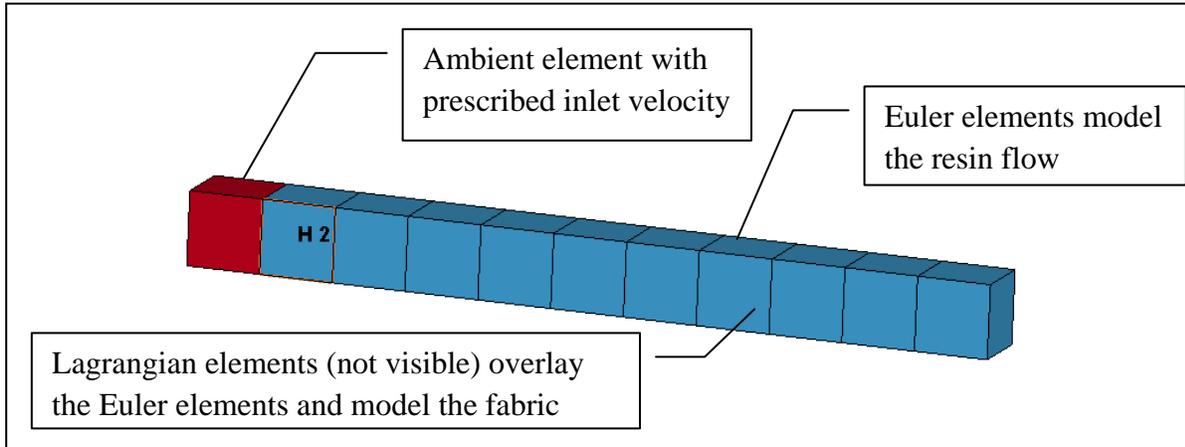
Shown above is the test problem, square_pipe_velocity.k, discussed in Part 1 of this report. The objective was to calculate the pressure drop over the length of the rod due to a porous media flow with a specified inlet velocity (v = 0.0005 m/s). The pressure drop was calculated as 1.e+05 Pa. Now we will specify the pressure and calculate the velocity and mass flow rate. The initial relative volume to create an inlet pressure of 1.e+05 Pa is

$$v_r = \frac{1}{\frac{P}{\rho_0 C^2} + 1} = \frac{1}{\frac{100000}{(1000)(150)^2} + 1} = 0.9955752$$

This value is entered as parameter V0 on card 2 of the EOS_GRUNEISEN keyword. Velocity vectors at the final time when the rod is completely filled with resin is shown below. Remember that we artificially scaled the velocity and sound speed to increase the computational time step as explained in detail in Part 1 of this report. Therefore, the scaled sound speed of C=150 is used in the above equation and the calculated velocity, v=0.5 m/s, is 1000 times greater than real life, v=0.0005 m/s. The mass flow rate is calculated by

$$\dot{m} = \rho A v = (1000)(0.01)^2(0.0005) = 5 * 10^{-5} \text{ kg/s}$$
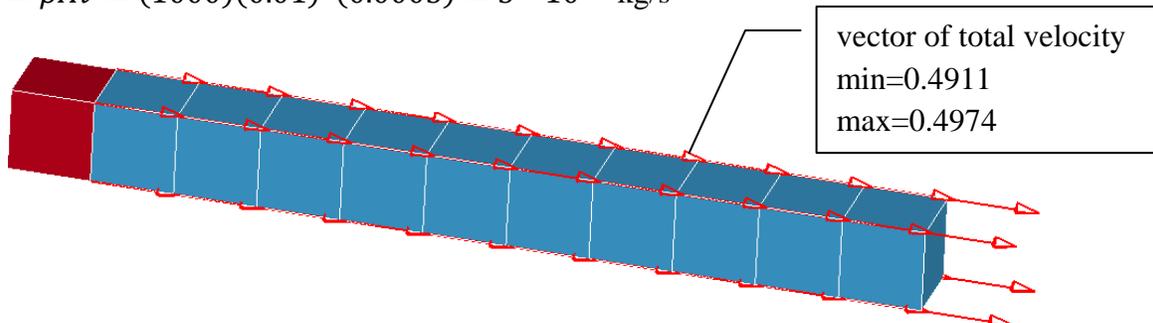
ALE_ESSENTIAL_BOUNDARY keyword

In RTM models, the flow needs to be constrained over the Euler mesh surface. Otherwise, the flow will exit the domain as shown by the velocity vectors in the left hand figure below. In a properly constrained model, the flow will remain inside the domain as shown in the right hand figure.
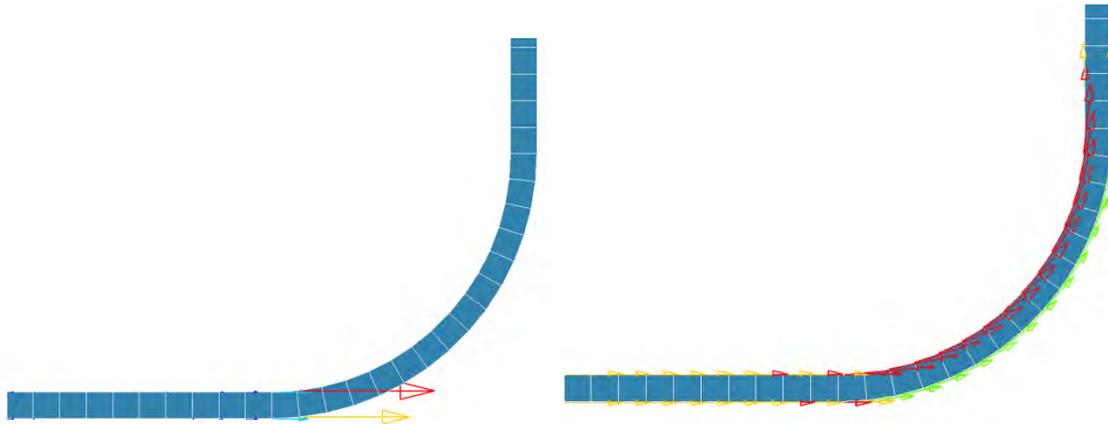

Defining SPC boundary conditions with a local coordinate system at each boundary node could do the job but it is cumbersome and time consuming. This keyword applies the flow condition (e.g., stick, or slip) along the ALE Euler mesh surface automatically. For complicated ALE mesh boundary geometries, instead of using part/part set definitions, users should generate multiple segment sets with each segment set describing a separate piece of the ALE boundary geometry. In the top-hat geometry shown, 4 segment sets are required to define the surface constraints: (1) top surface, (2) bottom surface, (3) x-z symmetry plane, and (4) y-z symmetry plane. Segments sets for the inlet and outlet are not defined.

Thermal Analysis → straight_pipe_thermal.k

The resin enters the mold at a high temperature so that the resin doesn't begin to cure before it fills the mold. The thermal model includes specifying a resin inlet temperature and a heat transfer coefficient between the resin and mold. The geometry for this problem is shown below. Several

of the mold elements have been deleted to reveal the ALE Euler mesh modeling the resin flow channel.
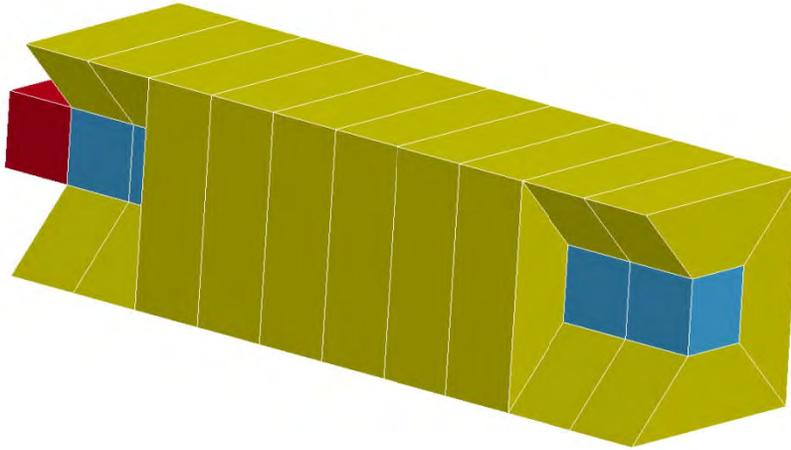
The thermal problem specification is hypothetical so that we can compare the answer with an analytical solution.

1. $T_i = 1.$ resin inlet temperature
2. $T_w = 0.$ mold wall temperature held fixed
3. $l = 0.1$ mold length
4. $w = 0.01$ mold width
5. $h = 0.01$ mold height
6. $P = 4(0.01)$ mold flow perimeter
7. $h = 0.01$ Heat transfer coefficient between flowing resin and mold, $h = 0.01$
8. $\rho = 2.$ resin density
9. $c = 1.$ resin heat capacity
10. $k = 1.e-06$ thermal conductivity of resin is set to a small value to eliminate resin conduction heat transfer. Therefore,
11. Inlet velocity v=(scale factor)(velocity)=(1000)(0.0005)=0.5
12. Mass flow rate

A mathematical description of this problem is

$$h(P\,\delta x)(T_w - T)$$

$$\dot{m}cT \qquad \dot{m}c\left(T + \frac{dT}{dx}\delta x\right)$$
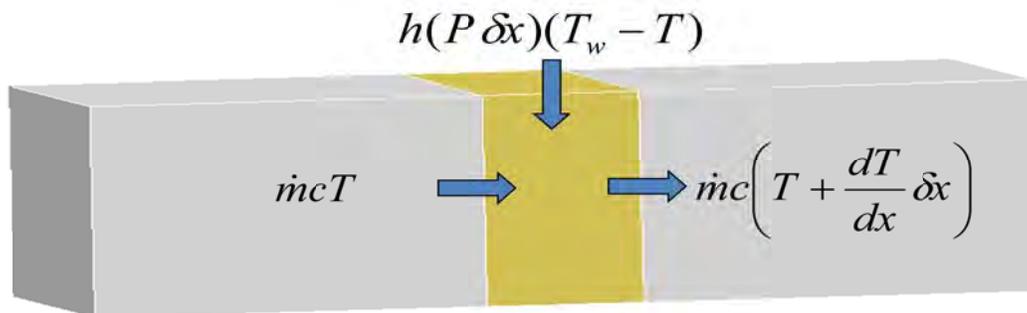
Note that the conduction terms $(k * d^2T/dx^2)$ do not show up because they are considered negligible based on our assumption of a very small value for the resin thermal conductivity, k=1.e-06. Performing a macro energy balance where (energy in) = (energy out) we obtain

$$\dot{m}c\left(T + \frac{dT}{dx}\right)\delta x = \dot{m}cT + h(P\delta x)(T_w - T)$$

Rearranging

$$\frac{dT}{dx} + \frac{hP}{\dot{m}c}T = \frac{hP}{\dot{m}c}T_w$$

Integrating between x=0 and x=l, and using the boundary condition T=Ti at x=0, we obtain

$$T_l = T_w + (T_i - T_w)exp\left(-\frac{hPl}{\dot{m}c}\right) = T_i exp\left(-\frac{hPl}{\dot{m}c}\right)$$

$$T = 1 * exp\left(-\frac{(0.01)(4*0.01)(0.1)}{(0.0001)(1)}\right) = 0.670$$

LS-DYNA results are shown below

specified inlet T = 1

calculated exit T = 0.677
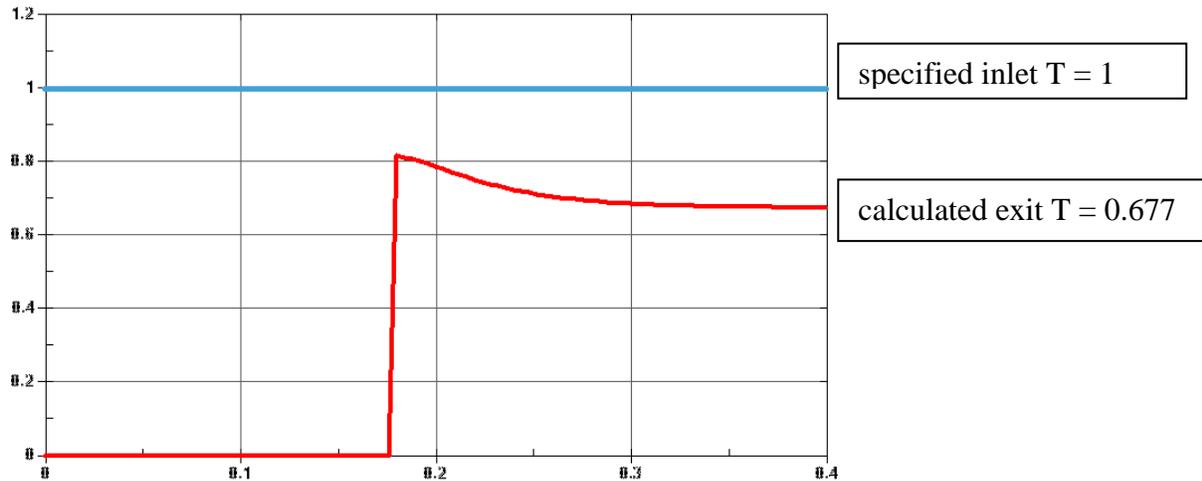
LSTC releases LS-TaSC™ Version 2.1

Willem Roux

Livermore Software Technology Corporation, Livermore, CA, USA

Abstract

LSTC just released LS-TaSC™ version 2.1, a topology optimization tool using LS-DYNA® for the analysis of nonlinear structural behavior. This paper focuses is on its capabilities, current development directions, and integration into an industrial design environment. Examples of using the new developments such as dynamic load scaling are given.

1   Overview

The goal of topology optimization is to find the shape of a structure with the best use of the material. An alternate view of the process is that of selecting the best load path for the specified use of the structure. For dynamic problems like crashworthiness simulations, this is achieved by designing for a uniform internal energy density in the structure while keeping the mass constrained.

The overall LS-TaSC [1] process consists of (i) the design problem definition, (ii) performing the design optimization iteratively using LS-DYNA [2], and (iii) post-processing the results.

The topology design problem is defined by (i) the allowable geometric domain, (ii) how the part will be used, and (iii) properties of the part such as manufacturing constraints. Additionally, you have to specify methodology requirements such as termination criteria and management of the LS-DYNA$^{®}$ evaluations.

The initial parts specify the design domain – the optimum parts computed will be inside the boundaries delimited by the initial parts. The parts must be modeled using solid or shell elements.

The part may contain holes; a structured mesh is accordingly not required and there is no node or element numbering convention as in other approaches. Geometry constraints such as being an extrusion or a casting direction may be specified.

The use of the part is described by LS-DYNA input deck. The design process aims for a uniform internal energy density in the structure as computed by LS-DYNA using this input deck.

The final shape of the part is described by the subset of the initial elements used. The use of an element is controlled by changing the amount of material in the element. This is achieved by assigning a design variable to the density of each element. The material is parameterized using a so-called *density approach.* In this approach, a design variable is directly linked to the individual material element such that each cell has its own material model. The design variable *x,* also known as relative density, varies from 0 to 1 where 0 indicates void and 1 represents the full material. The material properties corresponding to the values of design variables are obtained using an appropriate interpolation model as described in the manual [1].

2   Methodology

The typical goal of topology optimization is to obtain maximum utility of the material. Obtaining uniform internal energy density in the structure is used as the objective for optimization. This follows the formulation proposed by Patel [3], with the resulting implementation being similar to the fully-stressed design and uniform strain energy density approaches (Haftka and Gurdal [4], Patnaik and Hopkins [5]).

The optimization problem is formulated as,

$$\min_{x} \sum_{i=}^{N} \sum_{j=1}^{L} \left(w_j U_j (x_i) - U_j^* \right),$$

$$\text{subject to} \sum_{i=1}^{N} \rho (x_i) V_i \leq M^*$$

$$C_j^l \leq C_j \leq C_j^u, \quad j = 1, 2, \ldots J.$$

$$x_{\min} \leq x_i \leq 1.0.$$

where U represents the internal energy density of the ith element, V_i is the volume of ith element, U^* represents internal energy density set point, and C_j is the jth constraint. There are L load cases with a total of J constraints. The superscripts 'l' and 'u' represent lower and upper bounds on the constraints, respectively.

The change in the design variable of ith variable (Δx_i) is computed as,

$$\Delta x_i^t = K \left(U_i^t - U^* \right) / U^*.$$

where K is a scaling factor and U^* denotes the internal energy density set point. The design variable is updated as,

$$x_i^{t+1} = x_i^t + \Delta x_i^t.$$

3   Overview of current capabilities

This product is for the topology optimization of non-linear problems involving dynamic loads and contact conditions. It can be used to find a concept design for most structures analyzed using LS-DYNA.

3.1   General capabilities

- Solid design using first-order hexahedrons, tetrahedral, and pentahedral elements
- Shell thickness design using first-order quadrilateral and triangular elements
- Global constraints
- Multiple load cases, including dynamic load case weighing
- Tight integration with LS-DYNA®
- Large models with millions of elements

3.2   Geometry definitions

- Multiple parts
- Extrusions
- Symmetry
- Casting, one sided
- Casting, two sided
- Forging

3.3   Postprocessing

- Design histories
- LS-PREPOST plots of the geometry evolution and the final design

4   New capabilities in version 2.1

The following new capabilities were added to create version 2.1:

- · ***Dynamic load case weighing*** Dynamic load case weighing is used to ensure that a part performs equally well for all design scenarios.

- · ***Forging geometry definitions*** This geometry definition is set to obtain a part that can be manufactured using a forging process.

- · ***Minor features:***

    - · Castings can have interior holes.
    - · Pentahedral elements are supported.
    - · The memory footprint is reduced more than a factor of 2 and an option is provided which can be set to reduce memory use by a further factor of 2.
    - · *MAT_ELASTIC is supported for the design part.
    - · Lightly used elements can be kept instead of deleted.
    - · The SIMP algorithm can be switched on and off.
    - · Coordinate systems are no longer limited to DIR=X.
    - · Restarting was improved to be faster by using more archived results.
    - · A fringe plot of the material utilization as considered in the design process can be viewed.
    - · The fraction of the original number of elements used in the design can be viewed as a history.

5 Dynamic load case weighing

It can happen that a single load case dominates the topology of the final design making the structure perform badly for the other load cases. This can be resolved by assigning different weights to the load cases, but it is difficult to know good weighing values in advance. Dynamic weighing of the load cases is used to select the load case weights based on the responses of the structure as the design evolves, thereby resulting in a design that performs well for all load cases.

The dynamic weighing is done by defining a desired relationship between the responses of all the load cases. The algorithm will scale the load case weights to achieve this relationship. Say we have constraint C_1 from the first load case and constraint C_2 from the second load case, then we write our desired behavior as $k_1 C_1 + offset_1 = k_2 C_2 + offset$ with C the constraint value, k a scale factor, and an offset added. The weight w_i of load case i is adjusted to change constraint C_i. The target value is computed as

$$C_{t\arg et} = \frac{\sum_{i}^{i=n} k_i C_i + offset_i}{n}$$ from which we compute $\Delta w_i = (C_{t\arg et} - k_i C_i - offset_i) / \P C_i / \P w_i$ with

the derivative approximated as ±1 and a maximum bound is place on Δw to ensure convergence in a reasonable number of iterations.

The final weights found are not suitable for restarting. They can be examined though for an indication of good values of the weights, but usually the final weights found using dynamic weighing are too large.

6 Examples

6.1 Forging example

This example is a solid part to be manufactured as a forging, which was accordingly imposed as a forging geometry definition including a minimum web thickness. The geometry and loading conditions

for this component are shown in Figure 1. The FE model has about 60 000 elements and a single linear implicit load case as shown was considered.
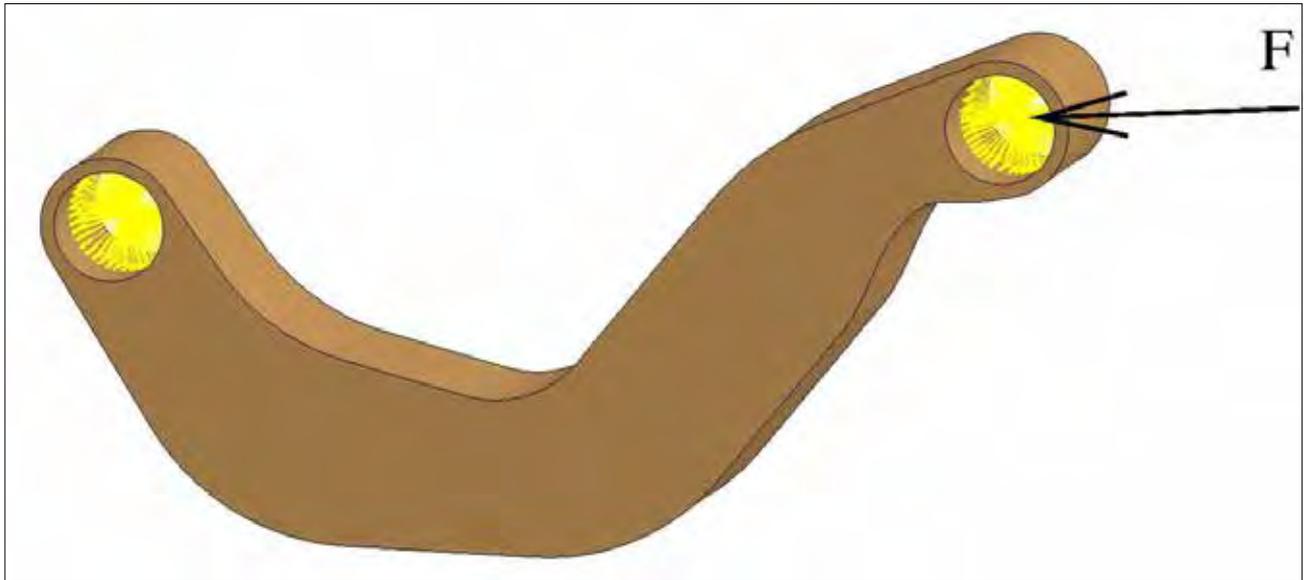

Figure 1: The initial geometry and loading conditions.

The final design is shown in Figure 2. Note the web that is required for forging manufacturing. Flanges and a rib were created to carry the bending load efficiently.
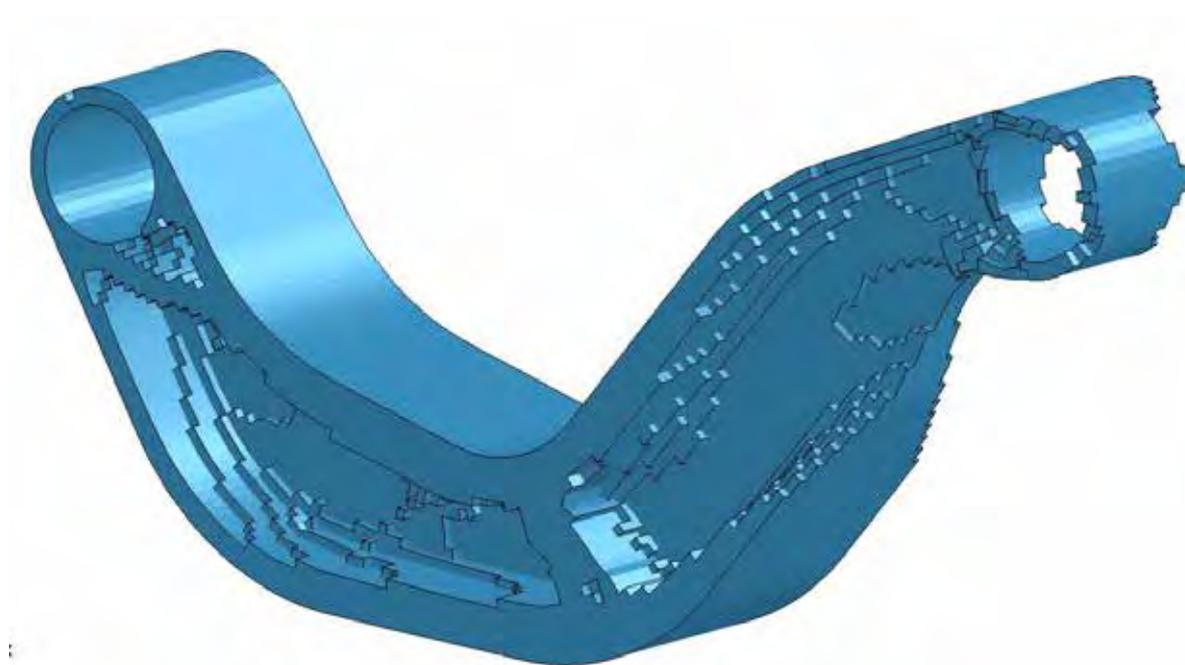

Figure 2 Design with forging geometry definition. Note the web that is required for forging.

6.2   Dynamic load scaling

This is a nonlinear structure compressed by an impactor in two load cases as shown in Figure 3. A symmetry geometry definition requiring symmetry around the center was imposed to remove the need for a right load additional to the shown left load. The center load case dominates the geometry of the final design if the load cases are not scaled – without load case scaling the design is much stiffer for

the center load than for the left load. It is difficult to know up front which values of the load case weights to use in order to have a balanced design. In this example it is shown how dynamic load scaling creates a balanced design.
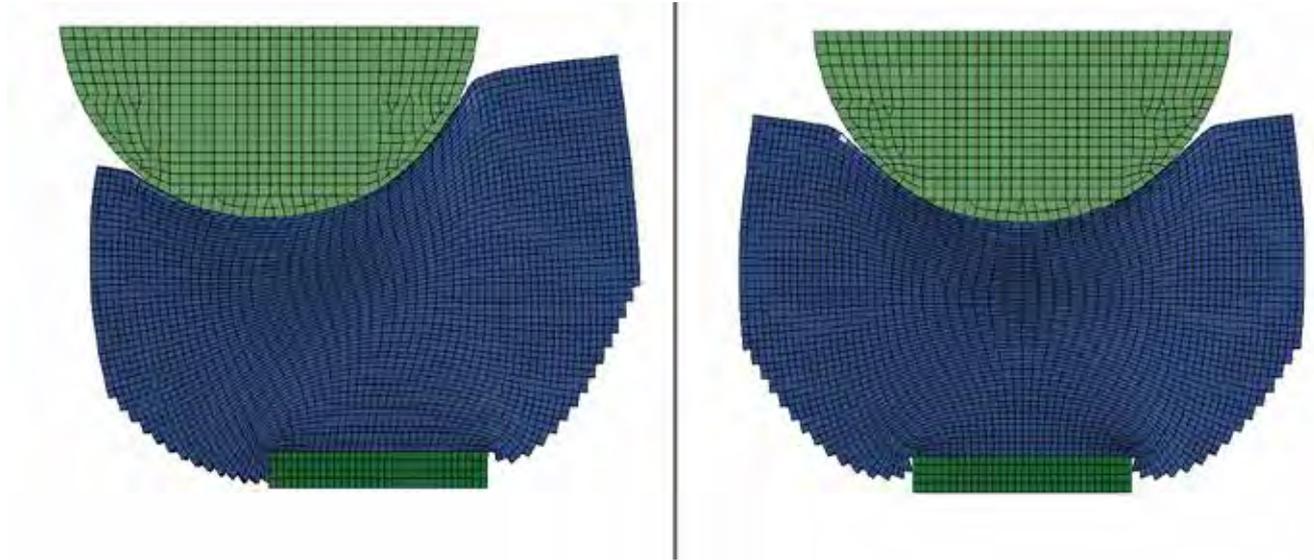

Figure 3 The geometry and loading conditions of the dynamic loading example. A left load, a center load, and symmetry around the center are applied.

In the following pictures we show the results from a standard study with both load case having equal weights, and a study with dynamic load balancing. In Figure 4 to the left we show the reaction forces for the standard design study in which the load cases were weighed equally. The dynamic load balancing was then set to have the final results for these two reaction force to be the same for both load cases, which allowed us to achieve the reaction force curves as shown to the right of Figure 4 for the balanced design.
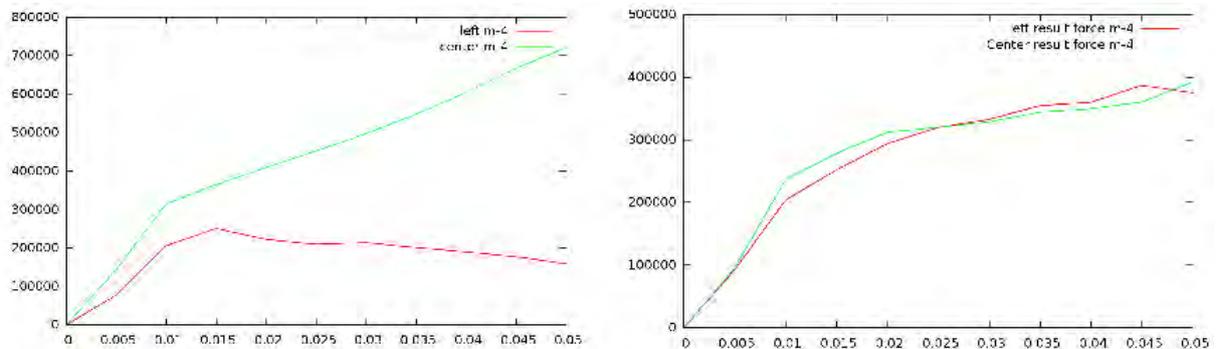

Figure 4: The reaction forces for the two load cases. To the left are the results from the original problem statement, from which it can be seen that the reaction forces differs greatly for the two load cases. To the right are the results with dynamic load scaling, from which it can be be seen that the reaction forces are now similar.

The designs obtained have different topologies as shown in Figure 5.
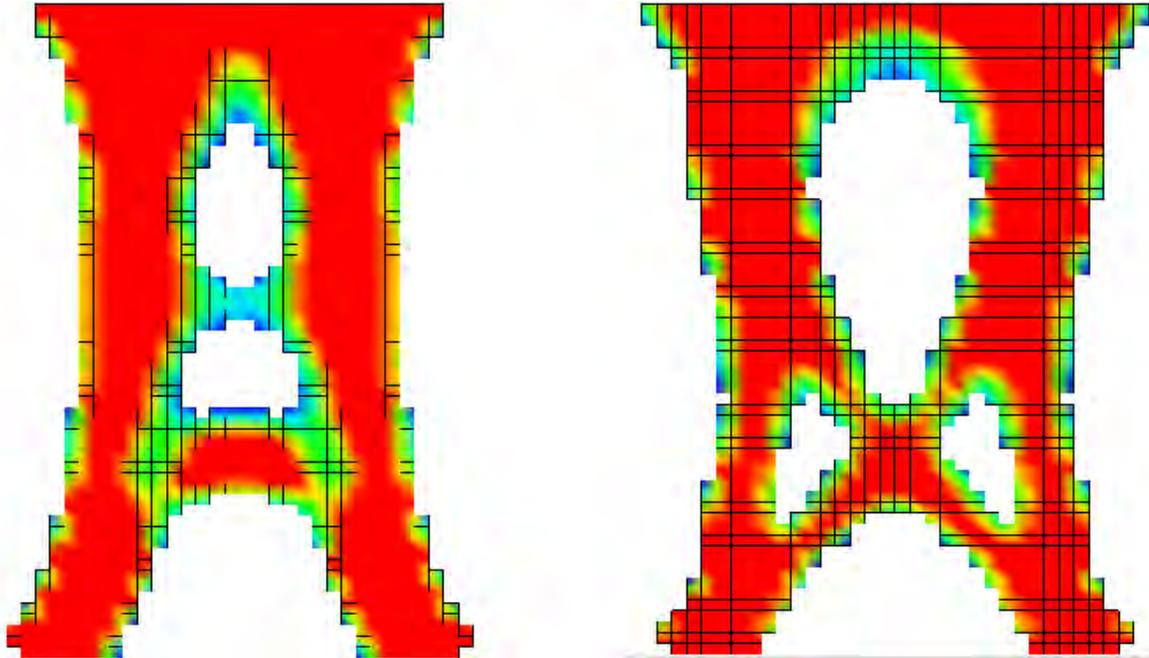
Figure 5 Final designs. The design using equal weighing is shown on the left, while the design using dynamic weighing is shown on the right. Note the bottom of the designs differ: the dynamic scaling has the pillars connected using a truss structure in order to provide more support for the offset load.

The convergence of the maximum reaction force value for both load cases is shown in Figure 6. The final weights are 0.12 for the center load case and 7.2 for the right load case.
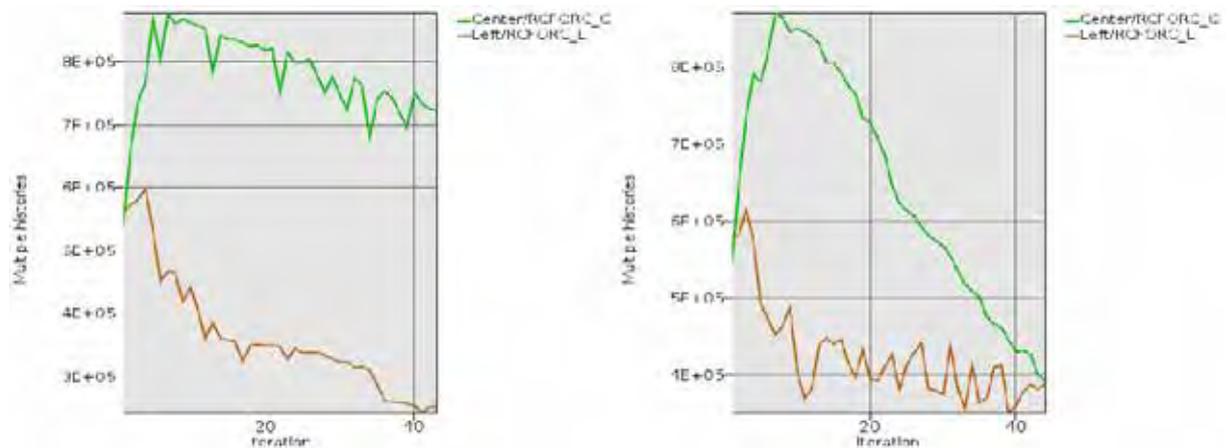

Figure 6 Convergence of the reaction forces. The plots are that of the maximum reaction force for each load case versus the iteration number. The equaly weighted design study is shown to the left, while the results for dynamic weighing is shown to the right.

7   Summary

LS-TaSC computes the shape of a structure with the best use of the material. It has been developed for non-linear structures analyzed in an industrial environment and is accordingly suitable for large linear problems. This tool considers solids and shells, global constraints, multiple parts, and manufacturing constraints. This tool has been extended to load case weighing and forging geometry definitions.

References

[1] Livermore Software Technology Corporation, LS-TaSC™: A Topology and Shape Computations for LS-DYNA, User's Manual, Version 2.1, Livermore Software Technology Corporation, Livermore, CA, 2011.

[2] Hallquist JO. LS-DYNA theoretical manual, Livermore Software Technology Corporation, Livermore, CA, 1998.

[3] NM Patel, *Crashworthiness Design Using Topology Optimization*, PhD thesis, University of Notre Dame, 2004.

[4] RT Haftka, Z Gurdal, MP Kamat, *Elements of Structural Optimization*, Kluwer Academic Publishers, Dordrecht, The Netherlands, 2nd ed., 1990.

[5] SN Patnaik, DA Hopkins, Optimality of Fully-Stressed Design, *Computer Methods in Applied Mechanics and Engineering*, 165, 215-221, 1998.

Total Human Model for Safety - THUMS

About

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.

THUMS is limited to civilian use and may under no circumstances be used in military applications.
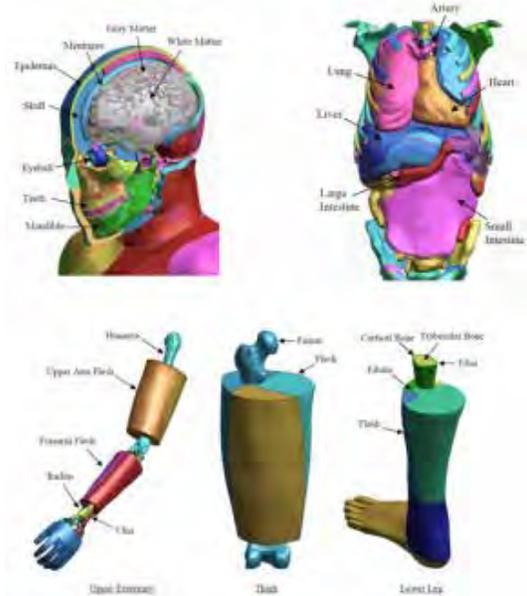
**Model Details: The latest Version of the THUMS model is Version 4.**

Three different sizes of the model are available:
- Adult female 5th percentile
- Adult male 50th percentile
- Adult male 95th percentile

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.

LSTC is the US distributor for THUMS. Commercial and academic licenses are available. For more information please contact us at THUMS@lstc.com .
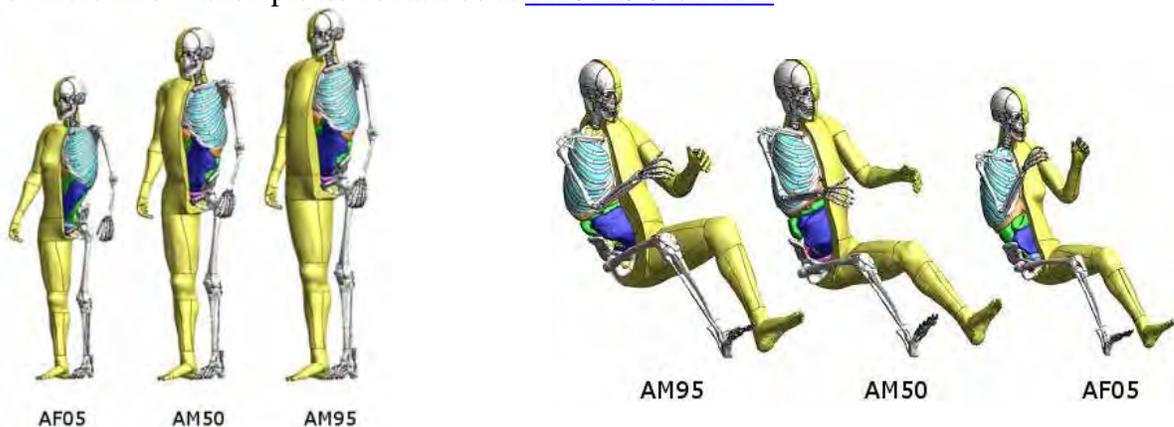
THUMS®, is a registered trademark of Toyota Central R&D Labs.

Incompressible CFD

The incompressible flow solver is based on state of the art Finite Element technology applied to fluid mechanics. It is fully coupled with the solid mechanics solver. This coupling permits robust FSI analysis via either an explicit technique when the FSI is weak, or using an implicit coupling when the FSI coupling is strong. In addition to being able to handle free surface flows, there is also a bi-phasic flow capability that involves modeling using a conservative level-set interface tracking technique. Basic turbulence models are also supported. This solver is the first in LS-DYNA to make use of a new volume mesher that takes nice surface meshes bounding the fluid domain as input. In addition, during the time advancement of the incompressible flow, the solution is adaptively re-meshed as an automatic feature of the solver. Another important feature of the mesher is the ability to create boundary layer meshes. These anisotropic meshes become a crucial part of the model when shear stresses are to be calculated near fluid walls.
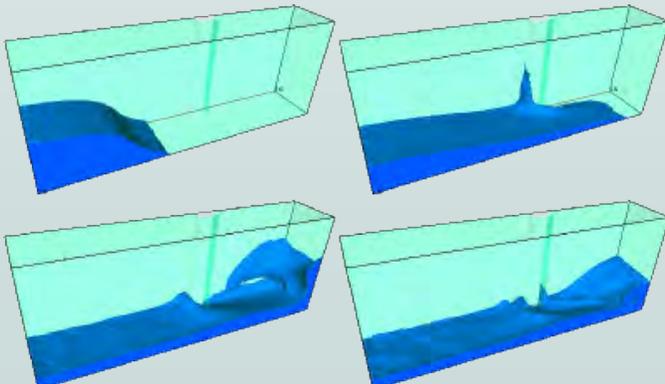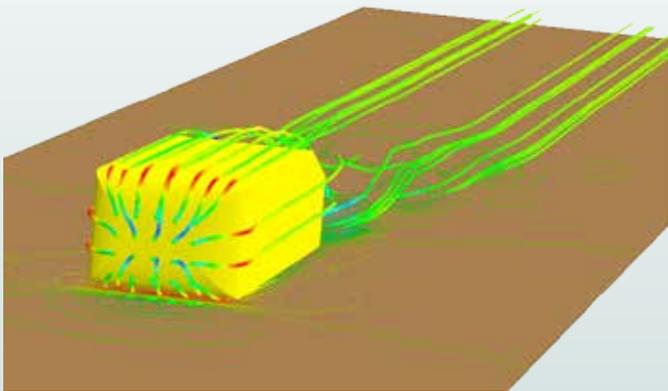
Features

¨ Automatic Volume Meshing

¨ Adaptive Meshing

¨ Fluid Structure Interaction

¨ Free Surface Flow

¨ Bi-Phasic Flow

¨ Turbulence Models

¨ Conjugate Heat Transfer

info@lstc.com

GOMPUTE

A Gridcore Company

GOMPUTE Inc.

Cloud Service FOR
LS-DYNA®

Gompute delivers professional and reliable High Performance Computing on demand for Technical and Scientific users.

Imperial Business Park,

4819 Emperor Boulevard,

Suite 400

Durham, NC. 27703 USA.

Contact:

info@gompute.com

www.gompute.com

Gompute also provides independent software vendors (ISVs) with a faster time to market by reselling their products on-demand. Users of Gompute range from large corporations to one-man consultant companies. Gompute's own technology allows the establishment of Virtual teams who can share resources and results collaboratively.

Gompute provides: CPU hours, storage, system administration and support for applications provided by third party Gompute partners.

Gompute supports: departmental license servers, simulation database repository, common documentation areas, etc.

**Hosting the Grand Reception - 12th Int'l LS-DYNA® Users Conference
June 03-05, 2012, Dearborn, MI
Visit our GOMPUTE Exhibitor Booth**