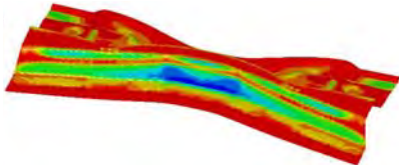# *FEA* Information   http://www.feainformation.com

Engineering Journal and Website Resource

An Enhanced One-Step Metal Forming Solution in LS-DYNA®
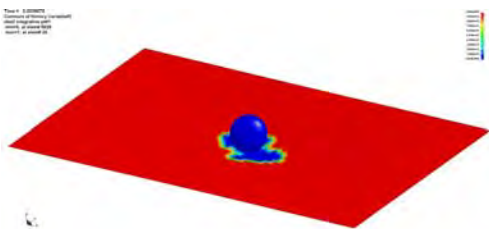
SGI Acquires OPENCFD LTD.

Enterprise Engineering Solutions Joins FEA Information Journal Community
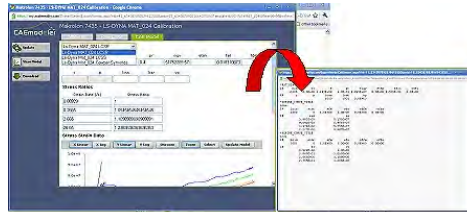
No Fee Info Day

FEA Information Day
DYNAmore Nordic
LS-DYNA & ANSA
No Fee Info Day

Synergy between LS-DYNA and modeFRONTIER

DatapointLabs Matereality's CAE Modeler

Sukhoi Su-30MKI

NATO reporting name: Flanker-H

TABLE OF CONTENTS

Articles

Solutions

EVENTS

Training Courses

35	CADFEM
36	LSTC
37	Dynamore Nordic
38	AS+
39	Shanghai Hengstar
40	ETA

LS-DYNA Users Challenge Your Knowledge

41	Challenge #1 Question & Answer
43	Challenge #2 Question & Answer
45	Challenge #3 Question & Answer

Formula One Student Information

47	Formula One – Universities – Projects – Teams

Directory Listings

48	FEA Consultants - Engineering Services - North America
49	FEA Consultants - Engineering Services - Europe
51	FEA Consultants - Engineering Services - Asia Pacific
51	LS-DYNA Distributors
55	LSTC Model Development Team

Advertisement

56	Shape The Future - LS-TaSC™ Topology Design

Announcements

LSTC:

LSTC is now accepting abstracts and early acceptance is being issued for papers for the 12th International LS-DYNA Users Conference. papers@lstc.com Sponsors and Exhibitors are additionally being given first choice on booths, upon sign up. Cathie@lstc.com

FEA Information Inc Joining our Community:

Mexico, is **e2s** Enterprise Engineering Solutions, bringing LS-DYNA commercial and educational products to Universities and Companies in Mexico.

Brazil we are pleased to let you know that **Universidade Federal De Santa Catarina** - Fórmula UFSC is now listed on the Formula Team page and will publish in depth information in September.

FEA Information Not to Miss:

 TechJobsCafe http://www.techjobscafe.com/ brought to you by MCADCafe

Gompute User Meeting "Leading the HPC Cloud"
28th and 29th of September 2011 - Scandic Crown Hotel –
Polhemsplatsen 3 - 41 111 Gothenburg – Sweden   www.simdi.se

Sincerely,   Marsha J. Victory, President, FEA Information Inc
mv@feainformation.com

From engineering to horses - http://www.livermorehorses.com

Dusty,and Quincy, our24" tall miniature horses, who are not exercising, but staring at the grass to eat

Dusty right after I groomed him, and before he rolled in the dirt

Platinum

OASYS Ltd: http://www.oasys-software.com/dyna/en/	**JSOL Corporation**: http://www.jsol.co.jp/english/cae	**SGI:** http://www.sgi.com
ETA: http://www.eta.com	**DYNAmore GmbH** http://www.dynamore.de	**ESI Group**: http://www.esi-group.com
BETA CAE Systems S.A.: http://www.beta-cae.gr	**LSTC:** http://www.lstc.com	**Dalian Fukun Technology Co. Ltd.:**
MICROSOFT http://www.microsoft.com	**Panasas, Inc.** http://www.panasas.com	**Shanghai Hengstar Technology Co. Ltd** http://www.hengstar.com/
GridCore AB http://gridcore.se		

Bronze Participants

Data Point Labs	APACS	Lancemore Corporation Japan
e2s Enterprise Engineering Solutions		

A Contribution to New ALE 2D Method Validation

http://www.dynalook.com/international-conf-2010/Simulation-2-5.pdf

Nicolas VAN DORSSELAER (LSTC), Vincent LAPOUJADE
Alliance Services Plus,
5 avenue Didier DAURAT, Toulouse, France
n.van-dorsselaer@asplus.fr

Abstract

Since LS-DYNA® v971 r4, a new ALE 2D method is available. Several finite element studies were performed by AS+ to evaluate the precision of this method in pure Multi-Materials Euler studies .

Pure Multi-Materials Euler was tested on Impacts and Explosives studies from Defense and Spatial fields. A High Velocity Impact, a Long Rod Penetration, an Explosively Formed Projectile, a Shaped Charge Jet and an Air Blast were modeled using 2D axisymmetric models. Results were compared to experimental data extracted from reference papers. The very good precision obtained with the 2D ALE method and its ability to represent very dynamic phenomenon will be shown.
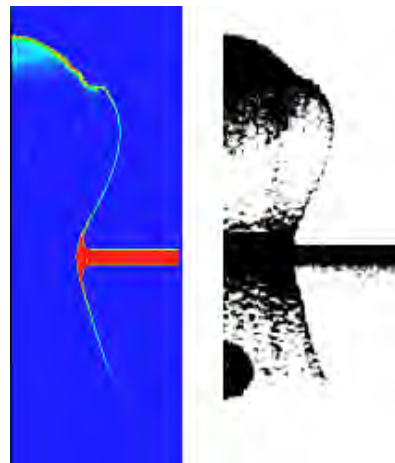
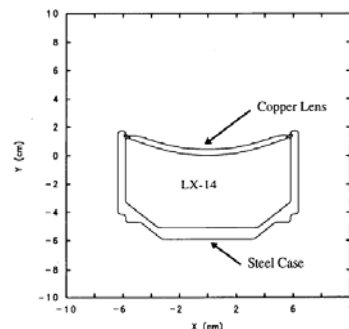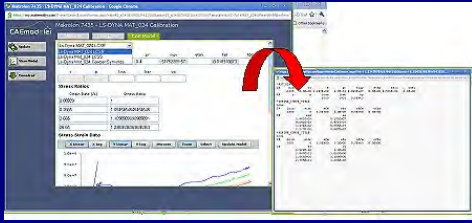Figure 3 : Views of simulation and experiment final nucleus shape

Figure 9 : Initial EFP configuration

Matereality's CAE Modeler Software for Creating LS-DYNA Material Cards from Stress-Strain Curves

Abstract: We present Matereality CAE Modeler software for the conversion of rate dependent stress-strain curves into MAT24 material models commonly used in LS-DYNA today. The interactive software gives the LS-DYNA user a structured way to create material cards along with support of the various rate dependency options including Cowper-Symonds, LCSS and LCSR. Additional features include management of LS-DYNA unit systems, material info, and the writing of outputs directly readable by LS-DYNA. Reverse modeling tools are also available to recreate 'stress-strain' data from rate dependent models material cards. Data loading software allows for the import and storage of rate dependent stress-strain data for later use in material modeling.

Converting Rate Dependent Stress-Strain Curves into MAT24 Material Models

Matereality's CAE Modeler converts raw material data into material models for LS-DYNA. The CAE Modeler comprises a data mining robot to locate and collate the correct data needed for a material model, material parameter conversion, and the writing of files readable by each supported CAE software. The interactive software gives the LS-DYNA user a structured way to inspect the raw data and pick the right features to include in the material card along with support of the various rate dependency options including Cowper-Symonds, LCSS and LCSR. The CAE Modeler creates an auto-fit curve, including rate dependency and initial plasticity; a graphical user interface allows the analyst to edit and tune the material model parameters as needed for higher fidelity. Users can change modulus to stiffen/soften material model, add missing data (such as Poisson's ratio or density), select a LS-DYNA unit system, or tune stress ratios to improve rate dependency fit as needed. For post-yield modeling, users can select points on a quasi-static curve. Applicable files (.dyn) may be downloaded directly to the desktop and are directly readable by LS-DYNA. Show Me How!
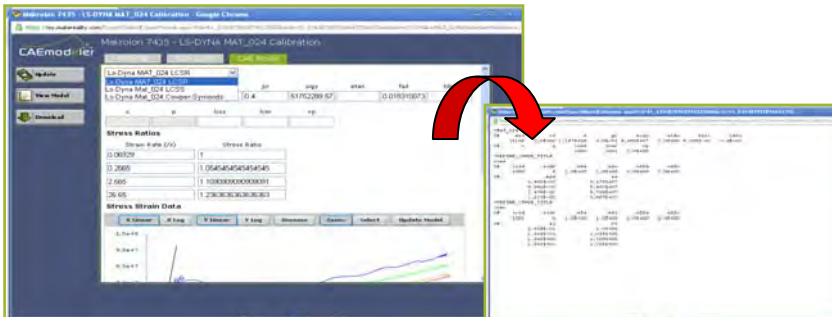
Reverse Material Modeling

Engineers sometimes receive crash material cards for CAE software other than the one they are using. With reverse material modeling, users input a material model from one CAE software (for example, a rate dependent Abaqus .inp material card). This tool recreates stress-strain or other raw data using only the material model. If necessary, this data can be reviewed for accuracy and edited, or additional data can be added at this time. Raw data may be saved to the appropriate Personal or Workgroup Material Database and tagged with material ID and sample details. The recreated raw data can then be converted into material cards (e.g., an LS-DYNA .dyn file) using Matereality's LS-DYNA CAE Modeler software. This advanced technology saves time and eliminates the need to interpret or edit exotic file types. [Show Me How](#)!

Accurate Data Loading Using Database Builder

The Database Builder software allows for the import and storage of rate dependent stress-strain data (or other data) for later use in material modeling. Matereality's Database Builder is a powerful engine that allows any user to populate their own material database. Data can be loaded, copied from other databases and edited, or imported from a variety of file formats including CAE material model files. The optional DataPipes Import feature reads material data from test instruments directly into your database. The Data Loading Wizard permits easy upload of both simple and complex material data. Once data is within the database, Matereality's software can be used to examine and transform it to other uses.
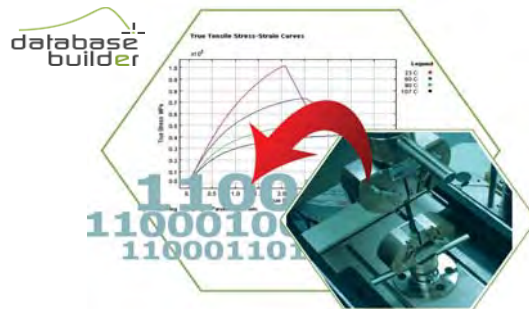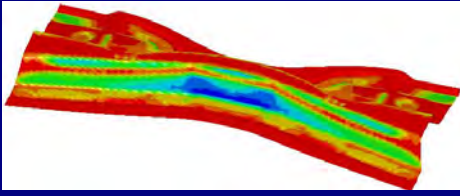
An Enhanced One-Step Metal Forming Solution in LS-DYNA®

Xinhai Zhu, Dilip Bhalsod, Li Zhang
Weiguo Lee, Philip Ho
LSTC

INTRODUCTION

One-step solution, also called "inverse" method, employing the deformation theory as opposed to the incremental theory, has been used in performing sheet metal forming simulation in early stage of a vehicle design, development and manufacturing process. Since the deformation loading is linear, it is strain path-independent and therefore history independent. It is most widely used for initial blank/material estimation, trim-lines unfolding, and initialization of plastic strains and blank thickness in crash simulation. One-step solution is known for its fast computational speeds and simplicity of input requirement. Typically, although it would help, the binder and addendum geometry are not required, saving a lot of efforts in creating these geometries; stamping die processes are not of concern, either; Furthermore, there is no need for contact treatment, since there are no tools and dies involved. Therefore, as far as the geometry goes, the trimmed product is all that is needed to start a one-step solution.

Commercial one-step solution for metal forming has been available for more than two decades. Now, requested by users in metal forming and crash application areas, and through intense and joint development efforts, the previously implemented one-step solution has been enhanced. The new release has included all necessary features required. It is now available in LS-DYNA® starting in Revision 67800 and later of the R6 release.

MAIN FEATURES

The one-step solution is activated with a keyword *CONTROL_FORMING_ONESTEP, with OPTION=7 (Card 1, 1st field). It utilizes some of the existing implicit static solver to iterate and to arrive at a converged solution. Most models need about 4 steps to achieve good results. Some of the details are as follows,

- INPUT – In addition to the usual input for part, material and physical properties, final product mesh in keyword format is needed. Any trimmed-out holes need to be filled. The filling can be done in LS-PrePost v3.2, by selecting *Mesh/EleGen/Shell/Shell by Fill_Holes/Auto Fill*.
- OUTPUT - Results are stored in an ASCII file named 'onestepresult'. It is essentially a dynain file storing final forming thickness and plastic strains, which can be plotted within LS-PrePost. The final estimated blank size (in its initial, flat shape) can be viewed and output through d3plot files, along with intermediate

shapes stored for each implicit step. The first state will be the initial estimated blank from LS-PrePost.

- Draw beads can be applied along the periphery of the final part in two ways. The first so-called 'Auto Beads' applies a user supplied lock percentage for a variable *AUTOLOCK* (Card1, 3rd field in *CONTROL_FORMING_ONESTEP) to all nodes along the part boundary automatically. A 30% lockage is set as default for the variable *AUTOLOCK*. Optionally, a node set can be defined along the part boundary and a lock percentage using the _DRAWBEAD option *CONTROL_FORMING_ONESTEP_DRAWBEAD. The lock percentage is based on a percentage of a fully locked force, determined by tensile strength and sheet thickness, and is automatically calculated within the solver. In most cases, default lockage of 30% from 'Auto Beads' achieves sufficiently good results.

- Optionally, friction can be accounted for, by using a new option *CONTROL_FORMING_ONESTEP_FRICTION. The frictional force is based on a "binder pressure", and is a percentage of the input pressure.

- Nodal restraints used in implicit calculation to prevent rigid body motion are automatically applied using a new keyword *CONTROL_FORMING_ONESTEP_AUTO_CONSTRAINT, with OPTION=1 (Card 1, 1st field).

- All other implicit cards, such as *CONTROL_IMPLICIT_GENERAL, _SOLUTION, _SOLVER, _AUTO, _TERMINATION, etc., are used to set the convergence tolerance, termination criterion, etc. It was determined, that the two important variables controlling the solution convergence, Deltau in *CONTROL_IMPLICIT_TERMINATION, and Dctol in *CONTROL_IMPLICIT_SOULTION, can be set to 0.001 and 0.01, respectively, to obtain the most efficient and best results.

The solver currently supports all element types, with output of results in one integration point in plane and three integration points in the thickness direction. Both material types *MAT_024 (tables supported) and *MAT_037 are enabled with the one-step solver. Multiple part IDs are permitted (for filled holes in different PID). Finally, a double precision solver needs to be used. The following control cards are typically used, noting that the no input for the variable AUTOLOCK means default lockage of 30%.

```
*CONTROL_TERMINATION
$    ENDTIM
        1.0
*CONTROL_IMPLICIT_GENERAL
$    IMFLAG       DT0
        1        0.25
*CONTROL_FORMING_ONESTEP
$    OPTION             AUTOLOCK
        7


*CONTROL_FORMING_ONESTEP_AUTO_CONSTRAINT
$    AUTOSPC
        1
*CONTROL_IMPLICIT_TERMINATION
$    DELTAU
      0.001
*CONTROL_IMPLICIT_SOLUTION
$    NSLOLVR    ILIMIT    MAXREF      DCTOL      ECTOL
        2        11      1200       0.01       1.00
*CONTROL_IMPLICIT_SOLVER
$    LSOLVR
        4
*CONTROL_IMPLICIT_AUTO
$    IAUTO    ITEOPT    ITEWIN     DTMIN     DTMAX
        0         0         0       0.0       0.0
```

EXAMPLES

Two examples are provided below.

1) NUMISHEET 2005 cross member – With average element size of 5 mm uniformly across the blank, the drawn part (with addendum and binder) consists of 25328 elements. A DP600 material properties with *MAT_037 was used. On a 1 CPU Linux machine, it took 3 min. to complete the run, with 4 steps. Thickness, plastic strain, and blank size prediction were reasonable, as shown in *Figures 1~3*.

Contours of shell thickness
min=1.43887, at elem# 19298
max=1.83093, at elem# 9038

1.600e+00
1.568e+00
1.536e+00
1.504e+00
1.472e+00
1.440e+00
1.408e+00
1.376e+00
1.344e+00
1.312e+00
1.280e+00

Figure 1 Sheet Thickness Prediction (t_0=1.6mm)

Contours of plastic strain
max ipt. value
min=0, at elem# 7
max=0.15, at elem# 20257

1.500e-01	
1.350e-01	
1.200e-01	
1.050e-01	
9.000e-02	
7.500e-02	
6.000e-02	
4.500e-02	
3.000e-02	
1.500e-02	
0.000e+00	

Figure 2 Plastic Strain Prediction

Figure 3 Initial Blank Size Prediction (not to be scaled)

2) NCAC Taurus model firewall - With average element size of 8 mm across the blank, the trimmed part (with holes filled) consists of 15490 elements, as shown in *Figure 4*. A BH210 material properties with *MAT_024 was used. On a 1 CPU Linux machine, it took 4 min. to complete the run with 4 steps total. Thickness, plastic strain, and blank size prediction were reasonable, as shown in *Figures 5~7*.
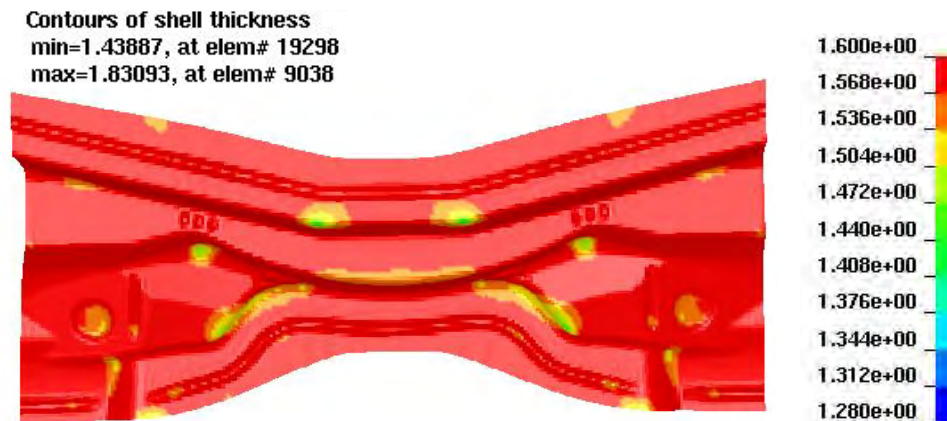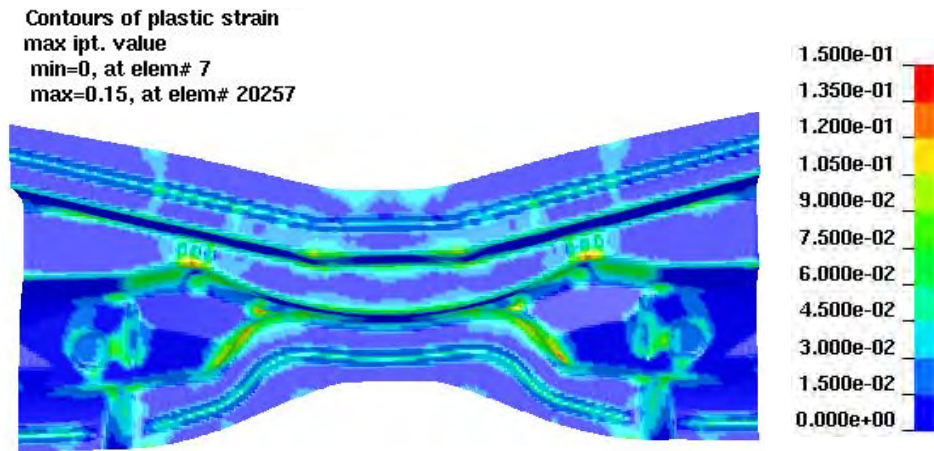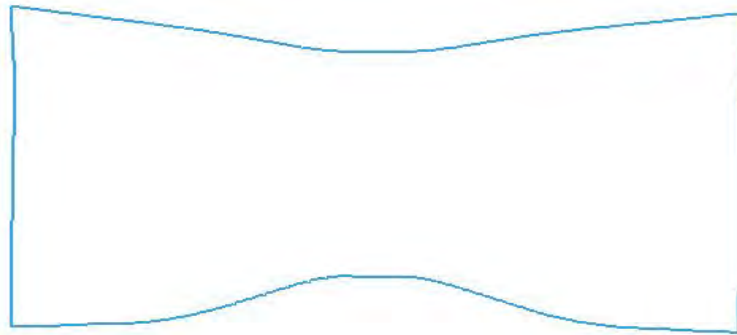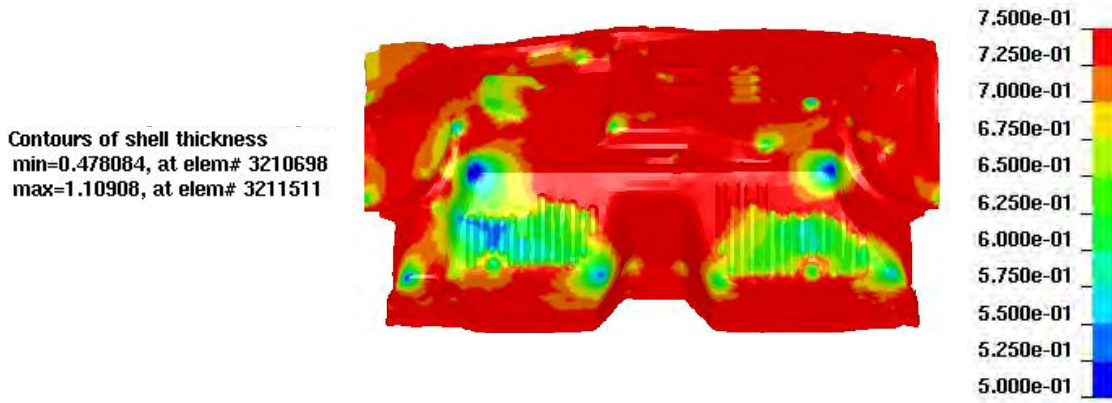

Figure 4 Sheet Thickness Prediction (t_0=0.75mm)

12

Figure 5 Sheet Thickness Prediction (t_0=0.75mm)
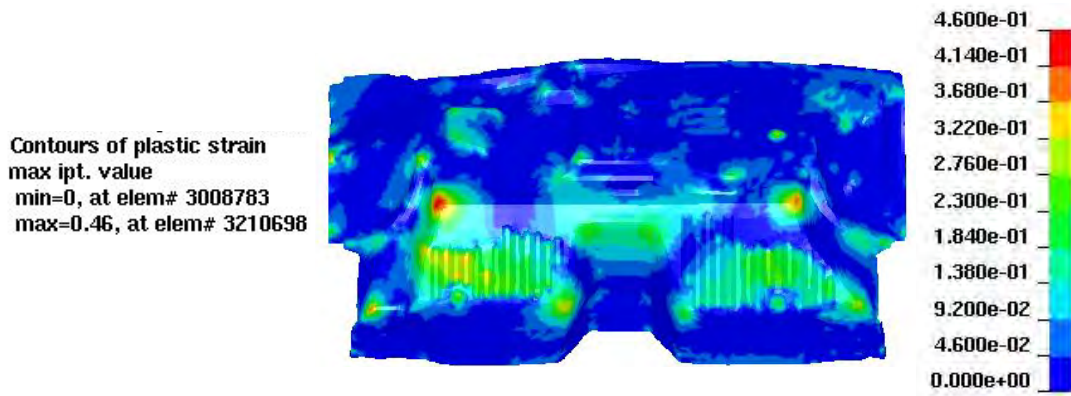
Figure 6 Plastic Strain Prediction
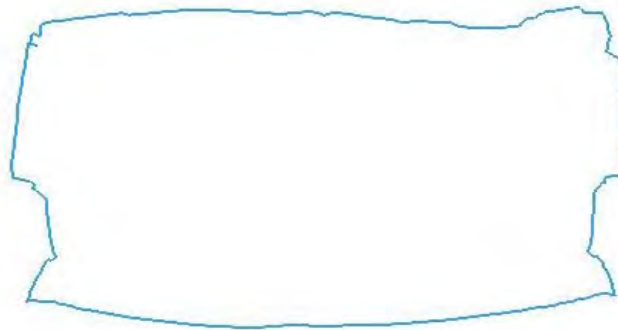
Figure 7 Initial Blank Size Prediction (not to be scaled)

DISCUSSION/CONCLUSION

The new one-step solver from LS-DYNA provides excellent results for a one-step solution. It is fast, robust, easy to use and can be applied in blank size prediction, material cost quotation in a manufacturing engineering environment, and in initializing forming effects for crash simulation. It is loaded with features that cover all of the needs for initializing forming for crash, such as, *MAT_024, multiple parts and tables for

13

material curves, etc. Since most users use LS-DYNA for crash simulation, this new feature provides a few added advantages and benefits – it is entirely compatible in forming/crash mapping; users can perform all forming and crash simulation in one software; it provides an opportunity for the management to consolidate and realize cost savings.

It is important to note that although one-step results provide a fast, initial estimate, and are good enough in some applications, its results are not as accurate as those provided by incremental solutions, which are close to reality. The following *(Figure 8)* illustrates the incremental results for the same cross member in two different views. It is noted that the results are much more realistic, when compared with experimental data provided by the benchmark.
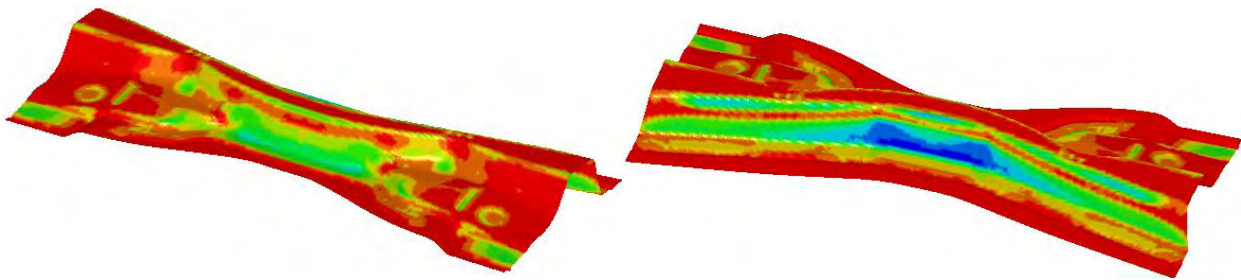
Figure 8 Thickness Prediction from LS-DYNA Incremental Results

Leader in Technical Computing Set to Reshape Product Design by Offering Open and Complete CFD Solution

SGI announced on August 15, 2011 that it had completed the acquisition of OpenCFD Ltd., the leader in open source computational fluid dynamics.

"Computational fluid dynamics (CFD) is one of the most important application areas for technical computing," said SGI CEO Mark J. Barrenechea. "With the acquisition of OpenCFD Ltd., SGI will be able to provide our customers the market's first fully integrated CFD solution, where all the hardware and software work together."

OpenCFD is the leader in open source CFD software and has a large, active user base across commercial engineering, science and the public sector. OpenCFD is the developer of the OpenFOAM® software, consisting of over 80 solver and 170 utility applications – solving anything from complex fluid flows involving chemical reactions, turbulence and heat transfer to solid dynamics and electromagnetics. Customers benefit from the fact that OpenFOAM is an open platform and offers them immediate access and freedom to customize and extend its existing capabilities.

"We are excited to be part of SGI and now be able to take OpenFOAM to the next level," said Henry Weller, founder of OpenCFD. "OpenFOAM is the highest

quality code available for CFD. The open source model provides customers with the code transparency and the full view into the inner workings of the software they need to have confidence in the results, particularly when they perform safety calculations for projects such as nuclear reactors or aircraft design."

CFD is accelerating product design and development across multiple industries by eliminating the need to build expensive physical prototypes. Using CFD, companies can model and simulate the interaction of liquids and gases with solid surfaces, addressing one of the key challenges in creating almost any product, from bikes to airplanes, propellers to turbines, or from potato chips to silicon chips. Companies worldwide are creating the future on SGI systems, and by providing an open platform for CFD, SGI is making CFD available to all product engineers around the world.

"In terms of cost, OpenFOAM dramatically lowers the barrier of entry to world class CFD software," said John O. Hallquist, president of LSTC. "By their investment in OpenFOAM, SGI now has the potential to significantly expand the CFD marketplace, not only to existing

users, but to new users. This is truly an exciting development."

"TotalSim believes that the acquisition of OpenCFD by SGI and the formation of the OpenFOAM Foundation will provide the long-term stability, resources and structure for the continued success of OpenFOAM," said Rob Lewis, managing director at OpenCFD customer TotalSim. "TotalSim is looking forward to continuing the relationship through SGI and working together to further expand the use and benefit of OpenFOAM to the engineering community."

The entire OpenCFD team, led by Henry Weller, has joined SGI as full-time employees and will be based out of SGI's EMEA Headquarters located in the UK. In addition, SGI has formed a non-profit foundation to make OpenFOAM accessible to everyone and allow for community contributions. SGI is fully committed to the continued development of OpenFOAM, which will continue to be open under the GNU Public License (GPL). The company invites every scientist, developer, engineer and student working in the field of CFD to join the OpenFoam community, download the software for a test drive and to make their own contributions.

In addition to providing free downloads of OpenFOAM source code from the SGI-sponsored OpenFOAM foundation's openfoam.org web site, SGI will offer:

- A fee-based OpenFOAM support subscription to help customers implement and run OpenFoam in their environments.
- A fee-based, fully supported SGI OpenFOAM distribution, including SGI message passing interface (MPI). SGI® OpenFOAM® and SGI® MPI work together to provide the best performance possible.
- Training and professional services for complex CFD deployments on SGI and in other environments.

For companies that would like to start using SGI OpenFOAM today, the full capability is available via the SGI® Cyclone cloud offering.

About SGI
SGI, a trusted leader in technical computing is focused on helping customers solve their most demanding business and technology challenges. Visit sgi.com for more information.

LS-DYNA
Solutions For Mexico
e2s Enterprise Engineering Solutions

FEA Information Inc. would like to welcome Tomas Vargas and e2s Enterprise Engineering Solutions to our Engineering Journal Community.

With the recent interest and growth of LS-DYNA in Mexico, LSTC has established e2s as a reseller, of their LSTC product line: LS-DYNA, LS-OPT, LS-PrePost, LSTC's ATD and Barrier Models.

30-day demonstration licenses are now available through e2s Enterprise Engineering Solutions - with local support from Tomas Vargas, assisted by LSTC support, as this new sales channel builds on a solid foundation of educational and commercial sales.

e2s is an engineering services company specializes in 3D simulation models. Founded in 1994 e2s has 12 years experience in the market providing service to diverse industry segments such as: Automotive, Aerospace, Electric & Electronics, Home Appliances, Food & Drinks, Steel, Glass, Pharmaceutical, among others.

For your LS-DYNA 30-day demo license, educational pricing, and commercial pricing, in Mexico for LS-DYNA or there other software products please contact:
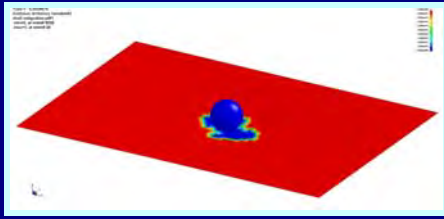
e2s Monterrey Mexico

Tomas Vargas
tomas.vargas@e2s.com.mx
Humberto Lobo 520 H-1
Plaza San Pedro
Colonia del Valle
Garza García, N. L., México 66220

Synergy between LS-DYNA® and modeFRONTIER to predict low velocity impact damage on CFRP plates.

**R. Borrelli[1], V. Primavera[2], M. Perillo[2]**
[1] CIRA scpa, Via Maiorise snc, 81041 Capua, Caserta, Italy
[2] EnginSoft SpA, Via A. Murri, 2 – Z.I., 72023 Mesagne, Brindisi, Italy

Engineering structural components made of composite materials is becoming increasingly important for use in various engineering applications. Composite materials are preferred with respect to the competing traditional metals due to a range of potential advantages:

- high strength-to-weight ratio
- high stiffness-to-weight ratio
- resistance to corrosion
- low coefficient of thermal expansion, etc.

Nevertheless, composite materials do suffer from poor resistance to impact loading. This results in damage that can cause severe structural degradation, such as reduction in compressive strength. The situation is more critical for low-velocity impacts which induce significant internal damage undetectable by visual inspection.
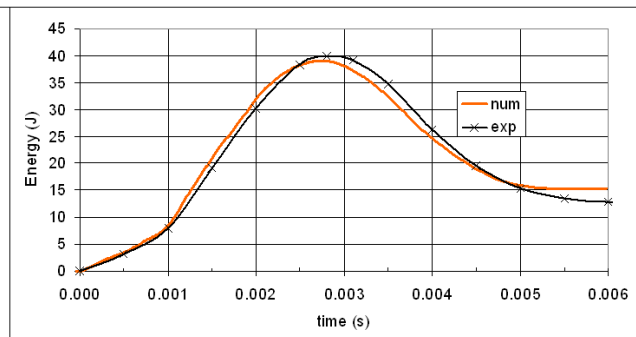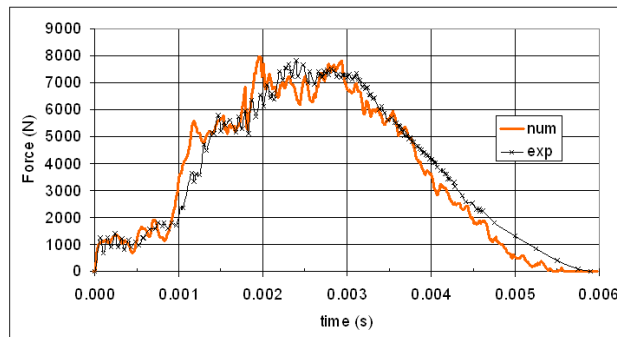
Comparison between numerical-experimental force and energy

Thus, numerical models able to simulate impact events (which can occur during maintenance operations), are needed in order to assess the residual structural capacity of a composite structure and, at same time, to reduce experimental tests which are quite expensive.

The full paper of this abstract will be presented at the EnginSoft International Conference 2011. The presenters from CIRA scpa Italy will demonstrate how a LS-DYNA numerical model was developed and coupled with

modeFRONTIER, a process integration and design optimization tool. modeFRONTIER is capable of exploring the design space (i.e. the free parameters dominions) and finding the configurations which satisfy several objective functions.

The definition of some numerical parameters of the LS-DYNA model (time step, damping coefficient, contact stiffness, etc.) is a considerable challenge because no reliable data is available in literature, the data is chosen on the basis of the analyst's experience. The LS-DYNA – modeFRONTIER integrated procedure developed in this work, allowed to obtain both a better understanding of the influence of such parameters on the simulation results (sensitivity analysis) and the configuration which provided the best agreement with the experimental data (optimization).
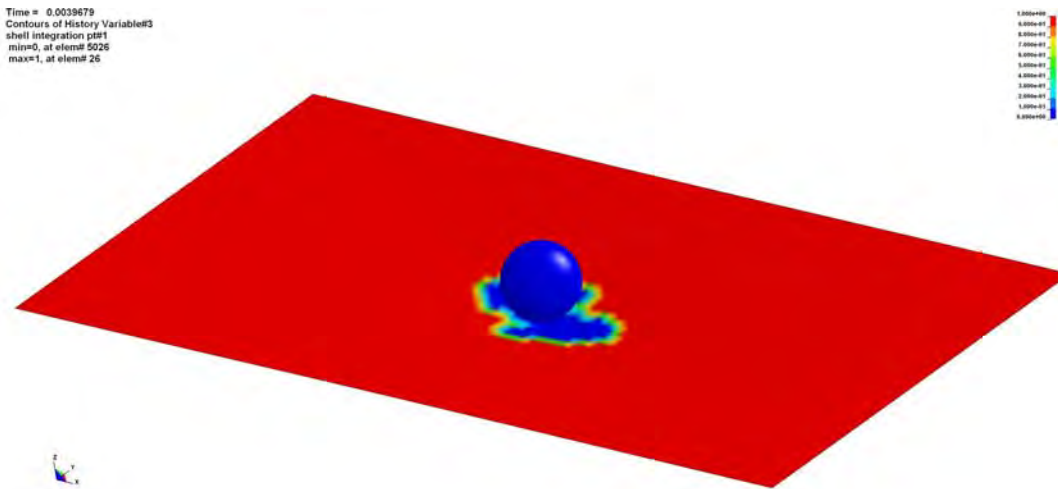
Matrix damage due to tensile load (HV3)

The procedure was validated against a test case extracted from literature which consists of a rectangular composite plate impacted by an hemispherical steel impactor with an energy impact of 40 Joule. The experimental data used for the validation activity were:
- Contact force vs. time curve
- Deflection vs. time curve
- Absorbed Energy

For each of these quantities, an objective function to be minimized (3 relative errors) was taken into account. The integrated procedure allowed to identify the configurations which are good compromises with respect to the above mentioned objective functions.

To hear more about this application, please visit the EnginSoft International Conference 2011, 20-21 October in Verona, Italy. Alternatively, please ask for the Conference Proceedings.
Please see www.CAEconference.com for further details.

Author contact: Rosario Borrelli, r.borrelli@cira.it

It is our pleasure to invite you to DYNAmore Nordic first LS-DYNA and ANSA Info day, October 11, in Göteborg, Sweden. At the Info day we will present news in LS-DYNA and ANSA and you will also get the opportunity to meet the personnel of the DYNAmore Group.

The Info day should not be mixed up with the Users' Conference that will be arranged in 2012, instead the Info day will focus on new applications in LS-DYNA and ANSA presented by LSTC, BETA and DYNAmore personnel.

Date: October 11, 2011
Time: 09.00 – 17.10
Place: Hotel 11, Göteborg Sweden
Registration:
http://www.dynamore.se/reg/

Attendance Fee: FREE OF CHARGE

Preliminary schedule

09.00	09.30	Coffee and registration
09.30	09.40	Welcome
09.40	10.00	Presentation of DYNAmore GmbH
10.00	10.30	LS-DYNA Selective mass scaling
10.30	11.00	LS-DYNA Failure models
11.00	11.30	Sponsor presentation
11.30	12.00	ANSA – LS-DYNA
12.00	13.00	Lunch
13.00	13.30	ANSA and mETA News
13.30	13.50	DYNAmore Nordic AB
13.50	14.40	News in LS-DYNA
14.40	15.30	Coffee
15.30	16.00	News in LS-OPT
16.00	16.30	LS-DYNA Process simulations
16.30	17.00	LS-DYNA Implicit
17.00	17.10	Closing

The following does not represent any software. It is solely based on our Staff's choice of a Jet Fighter deserving a showcase.

Sukhoi Su-30MKI

The Sukhoi Su-30MKI - (NATO reporting name: Flanker-H) is a multirole aircraft jointly developed by Russia's Sukhoi and India's Hindustan Aeronautics Limited (HAL) for the Indian Air Force (IAF). A variant of the Sukhoi Su-30, it is a heavy all-weather, long-range air superiority fighter which can also act as a strike fighter aircraft.

Development of the variant started after India signed a deal with Russia in 2000 to manufacture 140 Su-30 fighter jets.[4] The first Russian-made Su-30MKI variant was accepted into the Indian Air Force in 2002, while the first indigenously assembled Su-30MKI entered service with the IAF in 2004.[6]

In 2007, the IAF ordered 40 additional MKIs.

As of January 2011, the IAF has 142 MKIs under active service with plans to have an operational fleet of 280 MKIs by 2015.[1][8] The Su-30MKI is expected to form the backbone of the Indian Air Force's fighter fleet to 2020 and beyond.

The aircraft is tailor-made for Indian specifications and integrates Indian systems and avionics as well as[10] French and Israeli subsystems. It has abilities similar to the Sukhoi Su-35 with which it shares many features and components.

The time step of a finite element simulation using an explicit time integration should not exceed the smallest critical time step of the whole model for stability reasons.

The relation for the computation of the critical time step in LS-DYNA differs depending on the element type as well as on the material laws and parameters used.

Please download the file solid_shell.k
(http://www.dynasupport.com/links/fea-information-examples/solid_shell.k).

The critical time step of a 1-d beam element is given via dt=l*sqrt(rho/E).

In the above mentioned example, this relation yields a value of dt=10.0*sqrt(1.0E-08/10000.0)=1.0E-05, which equals the values of the smallest time steps given in the d3hsp file for the solid and the shell elements.

Thus, for the given material parameters, the computed critical time step of a beam, a solid as well as a shell element is the same.

1.) Set the Poisson's ratio to 0.3. Why do the critical time steps of the solid as well as of the shell elements change?

2.) Why differs the critical time step between the solid and the shell elements for Poisson's ratios unequal zero?

3.) Compute the critical time steps of the solid as well as of the shell elements by hand and compare the solution with the values given in the d3hsp file.

4.) Delete the keyword *CONTROL_BULK_VISCOSITY. Why is there an influence on the critical time step of the solid element?

5.) Increase the Poisson's ratio to the limit of 0.5. What can be observed regarding the critical time steps?

ANSWER ON NEXT PAGE

Answers:

1.) The critical time step depends on the speed of sound of the elements. For an elastic material, the speed of sound depends on both elastic material parameters, in this example the Young's modulus and the Poisson's ratio.

2.) The speed of sound is computed in a different way for solid and shell elements.

3.) Solid: $dt=l/c$ and $c=sqrt\{E(1-nu)/[rho(1+nu)(1-2nu)]\}$ --> $dt=8.618E-06$
   Shell: $dt=l/c$ and $c=sqrt\{E/[rho(1-nu^2)]\}$ --> $dt=9.539E-06$

4.) For solid elements, the critical time step depends on the bulk viscosity coefficients as well,
   cf. LS-DYNA Theory Manual, section 22, TIME STEP CONTROL.

5.) ERROR TERMINATION, because the calculated critical time step for the solid element is zero.
   For shell elements, the critical time step is reduced by a factor of 0.886 in comparison to a simulation with a Poisson's ratio of zero.

	[Finite Element Analysis Theory and Application with ANSYS (3rd Edition)](#)		[Arbitrary Langrangian-Eulerian and Fluid Structure Interaction](#).
	[Isogeometric Analysis: Toward Integration of CAD and FEA](#)		[NURBS for Curve & Surface Design: From Projective Geometry to Practical Use](#)
	[A First Course in Finite Elements](#)		[Engineering Numerical Analysis](#)
	[Go To Book at Amazon](#)		

Meshless Methods in Solid Mechanics

Principles of Geotechnical Engineering

Geotechnical Earthquake Engineering
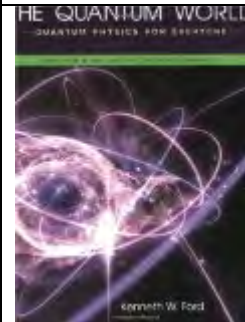
Microflows and Nanoflows
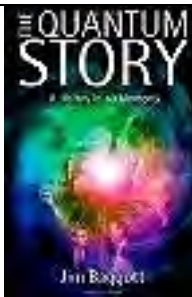
The Quantum Story: A History in 40 Moments

The Quantum World: Quantum Physics for Everyone
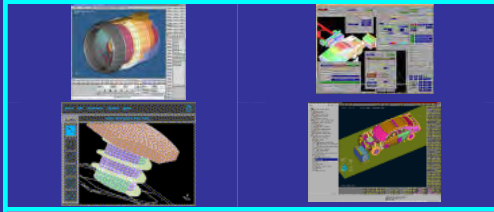
Introduction to Quantum Mechanics (2nd Edition

Practical Stress Analysis With Finite Elements

A preprocessor is a program that processes its input data to produce output. This data is then used as input to another program.

BETA CAE Systems S.A.

http://www.beta-cae.gr/

Provides complete CAE pre- and post-processing solutions. ANSA, the world wide standard pre-processor and full product modeler for LS-DYNA, with integrated Data Management and Task Automation. µETA, with special features for the high performance an effortless 3D & 2D post-processing of LS-DYNA results.

Engineering Technology Associates, Inc.

http://www.inventiumsuite.com

PreSys is an advanced Pre/Post Processor. PreSys is a full-featured, core solution that can be used on its own or with a variety of available add-on applications. The system offers advanced automeshing tools to provide the highest quality mesh with little CAD data preparation. It also features a scripting interface and model explorer feature for in-depth data navigation.

Oasys, Ltd

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

Oasys Primer is a model editor for preparation of LS-DYNA input decks. - Oasys D3Plot is a 3D visualization package for post-processing LS-DYNA analyses using OpenGL® (SGI) graphics.

JSOL Corporation

http://www.jsol.co.jp/english/cae/

JVISION is a general purpose pre-post processor for FEM software. Designed to prepare data for, as well as support, various types of analyses, and to facilitate the display of the subsequent results.

Livermore Software Technology Corporation

http://www.lstc.com

LS-PrePost is an advanced interactive program for preparing input data for LS-DYNA and processing the results from LS-DYNA analyses.

ETA – DYNAFORM & VPG

http://www.eta.com

Includes a complete CAD interface capable of importing, modeling and analyzing, any die design. Available for PC, LINUX and UNIX, DYNAFORM couples affordable software with today's high-end, low-cost hardware for a complete and affordable metal forming solution.

ETA – VPG

http://www.eta.com

Streamlined CAE software package provides an event-based simulation solution of nonlinear, dynamic problems. eta/VPG's single software package overcomes the limitations of existing CAE analysis methods. It is designed to analyze the behavior of mechanical and structural systems as simple as linkages, and as complex as full vehicles.

OASYS   software for LS-DYNA

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

Oasys software is custom-written for 100% compatibility with LS-DYNA. Oasys PRIMER offers model creation, editing and error removal, together with many specialist functions for rapid generation of error-free models. Oasys also offers post-processing software for in-depth analysis of results and automatic report generation.

ESI Group Visual-CRASH For DYNA

http://www.esi-group.com

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.

BETA CAE Systems S.A.– ANSA

http://www.beta-cae.gr

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

http://www.beta-cae.gr

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

Complete Information can be found on the SGI Website including:
- Cyclone™ and LS-DYNA® Success Story
- IDC White Paper   -   Cyclone Supported Applications
- Cyclone Usage Diagram
  http://www.sgi.com/products/hpc_cloud/cyclone/index.html

Cyclone™ is the world's first large scale on-demand cloud computing service specifically dedicated to technical applications. Cyclone capitalizes on over twenty years of SGI HPC expertise to address the growing science and engineering technical markets that rely on extremely high-end computational hardware, software and networking equipment to achieve rapid results. Cyclone supports a number of leading applications partners and five technical domains, including computational fluid dynamics, finite element analysis, computational chemistry and materials, computational biology and ontologies.

Two Service Models: Cyclone is available in two service models: Software as a Service (SaaS) and Infrastructure as a Service (IaaS). With SaaS, Cyclone customers can significantly reduce time to results by accessing leading-edge open source applications and best-of-breed commercial software platforms from top Independent Software Vendors (ISVs). The IaaS model enables customers to install and run their own applications.

LS-DYNA® Implicit Hybrid Technology on Advanced SGI® Architectures*
  White Paper pdf format is at URL: http://www.sgi.com/pdfs/4231.pdf
        Olivier Schreiber, Scott Shaw, Brian Thatch - SGI Application Engineering
        Bill Tang, - SGI System Engineering

Gompute on demand®

A Cloud HPC service oriented to Technical and Scientific users.

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) a 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.

Website: www.gompute.com
Website: www.gridcore.se

LS-DYNA On Demand Additional Core (cloud) Services

US	http://www.sgi.com/products/hpc_cloud/cyclone/index.html
Sweden Germany	http://www.gompute.com

LS-DYNA Information Sites

LS-DYNA Application/Capability	http://www.ls-dyna.com/
LS-DYNA LS-DYNA Benchmarks	http://www.topcrunch.org/
LS-DYNA Publications	http://www.dynalook.com/
LS-DYNA Consulting Companies	http://www.ls-dynaconsulting.com/
LS-DYNA Examples	http://www.dynaexamples.com/
LS-OPT Support	http://www.lsoptsupport.com
LS-OPT User Group	http://groups.google.com/group/lsopt_user_group
LS-PrePost Support	www.lstc.com/lspp
LS-DYNA Support	http://www.dynasupport.com/

ATD – Pedestrian – Barrier Models

ATD LSTC Models:	http:www.lstc.com/models
ATD LSTC Mailing List	atds@lstc.com
ATD Models - DYNAmore	http://dummymodels.com
Pedestrian Impact Model - ARUP	http://oasys-software.com/en/fe-models/pedestrian.shtml
Cellbond Barrier Models - ARUP	http://oasys-software.com/en/fe-models/barrier.shtml
RCAR Barrier Model - ARUP	http://oasys-software.com/en/fe-models/rcar.shtml

Additional LS-DYNA Information Sites

High Strain Rate Testing of Advanced High Strength Steels	http://thyme.ornl.gov/ASP_Main/crashtests/crashtests_main.cgi
High Strain Rate Characterization of Mg Alloys	http://thyme.ornl.gov/Mg_new
(FEM) models of semitrailer trucks for simulation of crash events	http://thyme.ornl.gov/FHWA/TractorTrailer
Single Unit Truck crash model documentation	http://thyme.ornl.gov/FHWA/F800WebPage

Call for Papers/Announcement
June 3-5, 2012
Dearborn, Michigan, USA

12th International LS-DYNA® Users Conference

Livermore Software Technology Corp.

http://www.ls-dynaconferences.com

The 12th International LS-DYNA® Users conference will be held in Dearborn, Michigan, USA, June 3-5, 2012.

The conference will provide an ideal forum for LS-DYNA worldwide users to share technology. Providing a venue to learn new features, and applications of LS-DYNA, LS-OPT, LS-PrePost, LS-TaSC, LSTC Dummy and Barrier Models, as well as the many software and hardware alliances of LSTC's products.

All of our users from industry, research and academia are invited to present a paper.

The exhibition area hosts the latest software and hardware developments providing you the opportunity to speak directly with the company about their products.

Call For Papers

Application areas being accepted for submission:
- Aerospace
- Automotive Crashworthiness
- Ballistic/Penetraion
- Bimechanics
- Civil Engineering
- Compressible Fluid Dynamics
- Electro Magnetics
- Heat Transfer
- Impact/Drop Testing
- Manufacturing Processes
- Metal Forming
- Modeling Techniques
- Nuclear Applications
- Occupant Safety
- Seismic Engineering
- Ship Building
- Transportation
- Virtual Proving Ground

Paper Submission*
For submission details contact papers@lstc.com

Sponsorship
For sponsor and/or Exhibit booth information contact conference@lstc.com

Registration Fees*
Conference Only $450
Training Only $450
Conference/Training $900

Hotel Accomodations
Conference attendees will be able to reserve a discounted room rate. Contact: conference@lstc.com

*The presenter of each accepted paper will receive free admission to the conference, provided that the presenter register for a room, at the Hyatt Regency, Dearborn under LSTC Conference registration

Conference Venue

Hyatt Regency Dearborn
600 Town Center Drive
Dearborn, MI
48126-2793, USA
www.dearborn.hyatt.com

IMPORTANT DATES

Abstract Deadline
     Nov. 11, 2011
Acceptance Notification
     January 5, 2012
Final Paper Deadline
     March 1, 2012

AGENDA

Sunday, June 3
 Registration
 Pre-Conference Seminars
 Reception/Entertainment

Monday, June 4
 Registration
 Plenary Presentations
 Keynote Presentations
 Technical Sessions
 Exhibit Area
 Banquet/Entertainment

Tuesday, June 5
 Registration
 Exhibition Area
 Presentations:
- John O. Hallquist
- Sponsors

POST CONFERENCES COURSES
     Wed/Thurs June 6-7

Suggested Post-Conferences Courses: Impact/Dummies & Barriers - Heat Transfer & hot stamping – ALE – EFG – SPH - Polymeric Material with LS-DYNA – Ballistics/Penetration – contact conference@lstc.com with questions/course requests.

Contact: conference@lstc.com
e-mail: papers@lstc.com   **tel**: 925-449-2500   **Fax**: 925-961-0806
Livermore Software Technology Corp., 7374, Las Positas Rd, Livermore, CA 94551 USA

EnginSoft INTERNATIONAL CONFERENCE 2011
CAE Technologies for Industry
   and ANSYS Italian Conference

Fiera Verona – Verona, Italy
20–21 October 2011

2nd Announcement and Invitation

Mark your diary for one of the most important events in the global CAE Calendar ! The 2 parallel Conferences present a wide range of Virtual Prototyping applications, with a large presence of both technical experts and business decision makers.

The provisional List of Papers already includes contributions from key companies: Ansaldo Breda - Franco Tosi Meccanica - Magneti Marelli – Piaggio - Nuovo Pignone - Tetra Pak – Iveco – Continental and many more to come.

Deadline for early bird registrations:
5th September 2011

Exhibition

The accompanying exhibition will see the world's leading solution providers showcasing products and services covering all aspects of CAE technologies and their successful implementation.
Delegates and exhibitors use the exhibition as an international networking forum. It is here where the experts meet and discuss:

How Virtual Prototyping can speed up design and product development delivering ROI in many forms - in just days!

Do not miss this opportunity to present your products and services to a highly selective audience!

Visit www.caeconference.com to plan your attendance, stay tuned to the Conference Program and for more information on the exhibition..

The Complete Training Courses Offered Can Be Found At: http://www.cadfem.de
Please check the site for accuracy and changes.
Among the many course offering are the following:

Explicit structural mechanics with ANSYS Workbench and LS-DYNA

Beside the trainings on all aspects of short time dynamics we offer also various seminars on new methods available in LS-DYNA.

- Seminar: Introduction to explicit structural mechanics with ANSYS LS-DYNA and LSTC LS-DYNA
- Seminar: Material modeling with LS-DYNA
- Seminar: Simulation of composites with ANSYS Composites PrepPost and LS-DYNA
- Online-Seminar: Contact modeling with LS-DYNA
- Online-Seminar: Modeling joints with LS-DYNA
- Seminar: Crash simulation with LS-DYNA

optiSLang

Parametric simulation and optimization with optiSLang optiSLang is one of the most popular solver for optimization and robust design analyses

Online-Seminar: Advanced parametric simulation with ANSYS Workbench and optiSLang

AnyBody

With AnyBody it is possible to simulate the kinematics of a human body like computing muscle forces for example.

- Seminar: Introduction to simulation of joint- and muscle- forces with AnyBody
- Seminar: Efficient coupling of AnyBody with ANSYS Workbench
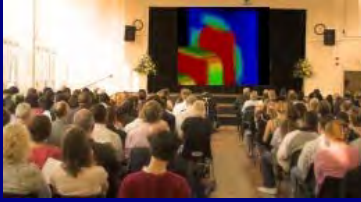
The Complete Training Courses Offered Can Be Found At http://www.lstc.com
Training Coordinator: Cathie Walton Cathie@lstc.com

Start Date	Location	Course
09/06/11	CA	Implicit
09/08/11	CA	Advanced Options in LS-DYNA
09/12/11	MI	LS-PrePost (no charge with Intro to LS-DYNA)
09/13/11	MI	Intro to LS-DYNA
09/19/11	CA	Contact in LS-DYNA
09/20/11	MI	LS-OPT

Courtesy Posting of Classes Offered by
Paul Du Bois and Len Schwer:

Concrete & Geomaterials
     10/4/2011 10/5/2011 Tue-Wed

Modeling & Simulation
     10/6/2011 10/7/2011 Thurs-Fri

Blast Modeling
     10/11/2011 10/12/2011 Tue-Wed

Penetration Modeling
     10/13/2011   10/14/2011   Thurs-Fri

LS-PrePost 3, introduction
  September 12, 2011

LS-DYNA, introductory
  September 13, 2011

LS-DYNA, Adv. training in impact
analysis
  September 20, 2011

LS-OPT, Optimization and
  robust design
  October 4, 2011

LS-DYNA, implicit analysis
  October 11, 2011

ANSA & Metapost, Introductory
  October 25, 2011

LS-DYNA, Simulation of sheet metal
forming processes October 31., 2011

LS-OPT, Optimization and robust design
  November 14, 2011

The complete Training Courses offered can be found at
http://www.asplus.fr/ls-dyna

Please check the site for accuracy and changes.

Among the many course offerings are the following:

Other regular courses (in Paris) ...

LS-DYNA Unified Introduction Implcit & Explicit Solver
   November 21-24

LS-OPT & LS-TaSC Introduction
   October 19-20

Switch to LS-DYNA
   October 5-6

Switch from LS-PrePost 2.X to 3.X
   September 28
   December 14

LS-DYNA Advanced Implicit Solver
   September 27

LS-DYNA ALE / FSI
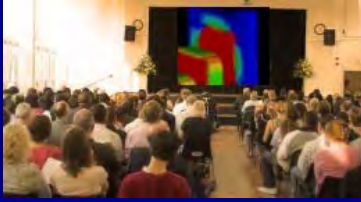   October 17-18

LS-DYNA SPH
   November 8-9

LS-PrePost 3.0 – Advanced meshing capabilities
   September 29
   December 15

LS-DYNA User Options
   to be announced

LS-DYNA – Plasticity, Damage & Failure – By Paul DU BOIS
   October 3-4

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

Email: info@hengstar.com
Phone: +86-021-61630122

2011	5	6	7	8	9	10	11	12
An Introduction to LS-DYNA(High Level)								
Crashworthiness Simulation with LS-DYNA								
Passive Safety and Restraint Systems Design								■
LS-Prepost, LS-DYNA MPP, Airbag Simulation with LS-DYNA	■							
Pedestrian Safety and Passive Safety Simulation with LS-DYNA				■				
Crashworthiness Theory and Technology, Introduction of LS-OPT which is based on LS-DYNA			■					
Concrete & Geomaterial Modeling, Blast Modeling with LS-DYNA								
Frontal Restraint Systems according to FMVSS 208 and Euro NCAP				■	■			
Crashworthy Car Body disinterested, Simulation, Optimization						■	■	
Hot stamping with LS-DYNA		■	■					
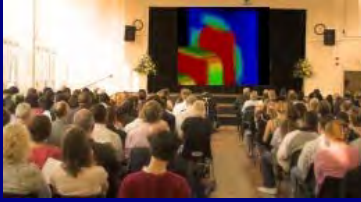
http://www.eta.com **for training dates and additional information**

Introduction to DYNAFORM

Introduction to DYNAFORM for sheetmetal forming applications. Includes Die Face Engineering and Blank Size Estimation tutorials.

Duration: 2 day course

Using PreSys with NISA

An introduction to PreSys for finite element modeling and the NISA finite element solver. This course will teach the student how to use PreSys to create their finite element model, set up a NISA simulation and review the results of the simulation. Workshop problems will be used to demonstrate of the principles discussed in the course material.

Duration: 1 day course

Introduction to PreSys

An introduction to the PreSys software for finite element modeling and results visualization. This course provides the basics for creating finite element model from CAD data, property definition and analysis preparation and visualization of simulation results. Workshop problems will be used to demonstrate all of the principles discussed in the course material.

Duration: 1 day course

Introduction to LS-DYNA

This course is intended for the new user, or those who might like a refresher on the basics of creating, running, debugging and analyzing an LS-DYNA model. The course will be in a lecture/workshop format, with the user running example models and post-processing the results.

Duration: 2 day course

Please contact support@eta.com This e-mail address is being protected from spambots. You need JavaScript enabled to view it , call (248) 729-3010, or register online to reserve a seat at the desired training session. Space is limited, so please reserve a seat as early as possible.

The following question was sent by Uli Franz of DYNAmore – DYNAmore is headquartered in Germany. http://www.dynamore.de

You may answer the questions and write the answers in the boxes below. The LS-DNYA Keyword User's Manual will help you to succeed.

C	O	N	T	R	O	L	_																			
C	O	N	T	R	O	L	_																			
M	A	T	_																							
M	A	T	_																							
C	O	N	T	R	O	L	_																			
C	O	N	T	R	O	L	_																			

QUESTIONS

Line 1:   In which card can you specify that a highly distorted shell element will be deleted if its Jacobian is negative?

Line 2:   Where can you set a flag for additional NaN checks in the force and moment arrays?

Line 3:   Which material in LS-DYNA has a hyphen in the name?

Line 4:   What is the name of material 181?

Line 5:   In which card can you switch bulk viscosity for shells on?

Line 6:   What is the variable (flag) name to force the contact algorithms to take the initial penetrations during the simulation into account, instead of moving the nodes to a non-penetration position at the beginning of a simulation?

Line 7:   In which card can you switch the time-stepping scheme?
 E.g. from explicit to implicit?

ANSWER Question #1:

QUESTION BOX

C	O	N	T	R	O	L	_															
C	O	N	T	R	O	L	_															
M	A	T	_																			
M	A	T	_																			
C	O	N	T	R	O	L	_															
C	O	N	T	R	O	L	_															

Answers:

C	O	N	T	R	O	L	_	S	H	E	L	L							
C	O	N	T	R	O	L	_	S	O	L	U	T	I	O	N				
M	A	T	_	M	O	O	N	E	Y	-	R	I	V	L	I	N	_	R	U	B	B	E	R
M	A	T	_	S	I	M	P	L	I	F	I	E	D	_	R	U	B	B	E	R	/	F	O	A	M
C	O	N	T	R	O	L	_	B	U	L	K	_	V	I	S	C	O	S	I	T	Y
I	G	N	O	R	E														
C	O	N	T	R	O	L	_	I	M	P	L	I	C	I	T	_	G	E	N	E	R	A	L

DYNAmore GmbH

Carrying out a FE simulation using an explicit time integration method, one of the main issues for stability is the upper limit of the time step, the so-called critical time step. This critical time step depends on a characteristic length of the elements and the wave speed, which is determined by the density and the stiffness of the material. The limitation of the time step is also known as the Courant-Friedrichs-Lewy (CFL) condition.

It is a necessary condition to make useful simulations. You may check detail in literature or at:

http://www.dynasupport.com/tutorial/ls-dyna-users-guide/time-integration.

By default, the appropriate time step is determined by LS-DYNA automatically. In addition, LS-DYNA allows the user to modify the time step size with several parameters.

A. Please download the LS-DYNA input file beam.k
   (http://www.dynasupport.com/links/fea-information-examples/beam.k)

B. Answer the following questions.

**The exercise intends to explain the CFL condition
   and its application in LS-DYNA.**

1.) Compute the critical time step of the beam and compare your result with the "smallest timestep" in LS-DYNA. Why is there a difference?

2.) How can you prompt LS-DYNA to output the analytical solution?

3.) Carry out a simulation, where DT2MS=-1.0E-3. Why is this time step not considered?

4.) How can you define a maximum time step size?

5.) What happens, if you carry out a simulation without mass scaling and TSSFAC=1.1?

Solutions:

1a.) analytical:
dt=l*sqrt(density/youngs_modulus)=10.0*sqrt(0.785E-5/210.0)=0.19334E-02

LS-DYNA: dt=0.17372E-02

1b.) Scale factor for computed time step is by default TSSFAC=0.9

2.) Include *CONTROL_TIMESTEP keyword and define TSSFAC=1.0

3.) LS-DYNA chose the greatest possible time step and the computed critical time step
is greater than the one defined via DT2MS.

4.) Define a load curve that limits the maximum time step size
(*CONTROL_TIMESTEP, LCTM)

5.) Error termination due to "out-of-range (rotational) velocities" --> simulation gets
unstable

Quiz:

Taking a closer look on the material models in LS-DYNA, one can distinguish between two different kinds of material laws. On the one hand, one can find material laws, which relate stresses to strains. Keeping thereby in mind that there exist several stress and strain measures, care must be taken on the interpretation of force vs. displacement or stress vs. strain diagrams. For example, the basis for a parameter identification of an elastoplastic material model in LS-DYNA is typically a true stress true strain curve, cf. http://www.dynasupport.com/tutorial/computational-plasticity . On the other hand, there exist material laws, which relate directly forces to displacements or moments to rotations, respectively. These types of material laws are known as discrete material laws and require other element formulations in LS-DYNA, cf. the keyword *SECTION_DISCRETE or beam element form six.

Please download the LS-DYNA input file beam2.k
(http://www.dynasupport.com/links/fea-information-examples/beam2.k/at_download/file)
and answer the following questions. The exercise intends to give a better understanding of the above mentioned two classes of material formulations:

1. Take a look on the force vs. displacement relationship of the beam. Why can you find a nonlinear relation between these two variables, although an elastic material law is chosen? How can you check this?
2. Reference within the part keyword of the beam the material and section keywords with the id two. Carry out the simulation and take a further look on the force vs. displacement curve. Why is there a linear relation between these two variables now?
3. Define a further node at the origin of the coordinate system and replace the second node of the beam by this new node. Carry out a simulation and take a look on the force vs. displacement curve of the beam. Why can you observe no difference in comparison with the beam of finite length?
4. Take a look on the critical time steps of the discrete beams with and without a certain length. Why can you find the same value for both configurations?
5. Divide the mass of the beam in halves. What is the consequence for the critical time step?
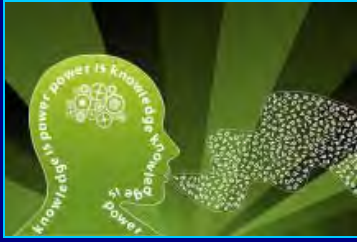
Solutions:

1.) *MAT_ELASTIC provides a material formulation between true stresses and strains and not between engineering stresses and strains or forces and displacements. If you plot true stresses vs. strains, you will find the linear relation between them.

2.) Now a discrete material law is used, whereby the force is directly related to the displacement.

3.) As a discrete material law relates forces to displacements and not to strains, where generally the displacement is related to the length of an element, the resulting force depends only on the displacement between the two nodes of the beam element and not on the length between them.

4.) The critical time step of an element with a discrete material law is independent of the length of the element.

5.) The critical time step decreases. The critical time step of a element with an discrete material law depends on the mass and the stiffness.

Students

Formula One

If your University is working on a Formula One please consider listing it here. Send the information to mv@feainformation.com

Brigham Young University:

PACE Formula One Race Car Project begins a new year. The prior year was a success of many collaborative efforts. C. Greg Jensen, Professor, Mechanical Engineering, Brigham Young University and his students studied crash analysis and built a model. Among the collaboration Suri Bala led the LS-DYNA effort with his software, D3VIEW, an online collaboration tool for LS-DYNA projects

Last Year's Article   http://www.lstc.com/pdf/a_pace_car.pdf

Slovak University of Technology

Stuba GreenTeam is a racing team representing Slovak University of Technology in Bratislava. Our goal is to develop, design and manufacture a racing, participate on Formula Student electric and take the challenge to compete other racing teams from all over the world.   www.sgteam.eu -

Universidade Federal De Santa Catarina - Fórmula UFSC
http://www.formula.ufsc.br

Canada   Metal Forming Analysis Corporation - MFAC -
               Contact: galb@mfac.com

USA   Engineering Technology Associates, Inc
               Contact: sales@eta.com

USA   SE&CS
               Contact: len@schwer.net

USA   Predictive Engineering
               Contact: george.laird@predictiveengineering.com

USA   CAE Associates
               Contact: info@caeai.com

USA   AEG Product Engineering Services
               Contact: support@engineering-group.com

USA   APACS Services Inc.
               Contact: apacs@comcast.net

DENMARK	FaurConAps Contact: faurholdt@faurcon.com
FRANCE	ALYOTECH TECHNOLOGIES Contact: nima.edjtemai@alyotech.fr
FRANCE	ALLIANCE SERVICES PLUS Contact: v.lapoujade@asplus.fr
Germany	CADFEM GmbH Contact: ls-dyna@cadfem.de
Germany	DYNAmore Contact: uli.franz@dynamore.de
ITALY	EnginSoft SpA Contact: info@enginsoft.it
Netherlands	Infinite Simulation Systems, B.V Contact: j.mathijssen@infinite.nl
Sweden	Engineering Research AB Contact: sales@erab.com
UK	OVE ARUP & PARTNERS Contact: brian.walker@arup.com

Australia	Leading Engineering Analysis Providers, LEAP
	Contact: info@leapaust.com
China	Ove Arup & Partners
	Contact: stephen.zhao@arup.com
China	ETA China
	Contact: lma@eta.com.cn
China	Shanghai Hengstar Technology Corp.
	http://www.hengstar.com
INDIA	nHance Engineering Solutions Pvt Ltd
	Contact: lavendra.singh@arup.com
INDIA	EASi Engineering
	Contact: rvenkate@easi.com
JAPAN	JSOL Corporation
	Contact: cae-info@sci.jsol.co.jp
JAPAN	Itochu Techo-Solutions Corp.
	Contact: ls-dyna@ctc-g.co.jp
JAPAN	LANCEMORE
	Contact: info@lancemore.jp
KOREA	THEME Engineering
	Contact: wschung@kornet.net
KOREA	KOREAN SIMULATION TECHNOLOGIES
	Contact: young@kostech.co.kr

Alpha Order by Country

Australia	Leading Eng. Analysis Providers - LEAP http://www.leapaust.com.au/ info@leapaust.com.au
Canada	Metal Forming Analysis Corp - MFAC http://www.mfac.com/ galb@mfac.com
China	ETA China http://www.eta.com.cn/ lma@eta.com.cn
China	OASYS Ltd. (software house of Arup) http://www.oasys-software.com/dyna/en stephen.zhao@arup.com
China	Shanghai Hengstar Technology Corp. http://www.hengstar.com
France	ALYOTECH TECH. http://www.alyotech.fr nima.edjtemai@alyotech.fr
France	ALLIANCE SVCE. PLUS - AS+ http://www.asplus.fr/ls-dyna v.lapoujade@asplus.fr
Germany	CADFEM http://www.cadfem.de/en lsdyna@cadfem.de
Germany	DYNAmore http://www.dynamore.de/ uli.franz@dynamore.de
Greece	PhilonNet Engineering Solutions http://www.philonnet.gr stavroula.stefanatou@philonnet.gr

India	OASYS Ltd. (software house of Arup) http://www.oasys-software.com/dyna/en lavendra.singh@arup.com
India	EASi Engineering http://www.easi.com/ rvenkate@easi.com
India	CADFEM Eng. Svce India http://www.cadfem.in/ info@cadfem.in
Italy	EnginSoft SpA http://www.enginsoft.it/ info@enginsoft.it
Japan	JSOL Corporation http://www.jsol.co.jp/english/cae cae-info@sci.jsol.co.jp
Japan	ITOCHU Techno-Solutions Corp. http://www.engineering-eye.com/ ls-dyna@ctc-g.co.jp
Japan	FUJITSU http://jp.fujitsu.com\solutions\hpc\app\lsdyna\

Korea	Theme Engineering http://www.lsdyna.co.kr/ wschung@kornet.net
Korea	Korea Simulation Technologies http://www.kostech.co.kr young@kostech.co.kr
Netherlands	Infinite Simulation Systems, BV http://www.infinite.nl/ j.mathijssen@infinite.nl
Sweden	Engineering Research AB http://www.erab.se/ sales@erab.se
Taiwan	Flotrend Corporation http://www.flotrend.com.tw/ gary@flotrend.tw
Russia	State Unitary Enterprise –STRELA info@ls-dynarussia.com

United Kingdom	OVE ARUP & PARTNERS http://www.oasys-software.com/dyna/en/ dyna.sales@arup.com
USA	Livermore Software Tech. Corp. - LSTC http://www.lstc.com/ sales@lstc.com
USA	Engineering Tech. Assc. Inc. – ETA http://www.eta.com/ sales@eta.com
USA	DYNAMAX http://www.dynamax-inc.com/ sales@dynamax-inc.com

The LSTC Models Development Team has decided to formalize the procedure of distributing news about LSTC's models in order to handle the increased number of recipients.

Previously, you have received news about LSTC's models, via direct e-mails from Sarba Guha. The previous mailing list was retired, as of the first invitation to the new mailing list.

If you arel interested in receiving the LSTC Models News:

1. **Subscribe** to this new mailing list at the following website:

    http://listserv.lstc.com/mailman/listinfo/lstc_models_news

2. **Receive:** You will receive an e-mail from the mail program
         with the following sender address:
            [lstc_models_news-request@listserve.lstc.com]
         with the subject line
            "confirm" and an alpha-numerical code.

3. **Confirm:** Per the instructions in that e-mail, please confirm your e-mail address

4. **Address Book:** To ensure receipt of emails from LSTC Models News, add
            [lstc_models_news@listserv.lstc.com]
            to your address book or safe list

5. **Confidentiality:** Your e-mail address will not be distributed outside of LSTC
             and only used for the LSTC Models News.

Thank you for your interest in our products!

Best regards, LSTC Models Development Team

Shape The Future LS-TaSC™ Topology Design

The future of time to market in concept design, for engineering analysts optimizing structures using LS-DYNA. LS-TaSC (**T**opology **a**nd **S**hape **C**omputation) delivers results with the LS-DYNA implicit and explicit highly scalable solutions.

LS-TaSC handles topology optimization of large non-linear problems, involving dynamic loads and contact conditions.

Above: Initial Geometry & Loading Conditions
Right: Final Design with casting geometry definitions

In LS-TaSC, nonlinear phenomena, such as contact definitions, do not require special consideration.

General Capabilities

Solid design using first-order hexahedrons and tetrahedral elements
Shell design using first-order quadrilateral and triangular elements
Global constraints
Multiple load cases
Tight integration with LS--DYNA
Large models with millions of elements

Geometry Definitions

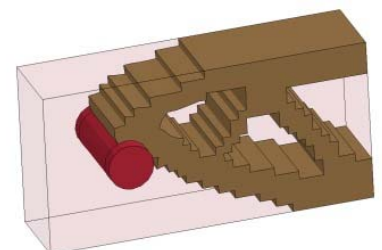Multiple parts
Extrusions
Casting, one sided
Casting, two sided
Symmetry

LS-TaSC is included at no additional cost as an additional module with LSTC's suite of software:

LS-DYNA
LS-OPT
LS-PrePost
LSTC ATD
Barrier Models

Postprocessing

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

No special treatment is required for nonlinearities such as contact.

LS-TaSC™
Livermore Software Technology Corp.

sales@lstc.com

LS-TaSC™ for Implicit And Explicit Results