JULY
2006

feaINFORMATION

WWW.FEAINFORMATION.COM

COVER STORY

MSC Software Corporation

Newman Haas Racing and Enterprise

Simulation - The Winning Combination

PRODUCT SPOTLIGHT

The ANSYS®

Multiphysics™ Solution

PRODUCT SPOTLIGHT

ESI Group's

Vehicale Trim Modeling

with RAYON-VTM

FEA Information Worldwide Participants

Contents

Editor:
Trent Eggleston
Managing Editor:
Marsha Victory
Technical Editor:
Art Shapiro
Graphic Designer:
Wayne L. Mindle

Technical Writers:
Dr. David Benson
Uli Franz
Dr. Ala Tabiei
Suri Bala
Technical Consultants:
Steve Pilz
Reza Sadeghi

FEA Information Announcements

2007 LS-DYNA Conference

With the success of the 9th International Conference held June 2006 we are pleased to announce the 2007 LS-DYNA Conference will be held in Gothenburg, Sweden, hosted by ERAB. 2008 LS-DYNA Conference will return to Detroit, MI, US, hosted by LSTC.

6th European LS-DYNA Conference
May 28-29, 2007
http://www.erab.se/conference2007/

Other LS-DYNA Conferences/Workshops 2006

Sept 19-20	**JAPAN LS-DYNA Users Conference 2006** Tokyo, Japan Hosted by JRI
Sept 25	**11th Korea LS-DYNA Users Conference 2006, Seoul, Korea Hosted by Theme Engineering Inc.**
Oct 12-13	**LS-DYNA Users Meeting in Ulm.** Hosted by DYNAmore
Oct 25-27	**2006 CADFEM Users Meeting** International Congress on FEM Technology Stuttgart area - Germany

1st Mikkeli Computational Mechanics Workshop'2006

Additionally, we are pleased to announce:

- **1st Mikkeli Computational Mechanics Workshop'2006**
- **Mikkeli Univ. of Applied Sciences Research Center YTI, Finland.**
- **Among the sponsors are LSTC and FEA Information, Inc.**
- **August 7-11,2006**
- www.compmechworkshop.com

Sincerely, ***Trent Eggleston & Marsha Victory***

DYNA*more* GmbH

Press Release 2 / 2006
The Group Research of Volkswagen AG is using LS-DYNA for Metal Forming Analysis - *Stuttgart, June 2006* –

DYNA*more* GmbH, the Distributor of LS-DYNA in Germany and some neighbouring countries is announcing, that with the Volkswagen AG another major car manufacturer will use their core product, the crash and metal forming simulation software package LS-DYNA developed by LSTC based in Livermore, USA

Figure Volkswagen AG: Complex deep drawing simulation

.

LS-DYNA will be used by the Group Research of Volkswagen AG for the development and improvement of innovative production concepts. Dr. Kulp from Group Research of Volkswagen, who has chosen LS-DYNA after a number of detailed benchmark studies, says, that the decision for LS-DYNA was based on the following facts: “With LS-DYNA we use a software tool, that has numerous advantages in the simulation of modern complex metal forming processes like the forming and spring back of high-strength steels or thermal deep drawing processes. Another application at Group Research of Volkswagen will be the determination of stamping forces in sheet metal forming. In particular, for this application we expect considerable progress and improvements for our processes. In general, LS-DYNA was selected since it provides a number of seamlessly integrated numerical methods in one executable, which allows comprehensive simulations of forming processes. “

Ulrich Franz, Managing Director of DYNA*more* GmbH, states: “The decision of the Group Research of Volkswagen is a big success for us, as it strengthens our position at the Volkswagen Group, considerably. The basis of Volkswagen’s decision was the continuing competent technical work delivered by DYNAmore and the close and flexible cooperation between Volkswagen, DYNAmore and LSTC for the development of

new features inside the LS-DYNA software package."

John O. Hallquist, President of LSTC, adds: "Our close partnership with DYNAmore is the key for our joint success at Volkswagen. We are very delighted about the decision of the Group Research at Volkswagen and will put strong effort to support their applications. Our main competitive advantages like the flexible usage and the wide range of application capabilities of LS-DYNA from static to explicit time step integration methods and the superior performance and scalability on parallel hardware architectures helps us to increase our world-wide market share. The decision of Group Research of Volkswagen encourages us to proceed providing one code for multiple applications. "

Contact Address:

DYNA*more* GmbH –
Industriestr. 2, D-70565 Stuttgart –
Tel.: 07 11 - 45 96 00 – 0
Fax: 07 11 - 45 96 00 – 29
e-mail: info@dynamore.de
Internet: www.dynamore.de

MSC Software Corporation

Newman Haas Racing and Enterprise Simulation - The Winning Combination

Six-time Champ Car World Series winner Newman/Haas Racing has refocused its simulation activities to include tire models and rough road surfaces in its vehicle dynamics simulations for predicting handling. Tires and rough road surfaces are crucial for damper setup and for predicting and understanding how a car will handle on a given track. Accurate tire models are difficult and costly to generate and the ability to run vehicle dynamics predictions and damper simulation over a rough surface can be cost prohibitive.

Detail of the MTS/Swift 20T wheel force transducer (magenta) for measuring forces and moments (torque) installed on the 2005 NRH Lola Champ-Car.

Since its first season in 1983, Newman/Haas Racing has collected many wins, poles and track records with talented drivers such as Mario and Michael Andretti, Nigel Mansell, Cristiano da Matta and most recently, Sebastien Bourdais. These have resulted in one of the most successful racing teams in the history of Champ Car.

The MTS Swift Transducer interface mounted to the right sidepod of the 2005 NHR Lola Champ-Car. Cable from left rear tire connects to computer mounted on right side of car for logging real time data acquisition.

Owners, Paul Newman, one of the most famous actors in the world, and Carl Haas, *"One of the most powerful men in the history of auto racing - USA Today,"* have won six Champ Car Championships, and started their 24th season with a win in the 2006 Toyota Grand Prix of Long Beach.

The 2005 NHR Lola Champ-Car being prepared to perform objective handling maneuvers on a skidpad (vehicle dynamics area). Tire warming blankets shown preheat the tires to oper-

ating temperatures prior to testing on the semi-smooth surface. The dusty, imperfect surface approximates a race track surface. Computers mounted on left and right sides of car are connected to each axle for measurements.

Building cars to achieve and maintain such success requires continuous innovation and engineering excellence. Using a manufacturer's tire model that only predicts smooth surface performance instead of replicating the rough surface of a real track produces inherently inaccurate results. Obviously, the need is for accurate results leading to improved handling and the potential for more wins.

Tire Modeling and Simulation

One or more optical slip sensors are used to measure the vehicle side-slip angle as the car moves through a curve. It captures how the grains are moving in the road in relation to fore and aft and side to side. Together with the wheel-force transducers the slip sensors calculate the longitudinal and lateral slip of each tire. This enables Newman-Haas Racing to regress their own tire models for use in ADAMS simulations

"The first thing to understand whenever generating a vehicle model is that the most important and the most difficult part of the car to characterize are its tires," said Brian Lisles, general manager, Newman Haas Racing. "Typically, racing teams rely on tire companies to provide tire data, but tire testing machines are expensive to operate and the tire companies don't necessarily generate the required data or in the manner required by the racing team."

Newman/Haas Racing uses equipment that allows the tires to be tested on the car and measure the forces and moments produced by tires. In order to understand exactly how the tire is presented to the road, the team uses ADAMS/Motorsports to emulate the maneuvers performed during the test, thereby producing and defining the same conditions that existed when the tire data was measured.

This involves running the vehicle in constant-radius circles on a skid-pad at gradually-increasing speeds, running at fixed speed with either stepped or gradual steering inputs or emulating an on-track corner-entry/mid-corner/exit maneuver to gather steady state and dynamic tire information. A skid-pad is a large flat area made of the same material as a race track.

All of the data is measured and recorded and then regressed to produce a tire model. The model is input to ADAMS to make sure its behavior matches the observed behavior of the tire during the physical test.

"Effectively, we do our own tire testing to create our own tire models, which is time and cost effective," said Lisles.

Rough Road Handling

In the past, tire data measurements were made on a smooth surface, so

when dynamic simulations were run, the user must assume a smooth surface. All the irregularities and bumps inherent in a real race track are absent - an idealization applied when performing pure handling analysis.

"We use ADAMS as a virtual a 7-post rig, i.e. put the car on four hydraulic platforms, vibrate the platforms underneath the wheels, and observe the response of the car," said Lisles. "Consumer cars use four post test methods, but a racing car requires three more actuators to apply the downforce, which is the aerodynamic load applied to the body. We built an ADAMS model of the vehicle/rig system and then validated the virtual vehicle and test rig against the physical ones."

This is a very straightforward thing to do. Having validated the results, the standard vehicle handling model can be run on a rough road instead of just a smooth road. This allows damper (shock absorber) development and setup to take place virtually, instead of requiring a four-post shake test.

Conclusion

Newman/Haas Racing has been using ADAMS to process independently-gathered tire data that can be applied to a dynamic vehicle simulation on a rough surface. This enables their engineers to develop and tune damper settings for cornering over a rough road.

"There are several ways of achieving our objectives including measuring real car data and working backwards to construct why the car responds the way it does. The surface determines how the car responds. The information can be regressed from the measurements to what the road surface would be to make the car respond in that manner."

To achieve this, a high fidelity model is required, which is why Newman/Haas Racing validates the ADAMS/Motorsports handling model on a virtual seven-post shake rig. The virtual model provides the flexibility of applying any input to the vehicle to produce a simulated response.

By making a virtual rig in ADAMS and correlating the results, the tire parameters can be adjusted until they match physical test results. According to Lisles, the engineers get the same response with the ADAMS model as measured on a test rig.

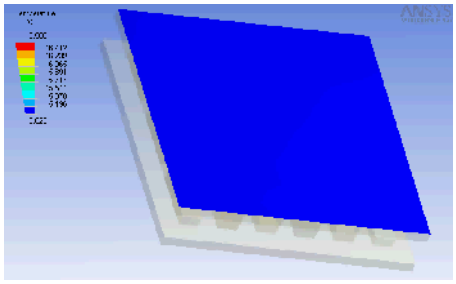
"In an ideal world, we could measure the track on a Friday afternoon, send it away and Saturday morning process the information, run the analysis and have suggestions for solving any problems," said Lisles. "We were trying to improve the vehicle dynamics simulation to answer more and more questions. We kept finding the tire information was out of date and wasn't what we wanted. With ADAMS, we found a way of overcoming the inaccurate data problem using the current tires and eventually we'll be able to do this very quickly."

"Instead of using expensive rig test time to test tires, we can just go and run a series of standard tests and get the information," said Lisles. "The nice thing is you are running on a real surface with a real car at real speeds. It's the real thing compared to just doing it on a tire testing machine."

The ANSYS® Multiphysics™ solution

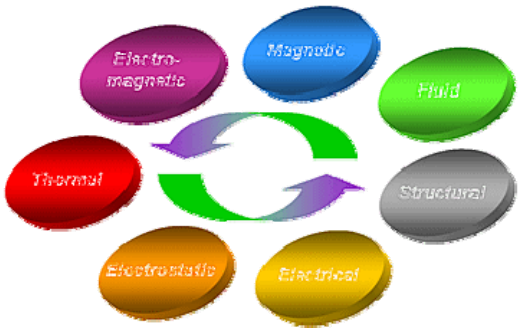
The ANSYS® Multiphysics™ solution provides the analysis industry's most advanced physics technology, combining structural, thermal, CFD, acoustic and electromagnetic simulation capabilities in a single software product.

With the ANSYS Multiphysics solution, you are getting the core physics of the entire ANSYS simulation suite in one convenient package! Applications involve everything from rotating machines (motors and alternators), sensors and actuators, power generators and transformer systems, and Micro Electro Mechanical Systems (MEMS).



An ANSYS Multiphysics coupled field analysis of a thermoelectric (Peltier) effect cooler

The ANSYS Multiphysics solution allows a user to combine the effects of two or more different, yet interrelated physics, within one, unified simulation environment



Coupled Physics

The ANSYS Multiphysics solution provides two fundamental, powerful methods to couple physics together — Direct and Iterative:

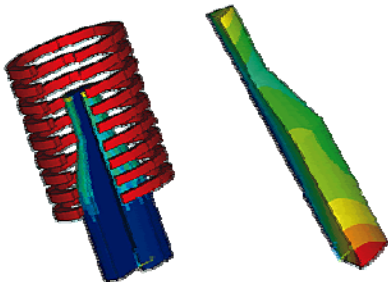
Direct

- Solves Coupled physics equations directly
- Solfes each physics simultaneously

Iterative

- Physics are coupled by passing loads across physics field interfaces
- Solves each physics simultaneously or sequentially
- At least two iterations, one for each physics, in sequence, are needed to achieve a coupled response

Providing both coupling methods allows ANSYS Multiphysics to address an extremely broad range of analysis applications



Multi-field solver example: Electromagnetic-Thermal-Mechanical analysis of induction heating process. Left image shows induced eddy currents in sample, right image shows resultant displacement due to thermal expansion.

Multiphysics Solution Benefits

ANSYS Multiphysics solution is unique and powerful with thousands of features, high performance and reliability to meet your analysis requirements. Can you afford the risk of not factoring in additional physics effects into your engineering process?

- Reduced assumptions that question certainty and compromise accuracy
- Analysis closely match reality – ANSYS Multiphysics is a virtual prototyping lab on your desktop
- Lower cost: Fewer analysis software tools to purchase, learn & manage
- Lower cost: R&D process compression
- Lower cost to slight modifications.

ESI Group's Vehicle Trim Modeling with RAYON-VTM

RAYON-VTM (Vehicle Trim Modeler), ESI Group's leading-edge technology makes possible low frequency vibro-acoustic analysis of fully trimmed structures that can be deployed as part of an industrial vehicle refinement program.

RAYON-VTM represents one of the latest innovative solutions for vibro-acoustic simulation, analysis and design.

In the automotive industry concurrent engineering practices make increasing demands for tools that allow engineers to perform virtual refinement, early in the design cycle, to meet ever decreasing time constraints.

RAYON-VTM is a unique simulation software that provides a process driven approach to the preparation of full vehicle FEM models created from a combination of structural, acoustic and trim FE meshes. The software automatically resolves incompatibilities of the differing FE meshes and integrates the result with modal information that facilitates calculation of interior noise. VTM offers fast turnaround of results that easily enables deployment of the solution as part of a vehicle refinement process, typically with overnight solution times.

This software is the result of a close collaboration between Renault, a leading car manufacturer, Trèves, a trim manufacturer and ESI Group. This partnership has led to a computationally efficient, validated tool, which provides significant cost savings coupled with increased refinement productivity. VTM allows engineers to optimize Noise Controls Treatments (NCT) in terms of weight, cost and effectiveness without the need to perform expensive and time consuming test cycles.

ESI Group is the only company offering a complete range of simulation software for vibro-acoustic simulation that covers the low, mid and high frequency spectrum. ESI Group proudly added in March 2006 RAYON-VTM to its product portfolio.

Virtual prototyping solutions reduce reliance on costly and time-consuming physical testing, speed up the product development process, and optimize the use of noise and vibration control materials.

ESI Group's vibro-acoustic portfolio includes:

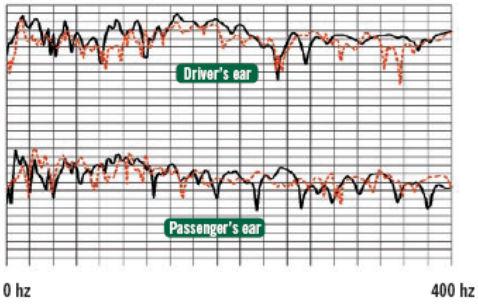
- **RAYON**, a low-frequency solution for characterization of complex noise sources and prediction of acoustic performance.
- **Vehicle Trim Modeler - VTM**: an automation module used for low frequency full vehicle sound package design.

- **AUTOSEA2**, the classic solution for high-frequency prediction of acoustic and vibration behavior of complex industrial systems.
- **FOAM X** identifies the acoustic properties of poro-elastic materials from impedance tube measurement.
- **NOVA** is an easy-to-use, flexible and powerful solution used to predict and optimize the absorption of acoustic materials and their integration into multi-layered structures.
- **PAM-VA One**: a newly integrated tool for the simulation and prediction of low, mid and high frequency vibro-acoustics response that combines statistical (AutoSEA2) and deterministic (RAYON) simulation methodologies in a common environment. PAM-VA One integrates work developed under the Structure-Borne Noise Module Consortium, an unprecedented full spectrum analysis that rigorously combines Finite Element Analysis (FEA) and Statistical Energy Analysis (SEA).

Benefits of RAYON-VTM:

- Full vehicle model of trimmed structure.
- Automatic harmonization of structural, fluid and trim meshes.
- Overnight computation to facilitate model refinement.
- Accurate interpretation of the influence of poro-elastic components on vibration and interior acoustics.
- Optimization of the design and positioning of trim inside a vehicle.

Sound Pressure Level



Courtesy of Renault

Yahoo Group Yammerings

Jim Kennedy
KBS2 Inc.
jmk@kbs2.com

Len Schwer
Schwer Engineering & Consulting Services
Len@Schwer.net

Jim Kennedy & Len Schwer plan to attend the 5th German LS-DYNA Forum, sponsored by DYNAmore, in Ulm Germany on 12 & 13 October. If you see us, please introduce yourself as a Yahoo Group participant, or Yammerings fan. Visit www.DYNAmore.de for more details.

Len Schwer will be conducting an LS-DYNA 'Discussion Group Forum' at the 77th Shock & Vibration Symposium, October 29 - November 3, 2006 at the Hyatt Regency Monterey in Monterey, CA. Visit www.saviac.org for more details.

This installment of "Yahoo Yammerings" features four questions, with responses, from the past month of postings to the LS-DYNA Yahoo Group:

1. *About Contact Card A the SOFT parameter?*
2. *How to determine acceleration at the center of mass?*
3. *Two questions related to contact stiffness.*
4. *How to use *DATABASE_D3DUMP keyword?*

Question: About Contact Card A the SOFT parameter?

What is the preferred setting for contact parameter SOFT? I have done a simple analysis to compare SOFT=0 or 1 with SOFT=2, and the results are different. The stress from SOFT=0 or 1 was larger than SOFT=2, moreover the velocity or stress history data from SOFT=2 were more noisy. Is it normal or abnormal? Did I make a mistake in my analysis?

Response by Jim Kennedy
I suggest that you look at the following notes:

http://www.dynasupport.com/Support/howto/contact.soft

also, sections 4.1, 4.2, and 6.3 from Suri Bala's notes

Bala, Suri, "Contact Modeling in LS-DYNA - Parts 1, 2, 3, and 4", Livermore Software Technology Corporation, 2001. (August, September, October, December, 2001 issues)

http://www.feapublications.com/pages/feanews2.shtml

Follow-up Question by Original Poster

I have read the notes and papers you suggest. But I am still confused about the results. When I use SOFT=2, I get noisy velocity and stress history data, and a negative sliding energy. But when I set SOFT=0 or 1, the results of these two settings are almost identical, and the history data was clear, no negative sliding energy. Which is more reliable? Could you give some suggestions?

Follow-up Response by Conrad Izatt

To answer your original question, there is no preferred option for the SOFT parameter - which option(s) work best depends entirely on the contact situation. Generally, I would recommend with SOFT = 0 unless the situation requires other wise.

I would suggest that in your case, since SOFT=0 or 1 seems to give reasonable answers and SOFT=2 doesn't, then stick with SOFT=0 or 1.

Follow-up Response by Sven Holcombe

Note that the following comments are not based on any deep knowledge of the algorithms used, rather just a few limited experiences of using the SOFT contact option.

Despite the name, the soft option doesn't dramatically change the contact calculation, it just calculates it using a different method. I tend to consider it as just using a different searching algorithm than normal contact. Therefore, the net result should be about the same, but perhaps the SOFT option performs slightly better in 'picking up' contacts that slipped through the regular algorithm. I think the LS-DYNA manual mentions two particular situations for which the SOFT algorithm is useful: surface-to-edge contacts (as opposed to surface-to-surface), and contact between very different materials, e.g. steel and foam.

Basically, I would just stick with the default contact, and if you run into a particular problem and you're not happy with the results (penetration, unusually spiky contact forces etc), then I'd see if SOFT contact works better.

Follow-up Response by Conrad Izatt

In general, I agree with your comments (even limited practical experience can be worth more than a deep understanding of theory).

However, depending on the situation, the SOFT parameter can significantly change the contact stiffness.

For the default SOFT=0, the contact stiffness is based on the element shape/size and the material bulk modulus. Therefore, for soft materials (i.e. low bulk modulus) the contact stiffness can be too low, allowing unrealistic surface penetrations, leading to possible contact instabilities.

For SOFT=1 or 2, the contact stiffness is based on the nodal masses and the time step. This means that the contact stiffness is maximized within the limits of numerical stability(i.e. like the usual time step calculation - such as a spring with masses on the ends). Note that this stiffness calculation does not consider the underlying material properties

(apart from the density for the nodal masses). Therefore, as long as the time step is not too large, or the nodal masses not too small, the SOFT=1 or 2 option can lead to much higher contact stiffnesses for soft materials.

The SOFT=0 or 1 options use the node-to-segment contact algorithm.

The SOFT=2 option uses the segment-to-segment contact algorithm, which can work better for 'difficult' contact problems, especially shell edge-to-edge.

Having said all that, my experience with a wide range of different contact problems tells me that there is no one set of contact parameters that will work in every situation. An understanding of the reason for a contact problem, and the effects of the contact parameters, will go a long way to solving the contact problem.

Follow-up Response by:
Sven Hocombe

Thanks for the details on the different flavors of SOFT option, knowing more about what's actually happening in the solver can only lead to better practices.

Question: How to determine acceleration at the center of mass?

How to find the acceleration at the center of mass in LS-DYNA? I don't have a node at the center of mass in my model. But, I would like to know the acceleration at the center of mass of the body. The body has 12000 nodes so it's difficult to find from the simple formula. Is there any way to find it in LS-DYNA?

Response by Len Schwer

A good approximation of the center of mass response, e.g. acceleration, velocity, & displacement, can be obtained with LS-PrePost.

Assign the portion of the model of interest a unique material ID and use the LS-PrePost ASCII MATSUM feature with any of the 'Rigid Body' sub-options. This provides a mass-averaged value of the quantity selected.

Note the keyword

*DATABASE_MATSUM must be present in the LS-DYNA input file to create the required ASCII file..

Response by Jim Kennedy

Some information on the location and inertias of the 'mass center' is provided in the d3hsp file.

If you want the acceleration at the center, you can place an accelerometer at that position using one of the several LS-DYNA options. If only a rough idea of the acceleration is good enough, look for an acceptable node that is near the center of mass as reported in the d3hsp file.

Question: Two questions related to contact stiffness.

From some simple tests, previous posts on this list, the LS-DYNA manuals, and the support web site, it appears that the nodal contact stiffness for the SOFT=0 and SOFT=1 options is inversely proportional to the element characteristic length. That is, if you have smaller elements in your contact

surface, the resulting contact stiffness will be higher.

Q1: Is the above statement true?

Q2: If the statement is true, is there a way to perform tests to investigate the convergence of a contact solution with respect to element size? If contact stiffness is a function of element size, then a simple element size convergence test (where everything remains the same except for element size) will never converge. In fact, it is conceivable that, with smaller elements, the solution may be worse due to increased contact stiffness!

Response by Conrad Izatt

To answer your first question, I believe that your statement is true in general, although the contact stiffness is dependent on the size and shape of the element e.g. area divided by minimum diagonal for shell elements.

Your second question is only true if there is no 'deformation' other than the penetrations of the contact surface. Usually, one would expect the material deformations to be sufficiently large to make any contact penetrations insignificant. Therefore, once the contact penetrations are a couple of orders of magnitude smaller than the material deformations, the contact stiffness will not make much difference and the solution (as far as contact stiffness is concerned) will have 'converged'

Follow-up to Original Question

A clarification with regard to my statement about element size – I was referring mainly to 8-node solid elements.

With respect to your second statement about deformation vs. penetration, perhaps my application is non-standard for LS-DYNA. My application generally does not have large material deformation and most of the stress occurs around contact regions. I have found that in both quasi-static and dynamic tests, the net contact forces and resulting kinematics of my models are not sensitive to element size, but the contact areas (and, therefore, contact pressures) are. I have refined my mesh to a practical limit and the change in contact areas does not seem to taper off at all.

Response by Conrad Izatt

With reference to solid elements, the contact stiffness is dependent on the area squared divided by the volume.

Your problem with contact pressures sounds similar to stress concentration problems. The finer the mesh (smaller the elements), the greater the stress concentration. What you are seeing suggests that the smaller elements are changing the local deformations in the contact area.

The only way that the contact stiffness could be significant is if the contact penetrations were of a similar order-of-magnitude as the local contact deformation. You could try increasing the contact stiffness artificially by increasing the penalty factor (although this may require a reduced time step to remain stable). Alternatively, you could try the SOFT=2 option.

Response by Original Poster

I checked one of my quasi-static tests and the deformation is on the order of 100 microns and the penetration is 30 to 50 microns, so they are in the same ballpark.

I had tried scaling up the contact stiffness. In general, a run with element

size “x” and 10 times contact stiffness produces the same contact area as a run with element size “x*0.5” with the default contact stiffness. In other words, doubling the mesh density had the same effect as increasing the contact stiffness by a factor of 10. The SOFT=2 option hasn’t worked out as well as the SOFT=1 so far.

Perhaps this is just the nature of the beast and I cannot use contact area as a variable to assess solution convergence with respect to element size in my application.

Question: How to use DATABASE_D3DUMP keyword?

My system crashes during my simulation and hence I wish to use the *DATABASE_D3DUMP keyword so that I will get the restart files during the simulation.

I tried to use the *DATABASE_D3DUMP keyword and gave the value for DT/Cycl as 0.01, but I didn’t get any restart files during the simulation. Please advise on the input value for this keyword, or what other keyword should be used to get restart files during a simulation.

Response by Len Schwer

I believe the option you want is RUNRSF, i.e.

*DATABASE_BINARY_RUNRSF
$ DT/CYCL LCD/NR

The second parameter, NR with default=1, allows you to specify how many running restart files will be written, and then reused in a cyclic fashion, e.g. NR=3 will write 3 running restart files and then re-use the first file to write the fourth running restart etc.

See page 9.9(DATABASE) [page 483/1564] in the Version 970 User Manual.

Response by Jim Kennedy

The output frequency is defined in cycles (time steps) for D3DUMP and RUNRSF, not a simulation time increment (DT) as for most other *DATABASE_options:

LS-DYNA Yahoo Groups

There are over 1850 subscribers from all over the world, and this list seems to grow by a hundred new subscribers ever few months; no small testament to the rapidly growing popularity of LS-DYNA. The group currently averages about 250 message per month, i.e. about 10 message per day. You can subscribe to the group by sending an email request to:
LS-DYNA-subscribe@yahoogroups.com
or by visiting the Yahoo Groups web site http://groups.yahoo.com

Generally, the quickest/best responses are to those questions posed with the most specifics. General questions such as “How do I use XXX feature?” either go unanswered, or are answered by Jim Kennedy with links to appropriate references in the growing LS-DYNA related literature, e.g. see the archive of LS-DYNA Conference proceedings at www.dynalook.com

LSTC Training Classes – 2006

The California office can accommodate 23 students at it's training center. The Michigan office has capacity for 12 students.

Classes are scheduled throughout the year at both locations. For the most current schedule visit www.lstc.com

Onsite training is also available.

Training Class	US $	Livermore, CA	Detroit, MI
Advanced LS-DYNA for Impact Analysis	$950	Sept 26-29	
Advanced Options in LS-DYNA	$750		Sept 05-06
ALE/Eulerian & Fluid/Structure Interaction in LS-DYNA	$750		
Concrete and Geomaterial Modeling with LS-DYNA	$750	Oct 24-25	
Contact in LS-DYNA	$750	Sept. 12-13	Aug 15-16
Introduction to LS-DYNA	$750	Aug. 01-04 Nov. 14-17	Oct 16-19 Dec. 11-14
Introduction to LS-OPT	$750	Nov. 07-10	
LS-DYNA Composite Materials	$750	Sept. 14-15	
LS-DYNA Implicit	$750	June 15-16	Sept. 07-08
LS-DYNA for Heat Transfer & Thermal-Stress Problems	$500		
Material Modeling Using LS-DYNA User Defined Options	$750	June 13-14	
MESH Free Methods in LS-DYNA (SPH and EFG)	$750		

EVENTS – 2006

If you want your event listed please send the information to:
mv@feainformation.com

2006	
August	**Altair Engineering's: South Asia CAE Users' Conference 2006**
Sept 19-20	**JAPAN LS-DYNA Users Conference 2006** Tokyo, Japan Hosted by JRI
Sept 25	**11th Korea LS-DYNA Users Conference 2006**, Seoul, Korea Hosted by Theme Engineering Inc.
Oct 12-13	**LS-DYNA Users Meeting in Ulm.** Hosted by DYNAmore
Oct 25-27	**2006 CADFEM Users Meeting** International Congress on FEM Technology Stuttgart area - Germany
Nov 14- 16	**Aerospace Design Expo 06** Anaheim, CA - US

EVENTS – 2007

6th European LS-DYNA Conference *May 28-29, 2007*

LS-DYNA Resource Page

Interface - Hardware - OS And General Information

Participant Hardware / OS that run LS-DYNA (alphabetical order).

LS-DYNA has been fully QA'd by Livermore Software Technology Corporation for All Hardware and OS listed below.

TABLE 1: SMP - Fully QA'd by LSTC	
AMD Opteron	Linux
FUJITSU Prime Power	SUN OS 5.8
FUJITSU VPP	Unix_System_V
HP PA-8x00	HP-UX 11.11 and above
HP IA-64	HP-UX 11.22 and above
HP Opteron	Linux CP4000/XC
HP Alpha	True 64
IBM Power 4/5	AIX 5.1, 5.2, 5.3
IBM Power 5	SUSE 9.0
INTEL IA32	Linux, Windows
INTEL IA64	Linux
INTEL Xeon EMT64	Linux
NEC SX6	Super-UX
SGI Mips	IRIX 6.5 X
SGI IA64	SUSE 9 with ProPack 4 Red Hat 3 with ProPack 3

LS-DYNA Resource Page

MPP Interconnect and MPI

FEA Information Inc. Participant's (alphabetical order)

Fully QA'd by Livermore Software Technology Corporation

TABLE 1: SMP - Fully QA'd by LSTC	
AMD Opteron	Linux
FUJITSU Prime Power	SUN OS 5.8
FUJITSU VPP	Unix_System_V
HP PA-8x00	HP-UX 11.11 and above
HP IA-64	HP-UX 11.22 and above
HP Opteron	Linux CP4000/XC
HP Alpha	True 64
IBM Power 4/5	AIX 5.1, 5.2, 5.3
IBM Power 5	SUSE 9.0
INTEL IA32	Linux, Windows
INTEL IA64	Linux
INTEL Xeon EMT64	Linux
NEC SX6	Super-UX
SGI Mips	IRIX 6.5 X
SGI IA64	SUSE 9 with ProPack 4 Red Hat 3 with ProPack 3

TABLE 2: MPP Interconnect and MPI			
Vendor	**O/S**	**HPC Intereconnect**	**MPI Software**
AMD Opteron	Linux	InfiniBand (SilverStorm), MyriCom, QLogic InfiniPath	LAM/MPI, MPICH, HP MPI, SCALI
FUJITSU Prime Power	SUN OS 5.8		
FUJITSU VPP	Unix_System_V		
HP PA8000	HPUX		
HPIA64	HPUX		
HP Alpha	True 64		
IBM Power 4/5	AIX 5.1, 5.2, 5.3		
IBM Power 5	SUSE 9.0		LAM/MPI
INTEL IA32	Linux, Windows	InfiniBand (Voltaire), MyriCom	LAM/MPI, MPICH, HP MPI, SCALI
INTEL IA64	Linux		LAM/MPI, MPICH, HP MPI
INTEL Xeon EMT64	Linux	InfiniBand (Topspin, Voltaire), MyriCom, QLogic InfiniPath	LAM/MPI, MPICH, HP MPI, INTEL MPI, SCALI
NEC SX6	Super-UX		
SGI Mips	IRIX 6.5	NUMAlink	MPT
SGI IA64	SUSE 9 w/ProPack 4 RedHat 3 w/ProPack 3	NUMAlink, InfiniBand, (Voltaire)	MPT, Intel MPI, MPICH

LS-DYNA Resource Page

Participant Software Interfacing or Embedding LS-DYNA

Each software program can interface to all, or a very specific and limited segment of the other software program. The following list are software programs interfacing to or having the LS-DYNA solver embedded within their product. For complete information on the software products visit the corporate website.

ANSYS - ANSYS/LS-DYNA

www.ansys.com/products/environment.asp

ANSYS/LS-DYNA - Built upon the successful ANSYS interface, ANSYS/LS-DYNA is an integrated pre and postprocessor for the worlds most respected explicit dynamics solver, LS-DYNA. The combination makes it possible to solve combined explicit/implicit simulations in a very efficient manner, as well as perform extensive coupled simulations in Robust Design by using mature structural, thermal, electromagnetic and CFD technologies.

AI*Environment: A high end pre and post processor for LS-DYNA, AI*Environment is a powerful tool for advanced modeling of complex structures found in automotive, aerospace, electronic and medical fields. Solid, Shell, Beam, Fluid and Electromagnetic meshing and mesh editing tools are included under a single interface, making AI*Environement highly capable, yet easy to use for advanced modeling needs.

ETA – DYNAFORM

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

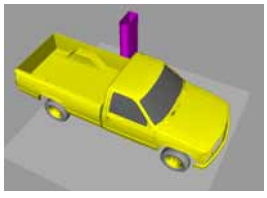
www.eta.com

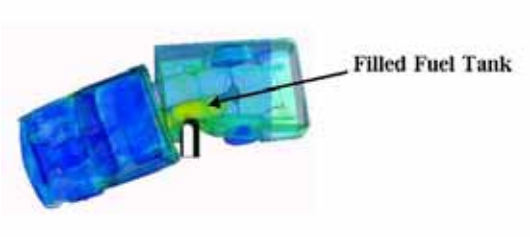
Streamlined CAE software package provides an event-based simulation solution of nonlinear, dynamic problems. eta/VPG's single software package overcomes the limitations of existing CAE analysis methods. It is designed to analyze the behavior of mechanical and structural systems as simple as linkages,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

MSC.Software - MSC.Dytran LS-DYNA

www.msc.software.com

Tightly-integrated solution that combines MSC.Dytran's advanced fluid-structure interaction capabilities with LS-DYNA's high-performance structural DMP within a common simulation environment. Innovative explicit nonlinear technology enables extreme, short-duration dynamic events to be simulated for a variety of industrial and commercial applications on UNIX, Linux, and Windows platforms. Joint solution can also be used in conjunction with a full suite of Virtual Product Development tools via a flexible, cost-effective MSC.MasterKey License System.





Side Impact With Fuel Oil Inside

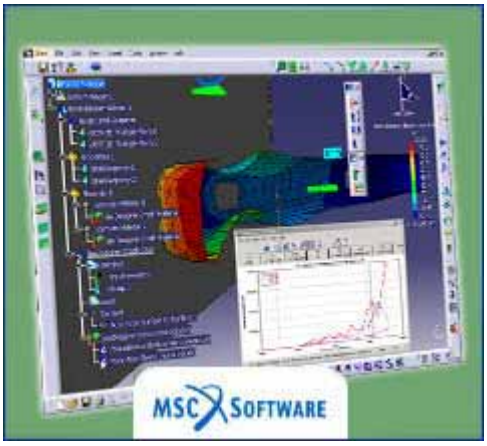
MSC.Software - MSC.Nastran/SOL 700

The MSC.Nastran™ Explicit Nonlinear product module (SOL 700) provides MSC.Nastran users the ability access the explicit nonlinear structural simulation capabilities of the MSC.Dytran LS-DYNA solver using the MSC.Nastran Bulk Data input format. This product module offers unprecedented capabilities to analyze a variety of problems involving short duration, highly dynamic events with severe geometric and material nonlinearities.

MSC.Nastran Explicit Nonlinear will allow users to work within one common modeling environment using the same Bulk Data interface. NVH, linear, and nonlinear models can be used for explicit applications such as crash, crush, and drop test simulations. This reduces the time required to build additional models for another analysis programs, lowers risk due to information transfer or translation issues, and eliminates the need for additional software training.

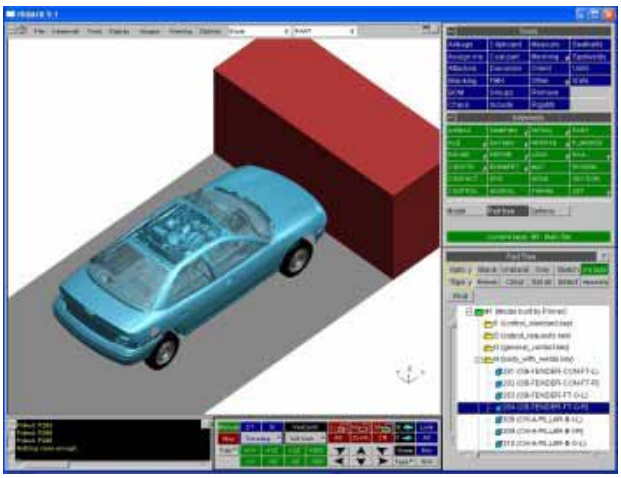
MSC.Software – Gateway for LS-DYNA

Gateway for LS-DYNA provides you with the ability to access basic LS-DYNA simulation capabilities in a fully integrated and generative way. Accessed via a specific Crash workbench on the GPS workspace, the application enhances CATIA V5 to allow finite element analysis models to be output to LS-DYNA and then results to be displayed back in CATIA. Gateway for LS-DYNA supports explicit nonlinear analysis such as crash, drop test, and rigid wall analysis.



Gateway products provide CATIA V5 users with the ability to directly interface with their existing corporate simulation resources, and exchange and archive associated simulation data.

Oasys software for LS-DYNA
www.arup.com/dyna



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.

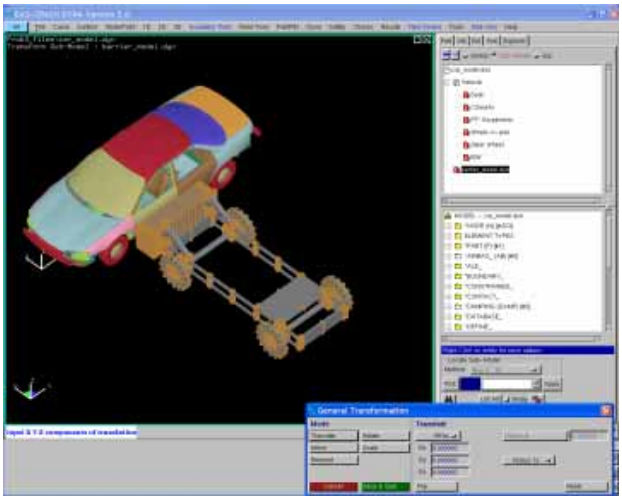
EASi-CRASH DYNA
http://www.esi-group.com/SimulationSoftware/EASi_CRASH-DYNA/

EASi-CRASH DYNA is the first fully integrated environment for crashworthiness and occupant safety simulations with LS-DYNA, and covers the complete CAE-process from model building and dataset preparation to result evaluation and design comparisons.

EASi-CRASH DYNA can be used for concept crash, FE crash and coupled rigid body/FE crash simulations in conjunction with MADYMO.

EASi-CRASH DYNA's main features include:

- Support of all keywords of LS-DYNA 970/971
- Powerful mesh editing features, such as automesh and remesh
- LS-DYNA/MADYMO coupling capabilities for pre- and post processing (support of MADYMO format till version 6.2.2)
- Model Assembler for organizing the model through sub assembly/sub models and included files
- Enhanced Weld tools for manipulation of connections and Weld comparison
- Simple dummy positing and seat belt routing
- Pre and Post processing in same environment
- Superpose and merge multiple models
- Animation and plotting
- Process compatible
- Full capability to handle IGES, CATIA V4, CATIA V5, UG and NASTRAN files



Hardware - Computing - Communication Products

www.amd.com

www.fujitsu.com

www.hp.com

www.ibm.com/servers/deepcomputing

www.intel.com

www.nec.com

www.sgi.com

www.qlogic.com

www.microsoft.com

Software Distributors

Alphabetical order by Country

Australia	**Leading Engineering Analysis Providers** www.leapaust.com.au
Canada	**Metal Forming Analysis Corporation** www.mfac.com
China	**ANSYS China** www.ansys.cn
China	**Arup** www.arup.com/eastasia/
China	**MSC. Software – China** www.mscsoftware.com.cn
Germany	**CAD-FEM** www.cadfem.de
Germany	**Dyna*More*** www.dynamore.de
India	**Altair Engineering India** www.altair-india.com
Italy	**Altair Engineering Italy** www.altairtorino.it
Italy	**Numerica SRL** www.numerica-srl.it
Japan	**Fujitsu Limited** www.fujitsu.com
Japan	**The Japan Research Institute** www.jri.co.jp
Japan	**CRC Solutions Corp.** www.engineering-eye.com
Korea	**Korean Simulation Technologies** www.kostech.co.kr
Korea	**Theme Engineering** www.lsdyna.co.kr

Software Distributors (cont.)

Alphabetical order by Country

Netherlands	**Infinite Simulation Systems B.V** www.infinite.nl
Russia	**Strela, LLC** www.ls-dynarussia.com
Sweden	**Engineering Research AB** www.erab.se
Taiwan	**Flotrend** www.flotrend.com.tw
USA	**Engineering Technology Associates** www.eta.com
USA	**Dynamax** www.dynamax-inc.com
USA	**Livermore Software Technology Corp.** www.lstc.com
UK	**Arup** www.arup.com/dyna/

Consulting and Engineering Services

Alphabetical Order By Country

Australia Manly, NSW www.leapaust.com.au	**Leading Engineering Analysis Providers** Greg Horner info@leapaust.com.au 02 8966 7888
Canada Kingston, Ontario www.mfac.com	**Metal Forming Analysis Corporation** Chris Galbraith galb@mfac.com (613) 547-5395
India Bangalore www.altair-india.com	**Altair Engineering India** Nelson Dias info-in@altair.com 91 (0)80 2658-8540
Italy Torino www.altairtorino.it	**Altair Engineering Italy** sales@altairtorino.it
Italy Firenze www.numerica-srl.it	**Numerica SRL** info@numerica-srl.it 39 055 432010
UK Solihull, West Midlands www.arup.com	**Arup** Brian Walker brian.walker@arup.com 44 (0) 121 213 3317
USA Austin, TX	**KBEC L.C** Khanh Bui kdbui@sbcglobal.net (512) 363-2739
USA Windsor, CA www.schwer.net/SECS	**SE&CS** Len Schwer len@schwer.net (707) 837-0559
USA Corvallis, OR www.predictiveengineering.com	**Predictive Engineering** George Laird (1-800) 345-4671 george.laird@predictiveengineering.com
USA Neenah, WI www.structuretechnology.com	**Structure Incorporated** Todd L. Peters (920) 722 7060 info@structuretechnology.com

Educational & Contributing Participants

Alphabetical Order By Country

China	Dr. Quing Zhou	Tsinghua University
India	Dr. Anindya Deb	Indian Institute of Science
Italy	Professor Gennaro Monacelli	Prode – Elasis & Univ. of Napoli, Frederico II
Russia	Dr. Alexey I. Borovkov	St. Petersburg State Tech. University
USA	Dr. Ted Belytschko	Northwestern University
USA	Dr. David Benson	University of California – San Diego
USA	Dr. Bhavin V. Mehta	Ohio University
USA	Dr. Taylan Altan	The Ohio State U – ERC/NSM
USA	Dr. Ala Tabiei	University of Cincinnati

FEA Information China Participants

Software, Hardware, Training, Consulting, Services

Arup **China: LS-DYNA sales, support, training**	**Kimbal Virdi** **Tel: +86 21 5396 6633** **Contact: Kimbal.virdi@arup.com** **Website: www.arup.com/dyna**

Ansys-China, Inc.	Tel: 86-10-84085558 Website: www.ansys.com.cn Contact: China@ansys.com.cn
Arup	Kimbal Virdi Tel: +86 21 5396 6633 Contact: Kimbal.virdi@arup.com Website: www.arup.com/dyna
Beijing Yuntong Forever CPC. Co. Ltd.	Tel: +86-10-82561200/01/03 Website: http://cpc.ytforever.com Sole Distributor of LINUX NETWORX, INC. (USA) in China Contact: service@ytforever.com
Engineering Technology Associates (China) Inc.	Martin Ma Tel: + 86-21-64385725 Contact: support@eta.com.cn
Hewlett-Packard Asia Pacific Ltd.	Jerry Huang Tel: +86-10-65645261 Contact: J.Huang@hp.com
IBM China	Ms. Ling WANG - Tel: +86-10-6539-1188 x4463 (T/L:901-4463) Website: http://www.ibm.com/cn/ Contact: wangling@cn.ibm.com
MSC. Software Corp.	Tel: +86-10-6849-2777 Website: www.mscsoftware.com.cn Contact: mscprc.contact@mscsoftware.com

FEA Information China Participants

Software, Hardware, Training, Consulting, Services

SGI China	Carl Zhang Tel: +86 -10 - 65228868 Ext. 3362 Contact: carl@sgi.com
Tsinghua University	Qing Zhou, PhD. - Professor Department of Automotive Engineering Beijing, 100084, China
Zhongfang Information Technology Ltd	Larry Liang Tel: +86-21-54973162 Website: http://www.cntech.com.cn Contact: info@cntech.com.cn
Zhong Guo ESI Co., Ltd	Yang Xiaojum Phone: +86 (020) 8235 6272 Contact : Yang Xiaojun

Informational Websites

The LSTC LS-DYNA Support site: www.dynasupport.com

LSTC LS-DYNA Support Site	**www.dynasupport.com**
FEA Informationwebsites	**www.feainformation.com**
TopCrunch – Benchmarks	**www.topcrunch.org**
LS-DYNA Examples (more than 100 Examples)	**www.dynaexamples.com**
LS-DYNA Conference Site	**www.ls-dynaconferences.com**
LS-DYNA Publications to Download On Line	**www.dynalook.com**
LS-DYNA Publications	**www.feapublications.com**
LS-DYNA CADFEM Portal	**www.lsdyna-portal.com.**

6th European LS-DYNA Conference
May 28-29, 2007
www.erab.se/conference2007/

Engineering Research Nordic will be hosting the 6th European LS-DYNA Users' Conference held at Quality Hotel 11 in Gothenburg, Sweden, May 28-29 2007.

The conference brings together LS-DYNA users, researchers and developers to discuss LS-DYNA simulations of complex mechanical problems. The conference also attracts a wide range of hardware and software companies wishing to showcase their special solutions when running LS-DYNA simulations.

The conference will cover LS-DYNA applications – Among them will be:

- Aerospace,
- Automotive,
- Biomechanics,
- Civil Engineering,
- Impact and Drop Testing,
- Material Modeling,
- Metal Forming,
- Nuclear,
- Occupant Pedestrian Safety,
- Seismic Engineering and more.

A software and hardware exhibition will be organized during the conference. Interested companies should contact Engineering Research Nordic.

We are expecting delegates and presentations from all over the world to discuss problems facing the industry today and in the future.

Dr. John Hallquist, will be a keynote speaker presenting new LS-DYNA features.

Specifications for Paper Submissions

- Paper length: 3.000 words maximum
- Abstract deadline: January 26th, 2007
- Notification of acceptance: February 9th, 2007
- Final paper deadline: April 1st, 2007

For information on Paper Submissions, Exhibit Booth and/or Available Sponsorships Contact:

- Dr. Marcus Redhe
  Engineering Research Nordic
  Brigadgatan 16
  SE-581 31 Linkoping
  Sweden

5th German LS-DYNA Forum
12th - 13th of October 2006 in Ulm, Germany

www.dynamore.de

The conference is an ideal forum for LS-DYNA users to share and discuss experiences, to obtain information on upcoming features of LS-DYNA and to learn more about new application areas.

The conference will be accompanied by an exhibition featuring the latest software and hardware developments related to LS-DYNA.

Conference language will be German and English.

The conference will take place in Ulm, the birthplace of Albert Einstein. The hotel is located right next to the famous river Danube. From the hotel you will have a marvelous view over the old town of Ulm, with the historic cathedral in the center.

Ulm can be easily reached from the airports of Frankfurt, Munich, and Stuttgart.

Keynote presentations on various topics will be :

Crash:	Jürgen Kohler	DaimlerChrysler AG
Occupant Safety:	Kurt Fograscher	Autoliv GmbH
Metal Forming:	Prof. Pavel Hora	Swiss Federal Institute of Technology Zurich
Optimization:	Dr. Stefan Schwarz	Dr. Ing. h.c. Porsche AG
Validation:	Dr. Len Schwer	Schwer Consulting
Material Modeling:	Paul Du Bois	Consultant
LS-DYNA Implicit:	Prof. Martin Pitzer	University Gießen
Linear Equation Solvers:	Prof. Ulrich Langer	University Linz
LS-DYNA:	Dr. John Hallquist	LSTC