We welcome Fujitsu Ltd. headquartered in Tokyo, Japan as a new participant.

FEA Information Company Participants
Please visit our Participant's Web Sites

(1) The contents of FEA Information International News is deemed to be accurate and complete. However, neither LSTC nor FEA Information Company guarantees or warrants accuracy or completeness of the material contained in this publication. **(2)** All trademarks are the property of their respective owners. **(3)** FEA Information International News is published for FEA Information Company, Livermore Software Technology Corporation and **delivered by e-mail free of charge** to those parties interested in the field of Finite Element Analysis and other Engineering Applications.

Explicit finite element methods were originally developed to solve problems in impact engineering, but they are currently used for many other applications such as sheet metal forming. The finite difference community of code physicists developed the first impact analysis programs, commonly called "hydrocodes." Many of them, such as HEMP and TENSOR, pioneered computational techniques that are used in DYNA3D and other nonlinear (implicit and explicit) finite element programs in engineering. In some cases, the finite difference methods differ from finite element methods only in that they are restricted to logically regular blocks of elements (i.e., the mesh blocks have their elements numbered by the element row and column). The plane strain stencil for HEMP is identical to the explicit finite element formulation for a 4-node plane strain quadrilateral with one point quadrature. In axisymmetric geometries the radial weighting is different. Most hydrocodes, developed at the national defense laboratories, are restricted and unavailable outside of the Department of Energy. In contrast, DYNA2D and DYNA3D, as engineering codes, were completely unrestricted and freely distributed through 1988 as public domain software.

Although the initial 1976 version of DYNA3D wasn't vectorized, the second release in 1979 benefited from a complete rewrite to take advantage of the Cray-1 supercomputer. For those too young to remember the Cray-1, it ran at 80 MHz, had 8 MB of RAM, and needless to say, it's too small and slow to run Microsoft Word. Although few computers today contain vector floating point units, their RISC architectures perform best with clean vectorized code. The analyses that the Cray-1 made possible were too large to interpret from printed numerical output. DYNA3D was therefore one of the first finite element programs to rely almost entirely on graphical post-processing instead of printed output.

Dynamic events usually occur over a short time period. For example, the duration of interest of a frontal automobile crash is around 80 milliseconds. Small time steps must be used to accurately resolve the complicated deformation history of the buckling sheet metal in the automobile. Implicit finite element methods are prohibitively expensive when thousands of time steps must be taken because of the cost of the solving their large sets of nonlinear equations. The explicit finite element solution is advanced in time, without solving any equations, by the central difference method

$$\dot{u}^{n+1/2} = \dot{u}^{n-1/2} + \Delta t \cdot M^{-1}(F^{external} - \int B^T \sigma dV)$$

$$u^{n+1} = u^{n-1} + \Delta t \cdot \dot{u}^{n+1/2}$$

where $F^{external}$ is the vector of applied forces associated with the boundary conditions and body forces, M is the diagonal mass matrix, B is the discrete gradient operator, and σ is the stress. Explicit finite element methods avoid the cost of inverting the mass matrix by using a diagonal, lumped mass matrix.

The original 1976 version of DYNA3D contained both linear and quadratic solid elements. The quadratic elements were extremely costly due to the number of operations they required per time step and their smaller stable time step size. The higher order elements were also impractical due to their numerical noise caused by the ad hoc way the mass was lumped to generate a diagonal mass matrix. Benchmarks demonstrated that constant stress elements with reduced integration and hourglass control were necessary for efficient, large-scale calculations. For a few problems, such as those involving spot welds with failure, no hourglass control method is entirely satisfactory, and fully integrated linear elements are today available in LS-DYNA.

Contact algorithms prevent surfaces from interpenetrating, and most nonlinear calculations require them. They can be categorized by their 1) contact search strategy, 2) contact and release conditions, and 3) contact force calculation. The contact search is a significant cost for most contact algorithms. The penalty method, which introduces a spring force to separate contacting surfaces, has proven to be the most versatile and robust method for contact with separation and slip in LS-DYNA.

LS-DYNA, which is based on the original 50,000 line DYNA3D code, is more than a decade old and approaching 2 million lines of code. LS-DYNA now incorporates both explicit and implicit methods; runs crash problems in massively parallel and shared memory environments; has adaptive, arbitrary Lagrangian Eulerian (ALE), SPH formulations; and heat transfer and fluids capabilities coupled to the solids solver. The strategies developed to make explicit finite element methods efficient have greatly contributed to the efficiency of all the recent additions.

Ala Tabiei[*] and Ivelin Ivanov[†]

The Center of Excellence in DYNA3D Analysis

Department of Aerospace Engineering and Engineering Mechanics

University of Cincinnati, OH 45221-0070, USA

THE COMPUTATIONAL MICRO-MECHANICAL MODEL

The Representative Volume Cell (RVC) approach is utilized in the micro-mechanical model development. The interlacing yarn pattern of the flexible weave fabric is depicted in the following figures. As a result of the deformations, the fill and the warp yarns are no longer orthogonal although at the unloaded state, they could be orthogonal (the angle between fill and warp direction is a user input parameter in the formulation).

The starting point for the homogenization of the material properties is the determination of the yarn stiffness matrices. The material of the yarn is assumed to be transversely isotropic. Initially, in free stress state, the discount factor is a small value and the material has very small resistance to shear deformation if any. In this way, the material behaves like a trellis mechanism with small resistance against the rotation of the yarns, corresponding to the friction between yarns. When the locking occurs, the fabric yarns are packed and they behave like elastic media.

The proposed micro-mechanical model is developed to account for the reorientation of the yarns. At any time the micro-model corresponds to some current state of the deformed RVC, when the directions of the yarns are different from the initial ones. The reorientation of the yarns involves geometrical non-linearity. The directions of the fill and the warp yarns in xyz-coordinate system are determined by the unit vectors $\{q_f\}$ and $\{q_w\}$, respectively. Initially, the yarn directional unit vectors are calculated from the initial values of the angles θ, β_f, β_w. The deformation gradient tensor, $[F]$ can be constructed from the engineering strain increment vector components:

$$[F]^\mathrm{T}[F] = [I] + 2[E] = [U] = [Q]^\mathrm{T}[D][Q]$$

where $[I]$ is identity matrix, $[E]$ - the strain increment matrix, $[Q]$ – an orthogonal matrix, and $[D]$ is a diagonal matrix with eigenvalues of $[U]$ matrix. The singular value decomposition can be easily calculated, because the $[U]$ matrix for membrane shell elements has the form:

$$[U] = \begin{bmatrix} 1 + 2\Delta\varepsilon_1 & \Delta\varepsilon_4 & 0 \\ \Delta\varepsilon_4 & 1 + 2\Delta\varepsilon_2 & 0 \\ 0 & 0 & 1 + 2\Delta\varepsilon_3 \end{bmatrix}$$

In this way, the deformation gradient could be found from singular value decomposition of $[U]$:

$$[F] = [Q]^\mathrm{T}\sqrt{[D]}\,[Q]$$

Now, the yarn directional unit vectors are rotated to their new positions and normalized:

$$\{q_f'\} = [F]\{q_f\}, \quad \{q_w'\} = [F]\{q_w\}$$

---

[*] Assistant Professor and Director, author to whom correspondence should be addressed
[†] Graduate Research Assistant

The new values of the angles determining the fiber directions are calculated from yarn directional unit vector components as follows:

$$\beta_f = \sin^{-1} q_{f3}, \quad \beta_w = \sin^{-1} q_{w3}, \quad \theta = \frac{\tan^{-1}(q_{f2}/q_{f1}) - \tan^{-1}(q_{w2}/q_{w1})}{2}$$

The above-calculated angles can be used at each incremental step to calculate the instantaneous effective stiffness matrix of the RVC.

NUMERICAL RESULTS AND CONCLUSION

The developed woven flexible fabric material model is programmed as a user defined material subroutine in the LSDYNA finite element software. The implementation in LSDYNA is for balanced woven fabrics. The user defined material model works with reduced and fully integrated membrane shell elements. Ballistic impact of a blunt projectile onto Kevlar® 129 piece of fabric was simulated. The projectile has mass 2.8 g and initial velocity 341 m/s. The projectile is a cylinder with diameter and height of 5.38 mm.

The results of the ballistic impact simulation can be seen in the following figures as sequential images of deflected fabric surface and its profile. The time history of the projectile displacement and the projectile velocity almost coincide with the experimental results. The history of the contact force between the projectile and the fabric is given and the fully absorbed kinetic energy history is also shown. A cylindrical airbag with two springs at the end is modeled. The inflation sequence, the end displacement due to contraction of the airbag, and the end force is shown in the following figures.
The model shows very good capability for simulating ballistic impact problems and is validated through an experimental test. The model is suitable for airbag simulations that involve significant fiber reorientation and scissoring effects.

REFERENCES

1. A. Tabiei and I. Inavov, "Computational Micro-mechanical Model for Flexible Woven Fabric for Finite Element Impact Simulation", *International Journal of Numerical Methods in Engineering*, **Accepted to appear** (2001).
2. A. Tabiei and Y. Jiang, "Woven fabric composite material model with material nonlinearity for nonlinear finite element simulation", *International Journal of Solids and Structures*, **36** (18), 2757-2771, (1999).
3. A. Tabiei, Y. Jiang, and Y. Witao, "Novel micromechanics-based woven fabric composite constitutive model with material nonlinear behavior", *AIAA Jounal*, **38** (8), 1437-1443, (2000).
4. R. Tanov and A. Tabiei, "Computationally Efficient Micromechanical Woven Fabric Composite Elastic Constitutive Models", *Journal of Applied Mechanic,* Vol 68, march 2001.
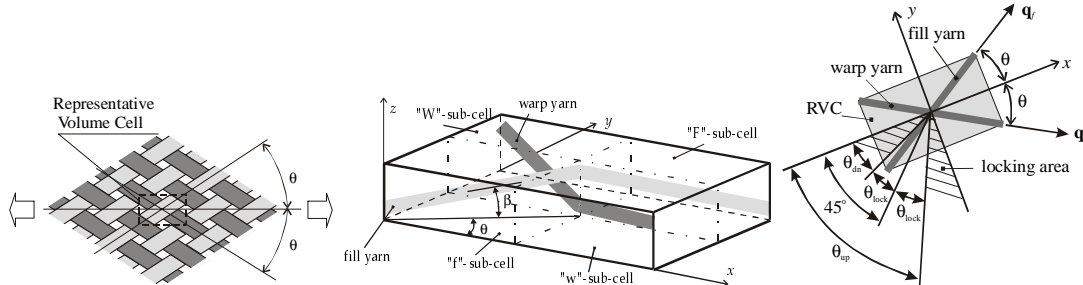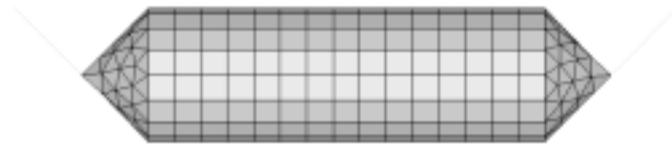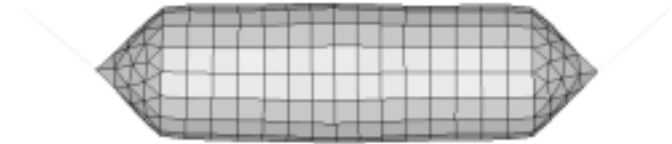
Blunt Body Impact Into Kevlar Fabric
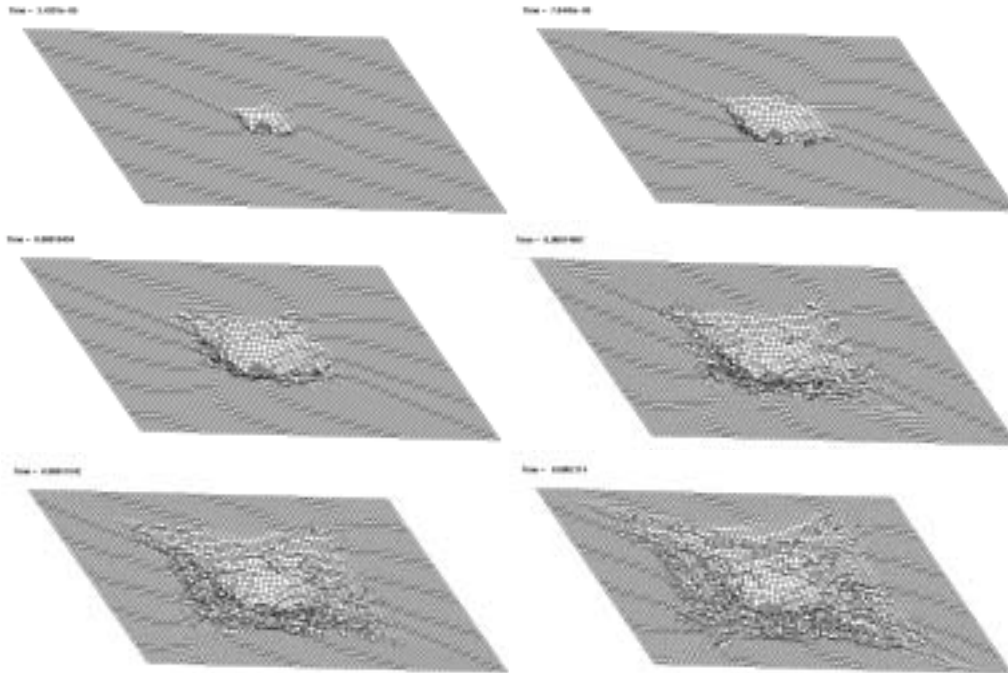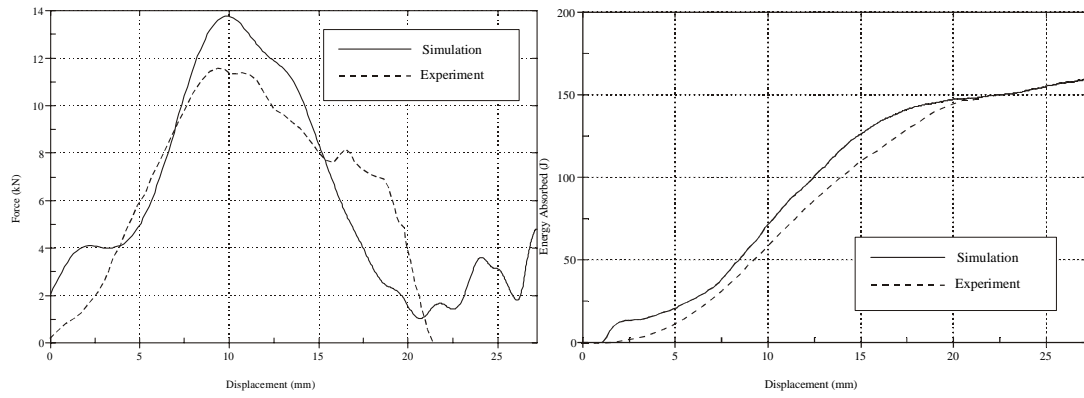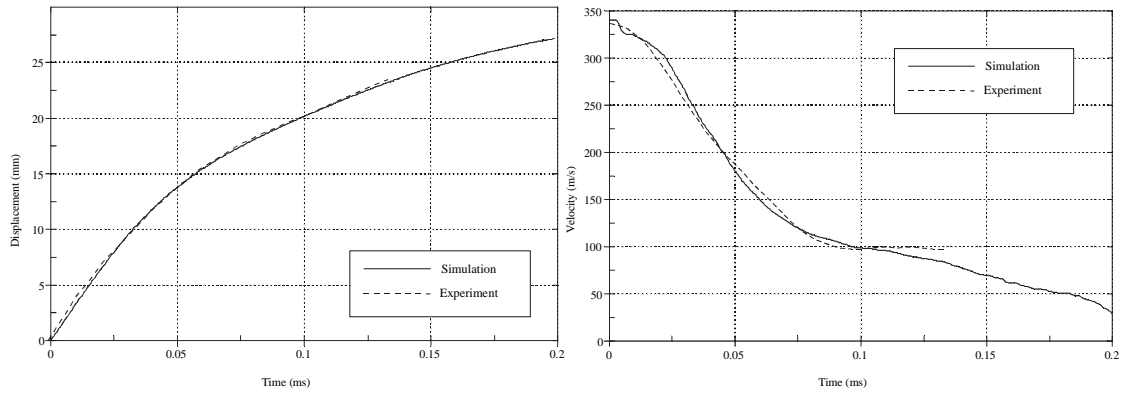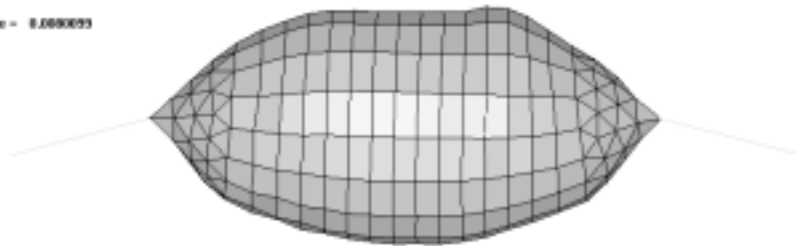
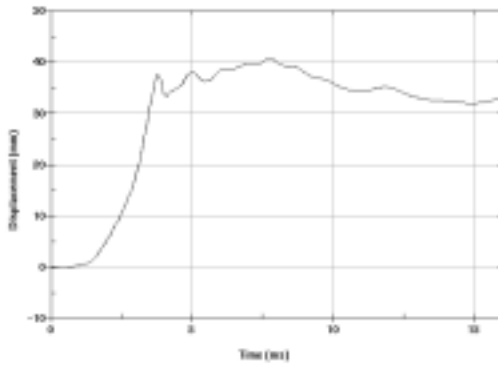Inflation simulation of small loose fabric airbag

Time = 0

Time = 0.0020000

Time = 0.0000099

Displacement and force of airbag ends in axial direction
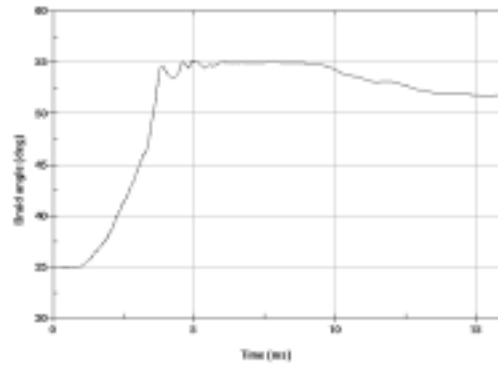
7

(Computer generated grapic by Dan Dowdy, courtesy of artist)

Confederate vessel H.L. Hunley is brought to the surface intact, thanks to analysis of hull structure.

More than 100 years before finite element analysis emerged, two New Orleans steam gauge manufacturing engineers used their engineering skills to build a Confederate submarine. As the H.L. Hunley set her mark in history on February 17, 1864, after sinking the Union warship U.S.S. Housatonic, she, too, sank, not to surface again for 136 years.

One of 2000's most dramatic developments for historians and archaeologists was the raising of the Hunley. It was also a significant success for Oceaneering International Inc., of Upper Marlboro, Maryland, USA, through its use of ANSYS/Mechanical.

Despite its pioneering in underwater warfare, the exact location of the Hunley had been virtually forgotten until its rediscovery in 1995. After sinking mysteriously in 1864 off the coast of Charleston, South Carolina, the vessel and her crew remained dormant in water less than 30 feet deep. Through the years, several efforts were made to locate the Confederate Navy's carefully guarded secret. Finally, in 1998, The Hunley Commission, a non profit organization dedicated to the recovery and preservation of the historic submarine, approached Oceaneering International about the feasibility of recovering the vessel.

Oceaneering International's prerecovery planning objective involved systematically evaluating different recovery methodologies and developing a concept of operations which included modeling the hull structure and analyzing the anticipated loads using ANSYS/Mechanical. The primary concern driving the engineers at Oceaneering International was that the Hunley's slender hull – 40 feet long, 4 feet wide, and 4 feet high – would come apart as it was lifted from the seabed and raised to the surface. Since the Hunley has tremendous historical significance, such disintegration would be disastrous.

Divers and marine archaeologists determined that many rivets in the hull had turned to powder. For example, the hull's copper rivets had essentially dissolved due to the decomposition of the iron hull plates by salt water. For the analysis, this meant that only sand and sediment held much of the hull plates together.

During the recovery, divers looped 32 slings around the wreck to support its 65,000 pounds and cushioned it with inflated foam pillows. The slings, attached to a steel frame, held the submarine as a crane lifted it to the surface, placing it on a barge to be towed back to shore.

Presently, the Hunley and her entombed ill-fated crew lie in a tub of water for preservation purposes at the Warren Lasch Conservation Center in the former Charleston Naval Shipyard. Researchers expect that it will take years to examine the vessel and its contents.

In the recovery of the Hunley, 32 slings attached to a steel frame supported the vessel's 65,000 pounds as a crane lifted the submarine to the surface.

Finite element model of the hull shows riveted connections in purple.
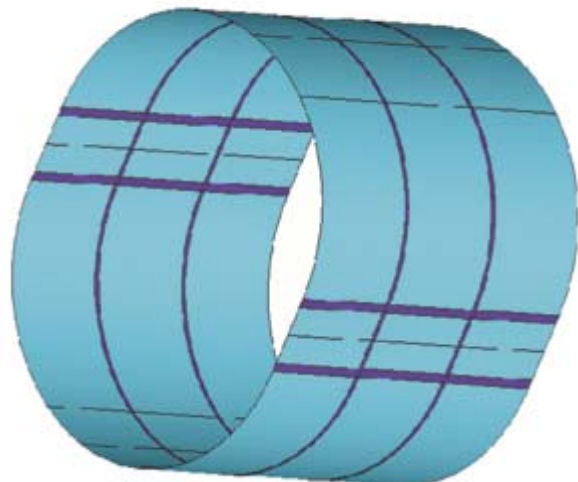
May 7, 2001
- **Showcased Product: Hewlett-Packard the HP Omni 500** laptop available under 3.5 pounds and one-inch thin.
- **Site**: Added Implicit FEA Note #7 to www.implicitfea.com
- **Site**: Added the web site CAE Information – www.caeinformation.com – This will be a links page – if you have a site you would like us to mention please send it to me for review.
- **Publication Title***: Use of LS-DYNA Shell Elements in The Analysis of Composite Plates with Unbalanced and Unsymmetric Layups*.

May 14, 2001
- **Showcased Software: ANSYS.FLOTRAN** release 5.7 – you can conduct free-surface simulations including surface tension effects.
- **Site:** Added Implicit FEA Note # 8 to www.implicitfea.com
- **Showcased Distributor: Engineering Research AB** (ERAB), the center of excellence in the field of Computational Structural Mechanics, CSM, and Simulation Based Design – headquartered in Sweden ERAB distributes LS-DYNA in Sweden, Denmark, Norway, Finland, Iceland, Estonia, Latvia and Lithuania.
- **Publication Title:** *Integration of Finite Element Analysis (LS-DYNA) with Rigid Body Dynamics (ATB) for Crash Simulation*.

May 21, 2001
- **Showcased Product: Server – Fujitsu PRIMERGY® ES200 & ES210:** the ideal solution when you're running a very tight budget and need to support services such as file and print, LAN mail server and retail or department back office functions.
- **Manual Announcement**: LS-DYNA Version 960 Keyword Manual available through LSTC – contact Claudia@lstc.com
- **Publication Title:** *Sidewall Indentation and Buckling of Deformed Aluminum Beverage Cans*

May 28, 2001
- **Showcased Participant:**

Livermore Software Technology Corporation's headquarters. LSTC's two new buildings, planned during the architectural phase, offer computer facilities and office networking. Powerful computer servers are centrally located with engineering workstations in every office. The classroom has 16 workstations for student training. During off hours, these 16 computers from a parallel processing node for computationally large problems

- **Publication Title**: *New Nonlinear Higher Order Shear Deformation Shell Element for Metal Forming and Crashworthiness Analysis. Part I. Formulation and Finite Element Equations*.

Thanks for your support: Marsha Victory, President, FEA Information Company

Events/Conferences

2001	
June 18-19 France	Third European LS-DYNA Conference will take place in Paris at the La Maison de la Chimie. 28, rue Saint Dominique, Paris, France For information email: dynalis@dynalis.fr
Aug 1-4 USA	Sixth US National Congress on Computational Mechanics, Dearborn, MI, USA. For information visit the site – USNCCM
Oct. 17-19 Germany	19th CAD-FEM Users' Meeting – International Congress on FEM Technology will be held October 17-19, 2001 at the DORINT SANSSOUCI Hotel in Potsdam, near Berlin.
May 19-21, 2002, USA	7th International LS-DYNA User's Conference at the Hyatt Regency Hotel & Conference Center – Fairlane Town Center, Dearborn, MI 48126

Classes/Seminars – July
Visit Participant's Web Site For Complete Details

2001	Country	Information	Class Title
10-11	USA	ANSYS Headquarters	Basic Structural nonlinearities
10-11	UK	Oasys	Automotive Crash Modeling
11-13	KR	Theme	LS-DYNA Introductory Training
11	KR	Kostech	Pre-post processing
11-13	USA	LSTC	ALE/Eularian & FSI in LS-DYNA
12-13	KR	Kostech	Introduction to LS-DYNA
16-20	USA	MSC	Introduction to MSC.Linux
17-18	USA	ETA	DYNAFORM – PC focused
17-19	USA	LSTC	Geomaterial Modeling with LS-DYNA
17-20	CN	MFAC	Sheet Metal Forming Using LS-DYNA
19-20	KR	Kostech	ALE/Eulerian & Fluid/Structure Interaction
23-25	USA	ANSYS Headquarters	Intro to ANSYS I
26-27	KR	Theme	Eta/FEMB Introductory Training
31-Aug3	USA	LSTC	Introductory LS-DYNA Training

FEA Information Participants

FEA Information Software, Hardware, O/S Participants

Livermore Software Technology (LSTC)	Livermore, CA, USA
Engineering Technology Associates, Inc (ETA)	Troy, Michigan, USA
Oasys, Ltd	United Kingdom
Japanese Research Institute	Tokyo, Japan
EASi Engineering	Madison Heights, MI, USA
ANSYS, INC.	Canonsburg, PA, USA
Hewlett-Packard (HP)	Cupertino, CA, USA
Silicon Grahics (SGI)	Mountain View, CA, USA
MSC.Linux	Costa Mesa, CA, USA
Fujitsu	Tokyo, Japan

FEA Information Educational Participants

Dr. Ted Belytschko	Northwestern University, Chicago, IL. USA
Dr. David Benson	University of California, San Diego, CA. USA
Dr. Bhavin V. Mehta	Ohio University, Ohio, USA
Dr. Taylan Altan	The Ohio State University, Ohio, USA
Professor Ala Tabiei	University of Cincinnati, Cincinnati, Ohio, USA

FEA Information Distributors and/or Consulting Participants:

Australia	Leading Engineering Analysis	Mexico	Livermore Software Technology Corp.
Canada	Metal Forming Analysis Corp.	Russia	Livermore Software Technology Corp.
Ireland	1. Livermore Software Technology Corp. 2. OAYSIS, LTD	Singapore Malaysia	Livermore Software Technology Corp.
Italy	Livermore Software Technology Corp.	Sweden	ERAB
Japan	Japanese Research Institute	Switzerland	Livermore Software Technology Corp.
Korea	Korean Simulation Technologies	S. America	Livermore Software Technology Corp.
Korea	THEME	Netherlands	Livermore Software Technology Corp
Germany	CADFEM	India	Gisseta
USA	1. Livermore Software Technology Corporation 2. Dynamax 3. ANSYS – ANSYS/LS-DYNA 4. Engineering Technology Associates (ETA)		