FEA Information International News

www.feainformation.com

Issue May, 2001

For FEA Information International News

FEA Information International News is a free e-mail publication mailed during the 3rd week of the month

A monthly synopsis of the additions/revisions to the FEA Information web sites.

When available information from our commercial participants, educational participants and parties interested in sharing technical information news.

Feel free to have your associates notify me to be added to our mailing list, Marsha – mv@feainformation.com

For information how to become a participate on the FEA Information Websites or to feature an article in the FEA Information News contact Marsha at mv@feainformation.com

FEATURES

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.

Dr. David Benson

A single structure may be broken into several substructures to permit different groups to analyze them and to enhance the overall efficiency of the structural analysis. The final structure is analyzed by assembling the stiffness matrices of the substructures into a single model of the structure. This approach is only valid if the substructures behave in a linear, elastic manner because the substructures are modeled by constant stiffness matrices. Substructures can be used in nonlinear analyses as long as the substructure itself behaves in a linear, elastic manner. The stiffness matrix for a substructure is commonly referred to as a "superelement". A superelement can be created, therefore, by simply writing out the stiffness matrix of a substructure. The efficiency of a calculation may be improved by reducing the number of degrees of freedom in the superelement before it is used.

There are many different techniques for reducing the degrees of freedom. All of them generate a transformation matrix, *H*, which expresses the original degrees of freedom of the substructure, d^o in terms of the reduced set for the superelement, d^s.

$$d^o = Hd^s$$

The superelement stiffness matrix, K^s, is calculated from the original stiffness matrix, K^o, according to

$$K^s = H^T K^o H.$$

Similar calculations give the expressions for the superelement mass and damping matrices,

$$M^s = H^T M^o H, \quad C^s = H^T C^o H,$$

and the superelement force vector, F^s, is calculated from the nodal force vector, F^o,

$$F^s = H^T F^o.$$

The number of equations in the superelement stiffness is equal to the number of degrees of freedom in the superelement, which can be a small fraction of the number degrees of freedom in the original structure. The accuracy of the solution will depend on how well the reduced number of degrees of freedom approximate the displacement of the substructure.

Perhaps the simplest method is static condensation, where the nodes are divided into "interior" and "attachment" nodes. The attachment nodes usually lie on the boundary, and are connected to the rest of the structure or have a load applied to them, while the interior nodes only interact with the other nodes belonging to the substructure. In a static problem, the displacements can be partitioned into $d^{a,}$ and d^i for the attachment and interior degrees of freedom, and the stiffness matrix and the applied loads can be partitioned in a similar manner,

$$\begin{bmatrix} K^{aa} & K^{ai} \\ K^{ia} & K^{ii} \end{bmatrix} \begin{Bmatrix} d^a \\ d^i \end{Bmatrix} = \begin{Bmatrix} f^a \\ 0 \end{Bmatrix},$$

noting that the interior, by definition, has no applied forces. Based on the partitioned matrix, the displacement of the interior is expressed in terms of the displacements of the attachment nodes,

$$d^i = -[K^{ii}]^{-1} K^{ia} d^a.$$

The transformation matrix is therefore

$$H = \begin{bmatrix} I \\ -[K^{ii}]^{-1} K^{ia} \end{bmatrix}.$$

Another approach to calculating the transformation matrix chooses a set of load patterns, P, which are representative of the distribution of loads on the substructure. The transformation matrix is calculated by solving the equation

$$K^{o} H = P.$$

The number of degrees of freedom in the superelement equals then number of load patterns, and the accuracy of the model is determined by how well the load patterns are chosen.

Component mode synthesis was developed by the aerospace industry to provide a method for reducing the number of degrees of freedom in a substructure without having to guess at load patterns. The interior displacements are modeled with a subset of the mode shapes from a special eigenvalue problem, namely one in which the attachment nodes are constrained to have zero displacement. For this procedure, the transformation matrix is written as

$$H = \begin{bmatrix} 0 & I \\ \Psi & -[K^{ii}]^{-1} K^{ia} \end{bmatrix},$$

where Ψ is the subset of the mode shapes. For this coordinate reduction, the displacement degrees of freedom are the amplitudes of the interior modes and the displacements of the attachment nodes.

LS-DYNA Version 960 has the ability to import mass, stiffness and damping matrices for superelements from NASTRAN and from the LS-DYNA EIGOUT file. Since the superelements can contain a mixture of different coordinate types, only the degrees of freedom associated with the attachment nodes can have either force or displacement boundary conditions applied to them. The attachment nodes are automatically determined by their specification as nodes within the LS-DYNA input file. LS-DYNA can recover the stress and strain within the substructure if the mesh associated with the substructure, its modes, and its elastic constants are specified. The recovered stress can be put out in any of the LS-DYNA output files and viewed with LS-POST.

Livermore Software Technology Corporation

LS-DYNA
CONSTRAINED_OFFSET AND BEAM_OFFSET

Because of an urgent request, we have updated the tied offset logic to use constraint equations or beam type springs. This update, though not in the User's Manual for version 960, has been added to versions 950f, 960, which is being ported for the production release, and the 970 version, which is under development. The penalty method for offsets is often unsatisfactory. If the added stiffness is too large then stability problems can occur or if, on the other hand, the added stiffness is too small then the offset condition is not accurately satisfied. The offset options applies to contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE. The following is the update to the User's Manual:

O*PTION4* specifies that offsets may be used with the tied contacts types. If one of the following three offset options is set, then offsets are permitted for the contact types, and, if not, the nodes are projected back to the contact surface during the initialization phase and a constraint formulation is used. Note that in a constraint formulation the nodal points of rigid bodies are not permitted in the definition.

OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. The OFFSET option switches the formulation from a constraint type formulation to one that is penalty based where the force and moment (if applicable) resultants are transferred discrete spring elements between the slave nodes and master segments. For the TIED_SHELL_EDGE_TO_SURFACE contact the BEAM _OFFSET option may be preferred. Rigid bodies can be used with this option.

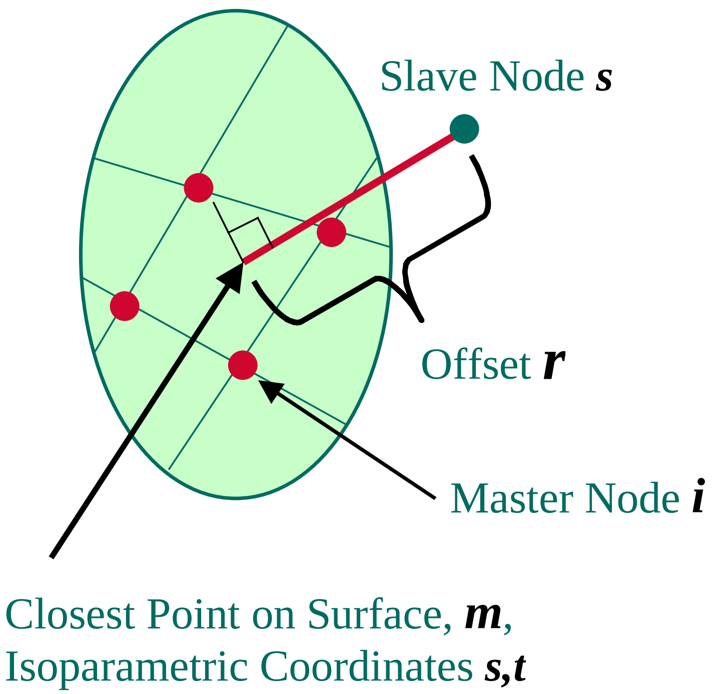
BEAM_OFFSET

This option applies only to contact type TIED_SHELL_EDGE_TO_SURFACE. If this option is set, then offsets are permitted for this contact type. The BEAM_OFFSET option switches the formulation from a constraint type formulation to one that is penalty based. Beam like springs are used to transfer force and moment resultants between the slave nodes and the master segments. Rigid bodies can be used with this option.

CONSTRAINED_OFFSET

Contact types TIED_NODES_TO_SURFACE, TIED_SHELL_EDGE_TO_SURFACE, and TIED_SURFACE_TO_SURFACE may be used with this option. If this option is set, then offsets are permitted for these contact types. The CONSTRAINED_OFFSET option is a constraint type formulation. The nodal points in the TIED_NODES_TO_SURFACE option and the TIED_ SURFACE_TO_SURFACE may not be connected to structural nodes, i.e., nodes with rotational degrees-of-freedom, since the rotational degrees-of-freedom are not affected, which will lead to an instability since the translational motions due to rotation are imposed on the slave nodes.

The CONSTRAINED_OFFSET option will be extended to implicit applications soon. The MPP version is being updated to include this important feature.
The formulation is a simple extension of the default tied constraint logic which accounts for the offset in the geometry. The equations are outlined in the box below.



$$F^m = F^s$$
$$F_i^m = F^m N_i(s,t) + F_i^m$$
$$M^m = r \times F^m$$
$$M_i^m = M^m N_i(s,t) + M_i^m$$
$$V^m = \sum V_i^m N_i(s,t)$$
$$\omega^m = \sum \omega_i^m N_i(s,t)$$
$$A^m = \sum A_i^m N_i(s,t)$$
$$\dot{\omega}^m = \sum \dot{\omega}_i^m N_i(s,t)$$
$$V^s = V^m + \omega^m \times r$$
$$A^s = A^m + \dot{\omega}^m \times r + \omega^m \times \left(\omega^m \times r\right)$$

NEW FEATURES: Negative Volume Error Handling

An improvement has been made to handle the situation when LS-DYNA encounters a brick element with negative volume. Previously, an error message was written and the simulation error terminated. Now, after writing the error message, control is passed to the automatic time step subroutine. If automatic time step control has been activated (*CONTROL_IMPLICIT_AUTO), the simulation will retreat and attempt to solve the step again using a smaller time step size. Otherwise, an error termination will result.

This is an important improvement for nonlinear implicit simulations. During the equilibrium iteration process, poor guesses at equilibrium geometry can routinely produce inverted elements. These poor guesses are often caused by a sharp nonlinearity in the model during the loading process, such as plastic yielding of the material, or a sudden change in contact conditions. During the iteration process, a trial equilibrium geometry is not accepted as a valid equilibrium solution until the convergence tests are satisfied. It is therefore a simple matter to abandon an equilibrium search when a negative volume element is detected, discard the bad trial geometry, and proceed to the automatic time step control subroutine to select a new time step size.

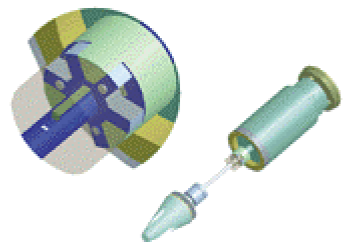
APPLICATIONS & BENCHMARKS – Springback Convergence Trouble

Springback in sheet metal stamping is an important application for implicit analysis. One procedure used for springback simulation is to use the keyword *INTERFACE_SPRINGBACK_DYNA3D to create an output file named DYNIN at the end of the forming simulation. This file contains deformed geometry and internal stress data for the work piece, and can be used to build a new input deck for a stand-alone implicit springback analysis.

An important feature in LS-DYNA is the ability to change the shell element thickness integration rule between stages in a multi-stage forming simulation. If the current thickness integration rule does not match the data contained in the DYNIN file, interpolation or extrapolation is used as necessary. An example where this feature could be effectively used is in gravity loading analysis, followed by stamping analysis. In the gravity loading phase the material behavior is elastic, so the simulation efficiency can be dramatically improved by using only two thickness integration points. Later, in the stamping analysis, five or more points are necessary to capture plastic stress distribution through the thickness of the sheet. LS-DYNA will correctly initialize all five integration points by extrapolating the two-point data written into the DYNIN file after elastic gravity loading.

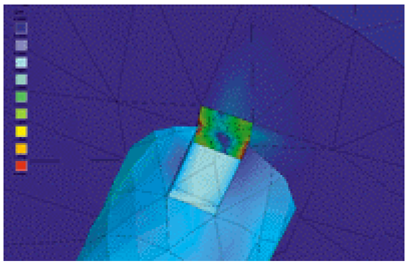
However, a dangerous error can be introduced using this feature! If the stress state in the DYNIN file is not elastic, interpolation and extrapolation cannot always be accurately employed. For example, if five integration points are used in a first-stage stamping simulation, and seven points are requested for a second-stage simulation, accurate initialization of the seven-point data is highly unlikely. Extrapolation and interpolation errors will produce misleading forming results, and cause convergence trouble and inaccuracy in springback simulations. For this reason it is essential to maintain the same thickness integration rule in multi-stage stamping simulations once plastic deformation has developed.

CASE Study: DesignSpace® Helps Visionary Graham Hawkes Prepare For Underwater 'Flight'
Reprinted with permission from the website of ANSYS, INC.



When it comes to exploring the ocean depths, Hawkes, Hobson, Konvalin, and Whiteaker (HHKW) Engineering Associates rely on DesignSpace software. HHKW is the main design team for Hawkes Ocean Technologies, the world's leading commercial manufacturer of single-person manned submersibles. "Deep Flight 1," one of the company's two submersible designs, has a depth rating of 1,000 meters and resembles an underwater jet plane. The design consists of a fully enclosed pressure hull that is as small and fast as possible. While requiring only a small research vessel for launch and retrieval, Deep Flight 1 can be quickly deployed for prompt and efficient deep-sea research and reconnaissance.

The design and development of Deep Flight 1 was a challenge for Graham Hawkes, chief designer, and Eric Hobson, senior mechanical engineer. HHKW Engineering Associates uses Autodesk Mechanical Desktop software for its mechanical design. The company began working with DesignSpace after Hobson received a 30-day evaluation copy of DesignSpace software and decided to put it to the test on Deep Flight 1 solid models. Prior to trying a mechanical simulation tool like DesignSpace, Hobson relied on basic hand calculations. "There are only a few critical components on Deep Flight 1 that really concern us. Obviously, the most critical is the pressure hull cockpit for the pilot. We also are concerned with some of the electronic housings which must maintain pressure at one atmosphere. In the past, if we were in doubt, we would add material to questionable areas of the structure," Hobson said.

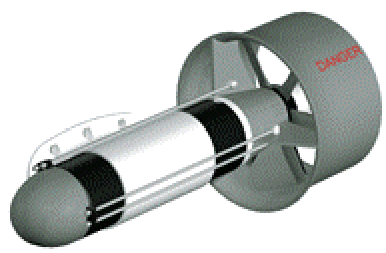


After installing DesignSpace, the HHKW design team was able to get up to speed very quickly with the software. "With DesignSpace, it was easy to analyze the Deep Flight 1 models. Within an hour, I was getting really good results from my first attempt with the software," said Hobson.

The pressure hull of Deep Flight 1 is handmade from composite materials that are slowly wrapped around a band roll. The process takes several days of wrapping and finishing to produce a single hull. The pressure hull is then extensively tested in a pressure tank prior to any human applications. If the pressure hull prototype fails, then the design team must go back to the drawing board. The pressure hull was one of the first designs that was tested using DesignSpace software. "We're seeing an incredible opportunity to cut back on our lengthy development process," stated Hobson.

"Graham Hawkes has developed some conceptual ideas for the pressure hull in his head for the last few years. We could never get any meaningful results from hand calculations to make one design stand out from another. Using DesignSpace, we quickly drew up the conceptual designs and ran comparative analyses. It quickly became obvious which designs were good and which were not. Essentially, DesignSpace enabled us to take the design concepts we've played with for years and, in only a couple of hours, decide which one to go with," said Hobson.

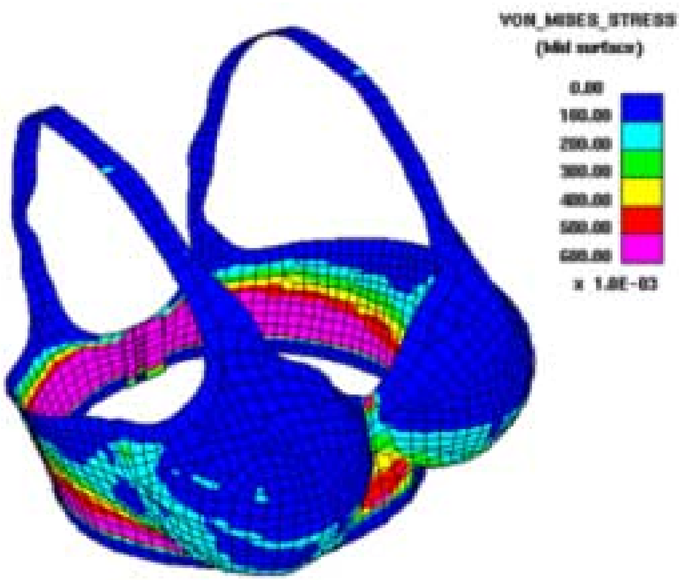
Through DesignSpace testing, the design team quickly discovered that the pressure hull design required changes. Those corrections were made easily on their Mechanical Desktop model. Since DesignSpace associates with the Mechanical Desktop geometry file, they updated their engineering simulations automatically and re-ran the analyses. In addition, DesignSpace Report provided them with a valuable engineering documentation feature, which helped Hobson compare his different pressure hull design scenarios. Hobson explained, "As opposed to our conventional development process, DesignSpace saves us a ton of time. It enables us to do things we just couldn't do before."



Making a good situation even better, the HHKW design team has recently started using DesignSpace v5 and can now perform multiple-component assembly analyses. "Version 5 provides huge benefits. In testing the pressure hull, which is a multiple-component assembly, we just bring in the entire assembly, throw the depth pressure on all surfaces, and know that the problem was solved correctly for the individual parts," said Hobson. "It's a real time saver."

Bra Analysis
Oasys (Ove Arup SYStems)
Excerpt reprinted from the website of Arup

A Project by Ove Arup & Partners' Advanced Technology Group, using dynamic non-linear finite element techniques to analyze the bras structural performance, in assisting Powell and Seymour (London based industrial designers) in assessing the performance of the existing bra design.



Until recently there has been little application of engineering to bra design despite an early patent brought by an aeronautical engineer. In fact, the basic design has not changed dramatically since its appearance in 1885. In the past year there has been an emergence of interest in designing the perfect bra. Most recently it became the subject of a program broadcast on UK TV Channel 4 on the 24 June 1998. 'Designs on Your Bra' followed London based industrial designers, Dick Powell and Richard Seymour, as they attempted a radical redesign of the bra. They were assisted during the project by the consulting engineers, Ove Arup & Partners' Advanced Technology Group, who were able to give an insight into the engineering performance of the bra using advanced computer techniques.

Modern bras, which are mostly designed from a fashion perspective to look flattering, are unable to provide the necessary support. Comfort is also an issue.

A study by Nottingham Trent University showed that 70% of women were wearing a bra that doesn't fit properly. Women cannot find sizes to fit them or the bra size alters with wear and washing. The problem for manufacturers is then to produce a bra with the necessary support and fit, with no loss of performance over time, which is attractive in design and capable of all of this in a wide range of sizes.

The role of Arup's Advanced Technology Group in assisting Powell and Seymour in this difficult task was to assess the performance of the existing bra design. Louise Waddingham, who undertook this unusual analysis task noted, "The group is used to advising a wide range of designers, engineers and manufacturers and specializes in using advanced engineering methods in the vehicle, nuclear and seismic engineering industries." In this particular case the Arup engineers used dynamic non-linear finite element techniques to analyze the bras structural performance. A computational representation of the bra on a body was created, by scanning the geometry of the model, Loen, who was featured in the program. The bra was then constructed using techniques usually associated with the modeling of airbags and seatbelts in cars. This enabled the non-linearity of the bra material, contact interaction with the body and large displacements to be represented. The performance of the bra was analyzed by applying vertical accelerations, to simulate a person lightly jogging, or briskly walking. The analysis displayed fluctuating stresses in the bra cups and straps, varying with the walking pattern. Additionally, it displayed a higher constant stress around the base band where the bra was pulled tightly onto the body. More interestingly, the analysis revealed the behavior of the underwiring.

As manufactured, the underwiring is a two dimensional form. When being worn the wire undergoes bending in two directions. It is bent around the body as the bra is put on, and also bends in the other direction as it supports the weight of the breast. In addition, the axial forces along the length of the wire are significant in its tendency to pop out of its casing after repeated wear. Seymour & Powell focused on the inefficiency of the underwiring and in their bra design proposal they replace the wire with 'Bioform', a large plastic molded support that extends to the underarm area. In their TV program they claim the Bioform bra fits all the requirements of support, comfort and washability as well as adapting across three sizes at once. In fact, their ideas were received well by Charnos, a leading bra manufacturer, and the production of a commercial Bioform bra is under consideration.

FEA Information Web Sites For April - April News is archived on the News Page

April 2nd, 2001

- **Showcase Distributor: (LEAP)** Leading Engineering Analysis Providers located in Australia.
- **Site**: Added #26-30 FAQ on Sheet Metal Stamping by Xinhai Zhu on www.ls-dyna.com

April 9th, 2001

- **Showcase Software**: **JSTAMP-Works,** a sheet metal forming simulation system developed by Japanese Research Institute (JRI) located in Tokyo, Japan
- **Site**: Added an archive of these newsletters to our publication site www.feapublications.com
- **Site**: Added Implicit Notes 3 & 4 to our site www.implicitfea.com directed by Dr. Bradley Maker.

April 16th, 2001

- **Site**: Added a job opportunity – Three tenure-track assistant or associate professor positions in the Dept. of Comp. Science – St. Cloud University to our educational forum on the www.feainformation.com site
- **Site:** Added Implicit Notes 5 to our site www.implicitfea.com directed by Dr. Bradley Maker

April 23rd, 2001

- **Site**: Added Page 2. Mathematically describing the implicit solution procedure to the www.implicitfea.com site directed by Dr. Bradley Maker
- **Showcase Software**: **eta/VPG** a streamlined CAE software that provides an event-based simulation solution of nonlinear, dynamic problems developed by Engineering Technology Associates located in Troy, MI, USA .

April 30th, 2001

- **Site**: Added Note #6 New Features Explained in Implicit in LS-DYNA to the www.implicitfea.com site directed by Dr. Bradley Maker
- **Showcase Participant**: **Oasys Ltd**. (**O**ve **A**rup **SYS**tems) is the software house of ARUP and the LS-DYNA distributor located in the United Kingdom and Ireland. Oasys markets its own peripheral software that is fully compatible with LS-DYNA.
- **Showcase Software**: **EASi-SEAL®** a productivity environment that enables concept design of door, hood, and decklid seal systems developed by EASI Engineering located in Madison Heights, MI, USA

April Publications: (if you need a copy e-mail vic@lstc.com)

- *Simulation of Structural Latches in an Automotive Seat System Using LS-DYNA* –Tuhin Halder (Lear Corporation)
- *Finite Element Modeling of Co-Mingled Glass/Thermoplastic Fabrics for Low-Cost/High-Volume Composites Manufacturing* – Patricia P. Buso, James A. Sherwood, Julie Chen (U. of Massachusetts-Lowell)
- *Micromechanics Based Composite Material Model for Impact and Crashworthiness Explicit Finite Element Simulation* – Ala Tabiei and Quing Chen (U. of Cincinnati)
- *High Efficient and Powerful Integrated Design Support System "DYNA-Works"* – Fuminori Oshita and Osamu Kunieda (Japan Research Institute)

Thanks for your support and for sending information to share on the web sites.

Sincerely,
Marsha Victory
President, FEA Information Company

Courses and Events will be limited to 1 page
For further information contact event/course sponsor

Events/Conferences

2001	
May 6-11	Precision Metal Forming Association Tradeshow in Cleveland, OH, USA. Please visit our participant (ETA) Engineering Technology Associates exhibiting DYNAFORM in Booth 612.
June 5-7, 8 Poland	Third International Conference on Thin-Walled Structures (Cracow, Poland) Workshop on FEM Applications to the Analysis of Thin-Walled Structures
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
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 -April - May

2001	Country	Information	Class Title
May 1	USA	www.eta.com	DYNAFORM - This class will be PC focused.
May 7	Korea	www.kostech.co.kr	Pre-Post Processing
May 8	USA	www.ansys.com Headquarters	Basic Structural Nonlinearities
May 9	Korea	www.theme.co.kr	Introduction to LS-DYNA
May 17	Korea	www.kostech.co.kr	Sheet Metal Forming
May 14	USA	profdev@sae.org	Fundamentals of Finite Element Linear Analysis in Solid/Structural Mechanics
May23	USA	www.lstc.com	Introductory LS-OPT
May 23	UK	www.arup.com	Oasys Primer V. 8.0a
May 24	UK	www.arup.com	Oasys D3Plot & T/HIS V. 8.0a
May 24	Korea	www.theme.co.kr	Introduction to eta/FEMB
May 30	USA	www.ansys.com Headquarters	Electromagnetic Analysis

FEA Information Participants

FEA Information Software, Hardware, O/S Participants

FEA Information Participants	Headquartered
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

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

Distributors and Consulting Company Participants:

For LS-DYNA product information if not listed contact mv@feainformation.com

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		
USA	1. Livermore Software Technology Corporation 2. Dynamax 3. ANSYS – ANSYS/LS-DYNA 4. Engineering Technology Associates (ETA)		

Software and/or Product Announcement

LS-DYNA Limited Version for Students and Professors: Single Processor PC w/FEMB limited to a maximum of 10,000 elements for your home or university PC.
For details contact Marsha**: mv@feainformation.com**