March 2003

LEAP -Australia	**ALTAIR - Western Region US**	**DYNAMAX - US**	**Cril Technology Simulation - France**
GISSETA - India	**MFAC -Canada**	**DYNA*more* - Germany**	**FLOTREND - Taiwan**
KOSTECH - Korea	**ERAB - Sweden**	**THEME - Korea**	**ALTAIR - Italy**
NUMERICA - Italy			
CAD-FEM -Germany	**Numerica SRL - Italy**	**Dr. Ted Belytschko - US**	**Prof Gennaro Monacelli - Italy**
Dr. Bhavin V. Mehta - US	**Dr. Taylan Altan - US**	**Dr. David Benson - US**	**Dr. Alexey I. Borovkov - Russia**
Prof. Ala Tabiei - US			

FEA Information Inc. Worldwide News

March - Volume 4

Issue 03-2003

Editor	**Trent Eggleston**
Editor Technical Content	**Arthur B. Shapiro**
Technical Writer	**David Benson**
Technical Writer	**Uli Franz**
Graphic Designer	**Wayne L. Mindle**
Feature Director	**Marsha Victory**

Announcement: LS-OPT, the premier optimization package from LSTC, will be included as part of the release of LS-DYNA Version 970. LS-OPT is available for the Unix/Linux operating systems.

Arthur B. Shapiro of LLNL will join Livermore Software Technology Corp. as of April, 2003. Additionally he will continue as FEA Information Inc.'s Technical Content Editor

LS-OPT Version 2 Copyright © 2003 by LSTC

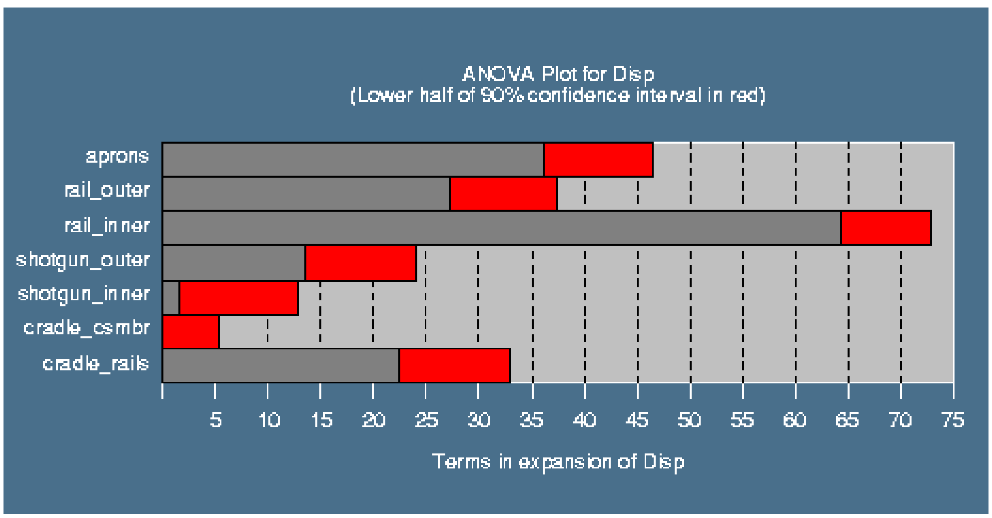
LS-OPT will be part of the 970 release

Free of charge for all LS-DYNA users

Accompanying the release of LS-DYNA Version 970 is the Version 2 release of LS-OPT, the simulation-based optimization program dedicated to LS-DYNA. LS-OPT features a few well-chosen, effective optimization technologies for design automation and has evolved into a very powerful tool by adding new capabilities and improving efficiency through features such as variable screening and simulation job distribution. The new hypertext User's Manual has been completely reworked and, in addition to the theory, features a number of new, and more complex examples. All the new features have been elegantly incorporated in the graphical interface: LS-OPT*ui*.

The following features are new in Version 2:

1. **Variable screening.** This tool allows the user to determine the importance of design parameters for the optimization process. The variable screening in LS-OPT is based on the Analysis of Variance (ANOVA) and the GUI depicts both the importance and its margin of confidence (in red below) of each variable. The variable screening output is an automatic byproduct of a design cycle or iteration in the process.



2. **Multidisciplinary Design Optimization (MDO).** In MDO the user can structure the design process into disciplines. Each discipline has its own variables and responses. These variables may overlap the variables of other disciplines. This approach is useful when optimizing multi-case (e.g. multiple crash impact cases) problems or multi-disciplinary problems (e.g. crashworthiness using explicit dynamics and ride comfort using eigenvalue analysis combined in the same design). Although certain LS-OPT entities, such as responses, are tied to the disciplines, composite functions are available to compute any global quantity.

3. **Mathematical expression library**. This library includes all the standard FORTRAN intrinsic functions as well as special functions that can be used to operate on response histories such as integrals, derivatives and minima/maxima. 'Lookup' functions allow the user to look up the time of occurrence of certain events such as the achievement of a specified value, a minimum or a

maximum. Integrals may involve variables other than time, e.g. energy can be computed by integrating force over deformation. Expressions can be specified in the LS-OPT user interface or solver/preprocessor template file.

4. **Distribution of simulation runs.** In addition to concurrent processing available in the earlier version, the LS-DYNA simulation runs can now be distributed across a network using a third-party queuing system. Input files are automatically transferred across the network and after the simulation, data extraction is conducted on the remote node and the data transferred to the local directory. This happens at ftp speed.

5. **Mode tracking.** For the purpose of using modal frequencies in noise and vibration design, a mode-tracking feature has been implemented to identify the mode of interest independent of the design. This feature avoids the problem associated with mode-switching when a design is changed. The mode-tracking feature interfaces with the eigenvalue analysis of LS-DYNA.

6. **New types of response surfaces.** Neural nets are introduced to provide global approximation, local updating of response surfaces, and trade-off studies using the entire design space. Traditional polynomial response surface approximations, especially linear approximations, are suitable only for local approximations and are difficult or impossible to update. In addition, the user is presented with a choice of order, e.g. linear, quadratic, etc.

7. **New types of experimental designs.** A new 'space filling' experimental design, in which the distance between any design pair is maximized, accompanies the neural nets approach.

Version 2.1 (to be released in April/May 2003) includes the following additional features:

8. **LS-DYNA binary database (binout).** The user now has the choice of extracting data from the ASCII files or the new binout database.

9. **Stochastic simulation.** Stochastic effects can be investigated using Monte Carlo simulation. The mean values of responses, coefficients of variation, extreme values, and reliability criteria such as the probability of exceeding a constraint bound value will be computed**.** The costs of the reliability computations can be reduced by considering substitute models such as response surfaces or neural networks instead of using the FEA results directly. The following statistical distributions are available: normal, uniform, lognormal, user-defined, and Weibull. Latin Hypercube Sampling is available to reduce costs.

10. **Stochastic optimization.** A sequential random search method based on Latin Hypercube Sampling, and using the standard constrained minimization formulation, has been implemented. By using domain reduction, the design sequence has been automated so that the method converges to the region of the optimal design. These design methods, being dependent on chance, are generally not as efficient as response surface methods, but may be useful as a means of finding a good starting design for the response surface method.

Newly Developed Capabilities of FEMB/PC Version 28

H. Ouyang, G. Xi and A. Esteves
Engineering Technology Associates, Inc.

Abstract

Finite element pre and post processor technology is continuing to develop at an accelerated pace, meeting the needs of users who are constructing ever more complex models which are being used to simulate an increasing variety of structural behaviors.

As a continuous effort, FEMB/PC 28 has been developed to meet increased user demands, bundling with LS-970 for synchronous distribution. This paper will discuss three key enhancements which give FEMB users the ability to easily construct more complex models and make use of the latest capabilities within LS-DYNA. These latest features include Topology AutoMeshing, Automated Tetra Meshing and a complete NASTRAN interface embedded within FEMB.

Introduction

FEMB/PC was originally developed to meet the needs of the emerging PC simulation market, and was bundled as a pre & post-processor with LS-DYNA/PC in 1996. Since that time, we have seen rapid development of PC capability in the areas of processing speed and graphics, making them a viable choice for simulations that were once targeted as supercomputing applications. This evolution of the engineering environment has required that continuous development of the pre & post processor be implemented to meet the needs of this constantly growing user community.

The original specifications of the FEMB PC software were such that all LS-DYNA capabilities would be accessible through the user interface. This specification drove the architecture of the FEMB software to be developed using a “template” system. This allowed users to access all of the LS-DYNA Keyword cards which were available in the User’s Manual. The benefit of the template approach is the ability to keep pace with new capabilities frequently added to LS-DYNA.

The complete interface also is a critical infrastructure that enable user to handle all processing tasks without text editing. This is an important feature since Windows-based users are not as familiar with the UNIX text editing tools such as VI, and less inclined to use such tools.

With this requirement of a “no text editing environment” underlying the GUI capabilities, all FEMB/PC development has provided a GUI which uses mouse clicks and smart input fields to guide the LS-DYNA PC user. In order to achieve this objective, the FEMB template system must be robust and complete, addressing all functionality available in LS-DYNA PC.

Meshing capability, of course, has been a primary requirement of the FEMB software. The original specification of FEMB made use of existing automeshing technology, and was adequate for the 1996 level of modeling sophistication. However, as simulation and computing capacity increased, the need for a more robust and accurate automeshing capability was evident.

To meet the needs of users for a more “hands-off” meshing tool, FEMB has implemented a meshing technique known as Topology Automeshing. This surface meshing approach yields meshes which require less repair of the mesh, and allows users to minimize manipulation of the surface data prior to meshing.

As element technology has been improved and implemented within LS-DYNA, the pre/post processing environment has had to keep abreast of these improvements, and enable users to access the latest solver enhancements. This has resulted in two key areas of development within FEMB; an automeshing routine for TETRA elements and the interoperability of the LS-DYNA and NASTRAN input files.

The following sections describe these major enhancements, and provide examples of how they may be implemented by users to improve their productivity.

Topology Automeshing

The single most critical task for a pre-processor is to generate quality mesh, efficiently. The existing FEMB meshing tools include the point/line/surface operations, node/element operations and manual meshing tools.

Within the existing automatic meshing capabilities, the Quad Dominant Mesher, Paving Mesher and Triangular Element Mesher are commonly used for plate and shell element applications. These have met the needs of users to create component models using limited CAD data.

As users needs have evolved, so has the need for extended capability in the area of automeshing technology. Users have additional pressures to complete the laborious modeling tasks faster, with less manual intervention. In addition, the typical model size has expanded significantly to fill the available memory and graphics capabilities.

To meet these evolving needs, FEMB has implemented a “Topology Automesher”. The technique makes use of the available CAD surface data and delivers users an improved mesh quality, with very little manipulation of the CAD data.

The Topology Automesher uses the topological information of the part surfaces to find and merge gaps/overlaps between adjacent surfaces. This technology allows the surface data to be repaired internally while generating the mesh. This reduces or eliminated the need to merge or stitch surface gaps and overlaps prior to the meshing operation. The meshing algorithm then uses these merged topological structures to fit the best possible mesh meeting the users specifications for the mesh size and various quality parameters.

The Topology Automesher has been shown to be a significant improvement over current meshing technology in terms of quality and speed.

Another mesh quality parameter that has been elusive and rather vague, is often described as an “eye-pleasing mesh”. This means that the mesh is quite regular and square where possible. The easiest way to describe this parameter, is that most engineers can look at a mesh pattern and easily identify that an automesher was used. The Topology Automesher greatly improves the quality of mesh with respect to this “eye-pleasing” quality, delivering a mesh that is more useful for engineers performing impact analyses.

Triangular elements in the mesh are greatly reduced through use of the Topology Automesher. Typical automotive applications have seen reductions from 15% using the Paving Mesh algorithm to 5-6% for the same components using the Topology Automeshing algorithm.

Future enhancements to this capability will be functions to optimize the mesh according to predefined mesh quality parameters. These functions (Auto-Repair and Enhanced Mesh) will allow users to automatically merge coincident nodes, repair boundaries and improve mesh parameters such as warpage, interior angles or minimum element size.

Topology Automesher Technology and Example

The advantage of the Topology Automesher is particularly noticeable for a full Body-in-White (BIW) modeling task.

The selected example is a full BIW design with the CAD surface produced from CATIA Version 4. The BIW CAD data shown in Figure 1is a typical vehicle structure model. There are over 100 unique components in this structure, much of which are shown in the Figure.

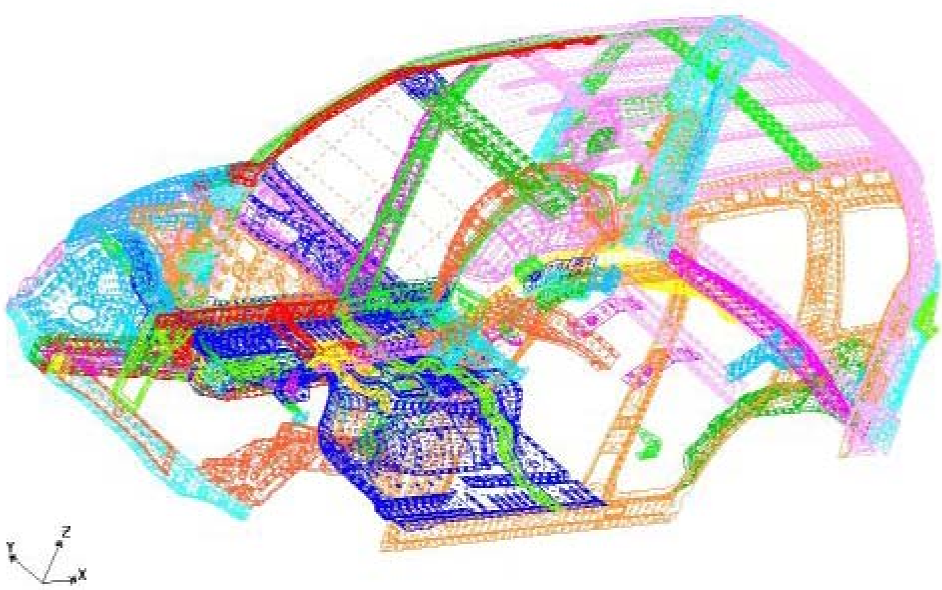


Figure 1. Full BIW CAD Surfaces (102 parts, 20958 Surfaces)

The CAD surface data is imported into FEMB via an IGES file. No prior manipulation of the surface data is required for use within the Topology Automesher.

The meshing parameters defined in the Topology Automeshing menu allow the user to define parameters which are suitable for their specific modeling task. In this case we have defined parameters consistent with those required for automotive body panels. Since the result of this mesher is a quad-dominated uniform size mesh, the user should check the desired element size, avoiding small elements which will impact the analysis time step.

Once the controlling parameters are set, the user has the option to automesh the full BIW automatically part-by-part in one command or individually one at a time.

The meshing module checks for duplicated surfaces and the integrity of the topology, allowing the user to exit and perform surface operations before finalizing the mesh. This prevents meshes from being created on corrupt or erroneous surface data.

The auto meshing process of the whole BIW will take approximately 20 minutes on a Pentium 4, 2 Ghz. PC.

Using the CAD data of the BIW shown in Figure 1 and a mesh size of 8mm, the model will have 347,401 elements and 355,499 nodes. The resulting mesh contains only 5.6% triangular elements, which is seen as a significant improvement over previous automeshing algorithms.

The mesh of the body side panel meets applicable quality criteria and also captures all geometry details with a "eye pleasing" mesh pattern, and requires a minimum manual editing and repair of elements.

As a comparison of the mesh improvements attained with the Topology Automesher, the same body side panel was meshed with the Paving Mesh option and the Topology Automeshing option. These meshes are shown in Figures 3 and 4. As can be seen from this comparison, the mesh patterns generated from the Topology Automeshing contain fewer triangular elements and have a very regular pattern.

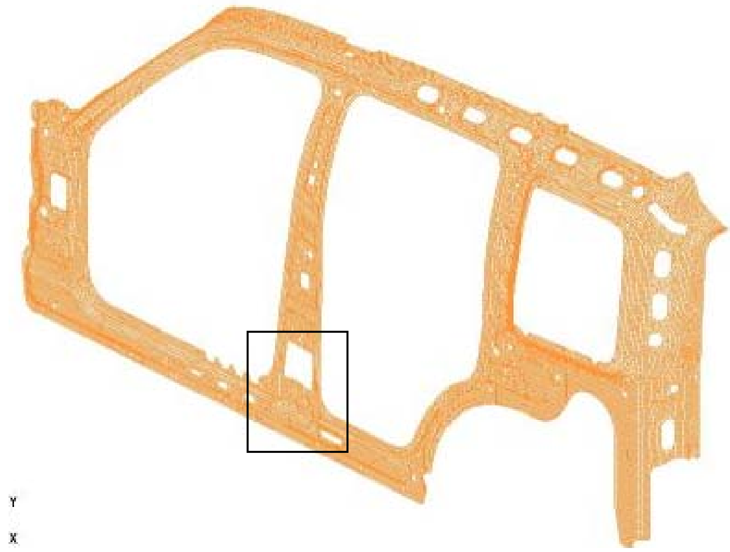


Figure 2. Body Side Topology Automesh result

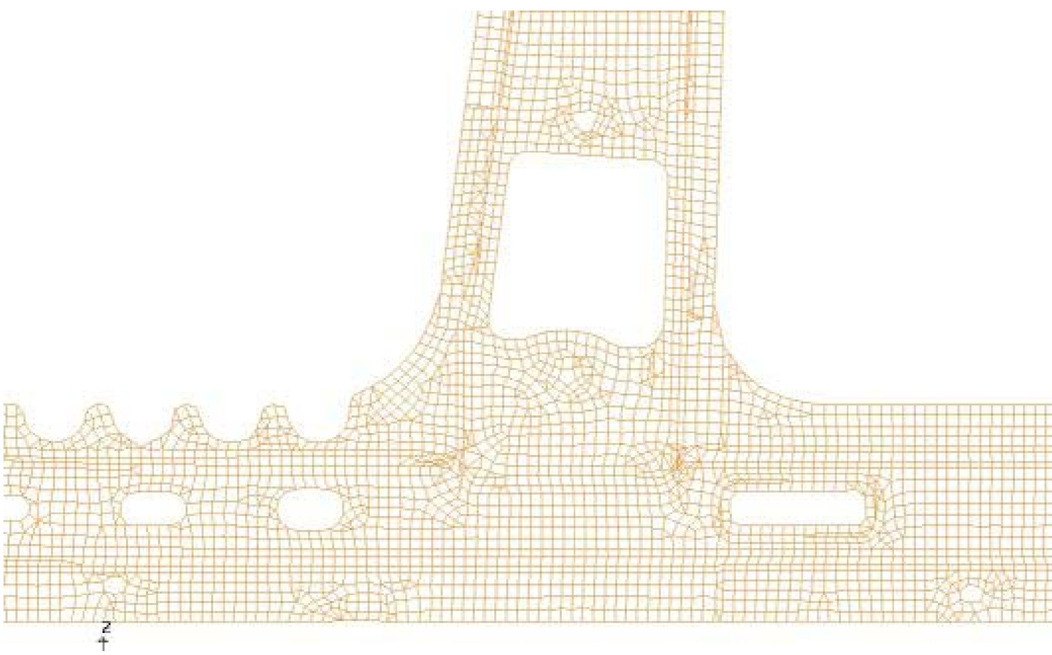


Figure 3 : Detail View of Paving Mesh Results of B-Pillar area

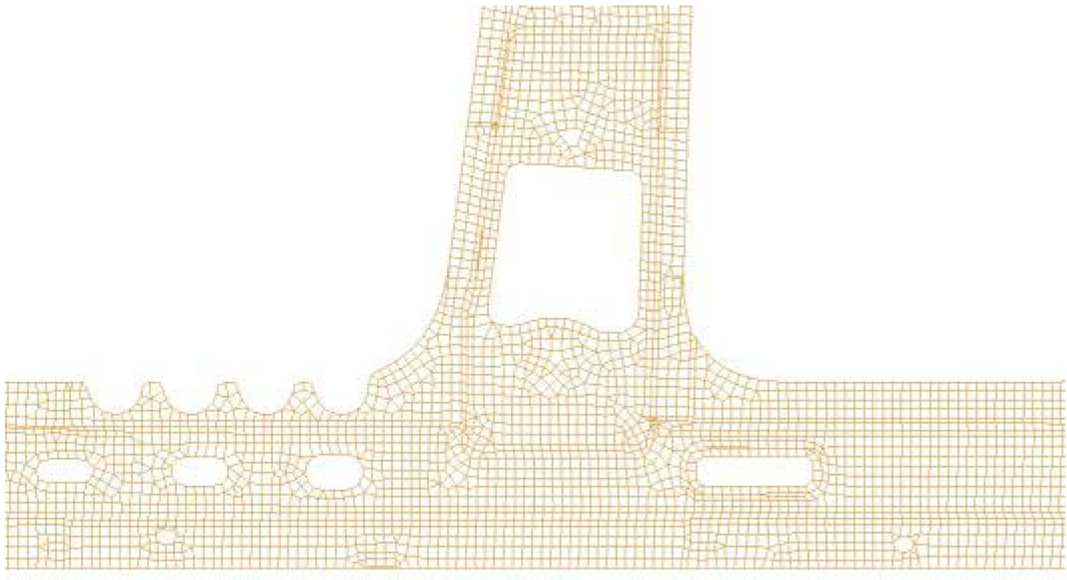


Figure 4: Detail View of Topology Automesher Results of B-Pillar area

Tetra Automeshing for Solids

LSTC's continuous effort of improving the quality and robustness of the Tetra Element, has resulted in a renewed interest in the use of this type of element. To enable use of this style of element, FEMB has implemented a Tetra Automesher within version 28. This is a new capability within FEMB.

The Tetra Automesher requires a "water tight" skin mesh that covers the entire surface of the solid object. The Topology Automesher algorithm provides a useful tool to generate the high quality outer skin mesh necessary to automatically generate tetra elements. Use of this process requires virtually no manual work and in a time span of minutes, a tetra mesh model is generated for a complicated solid object.

Implementation of this capability allows users to create solid models which have the ability to accurately simulate solid behaviors using the Tetra element. Further development of this capability will be aimed at ultimately delivering a complete solid meshing ability.

Tetra Automeshing Process and Example

The process required to create a Tetra mesh using the Tetra Automesher makes use of many automated features within FEMB. The following example outlines the process using a die structure as a typical application.

Step 1. Import CAD Surface of the solid object in either IGES or VDA format. It is recommended to conduct a visual check of the integrity of the surfaces, such as shading the surfaces to identify any bad surfaces. Grossly deficient CAD surfaces should be repaired prior to meshing using the FEMB point/line/surface tools.

Step 2. Using the Topology Automesher, select an appropriate average element size, a set of parameters to define the quality criteria. The user will select the specific surfaces to be meshed, or the entire structure, if desired.

The Quad-Dominated skin mesh will be generated. This skin mesh may be quickly divided into an all-triangular skin mesh.

Step 3. Check Free Boundary of the triangular skin mesh and conduct mesh repair tasks to ensure a water-tight or crack-free skin surface. Obviously, a uniform mesh pattern is desirable as well as discontinuities in mesh size and the avoidance of small elements.

Step 4. Activate the Tetra Mesh Generator and the mesh generation will be completed using the surface (skin) mesh as a boundary.

Step 5. Check for defects by dividing the object into various part for a detailed examination.

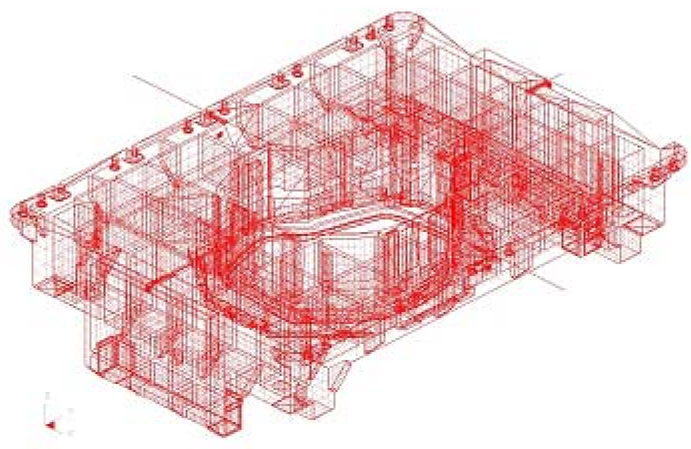


Figure 5. Die Structure IGES Surfaces

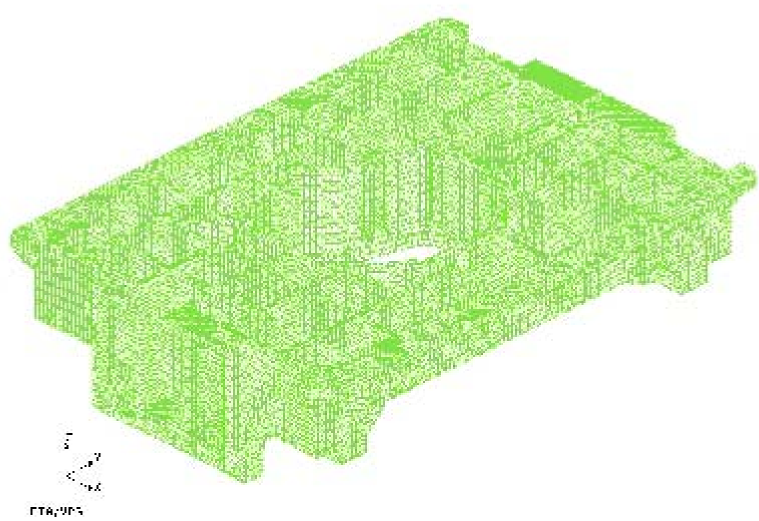


Figure 6. Die Structure Skin Mesh

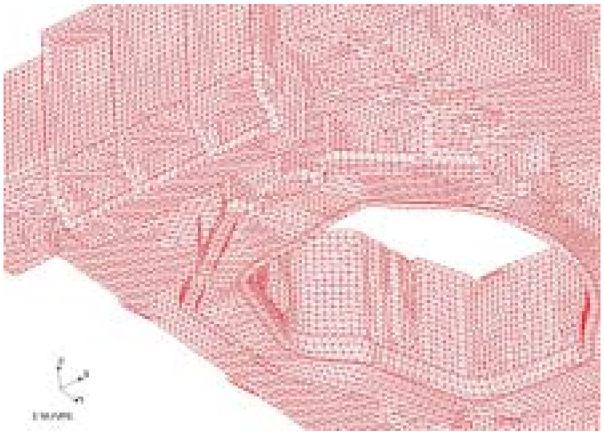


Figure 7. Tetra Mesh Results - Detail View

NASTRAN Interface Development

Most users are aware that LS-DYNA supports NASTRAN input file formats by defining the first Key Word Card *NASTRAN. This capability allows users to make use of an existing Nastran file and execute a LS-DYNA analysis. This very unique feature allows for the sharing of models in this somewhat generic format.

FEMB 28 implements a template-based NASTRAN interface capability enabling LS-DYNA users change the default LS-DYNA environment to a NASTRAN environment. The interface may also be used in the "opposite direction", converting the NASTRAN environment to the LS-DYNA environment. The template based interface offers the flexibility of updating the software with new capabilities rapidly, making conversions between the two software (NASTRAN and LS-DYNA) easier and providing read-in and write-out functions.

The approach used to support all of the LS-DYNA functionality was used when developing the NASTRAN interface. The philosophy of supporting all cards per user's manual and no text editing required environment was a primary specification for this template, allowing users ability to specify all NASTRAN software modeling functionality from within the pre processor environment.

Since typical NASTRAN applications are static and dynamic linear analyses, and the primary applications of LS-DYNA are non-linear dynamic applications the conversion from LS-DYNA to NASTRAN is easy but the reverse is rather difficult. In order to provide a standard method for translation of the various modeling entities, default values in LS-DYNA are assigned for the non-defined parameters in NASTRAN. It is recommended that user to take a close look of the conversion table and the LS-DYNA input file for deviation, as shown in the Appendix.

As an example of some of the many capabilities available in LS-DYNA which are not available in NASTRAN; Seat Belt Elements, Air Bag Contacts and Gap Elements, to name as few, are not supported in this conversion, therefore, users must be aware of such limitations.

Advantages of using Template System for LS-DYNA and NASTRAN

The FEMB interface with LS-DYNA was first developed to meet two requirements; namely, the need to be complete as a bundled product and also the ability to update the software rapidly and frequently in order to keep pace with the frequent releases of LS-DYNA.

To accomplish this, the FEMB template system was developed. The template includes all Keyword Cards of LS-DYNA User's Manual and all data entry which will be filtered, either read-in or write-out through this template system.

The primary advantage of using such template system is a complete interface in a way that all cards and default values of LS-DYNA are coded that user do not have to enter them but still have the flexibility to modify them, if desired.

While all default values on each of the keyword cards are defined according to the LS-DYNA manual specifications, modified default values suitable for a particular application may be saved and customized for future use.

All data entry is validated for potential conflict, automatically. For example, if the user entered a character string value for a value which only supports a numeric value, FEMB will detect the error send a message to identify the error mistake.

Creating "smart input fields" has allowed users to utilize additional functionality through the template system. The template manages cards via a condition check. For example some DYNA keywords will use a different card if a specifc field has different value. This is retained in the template, and it will adjust the data requirements accordingly. This puts the user in the position of only specifying the data necessary for their specific conditions.

Another advantage of using the template system is that data may be associated with proper objects such as nodes, elements, parts, etc. For example, clicking an element ID field, FEMB will pop up an element selection dialog. The software detects the entity selected, and displays the appropriate dialogue window for user input.

Since the implementation of such template system for LS-DYNA has been successful, the NASTRAN template was also implemented the same way. With very limited exceptions, all commonly used NASTRAN cards for automotive and aerospace applications are supported.

The requirement of conversion between two drastically different file formats and data structures from infrastructure level is the third most challenge task to test the flexibility and versatile of the template system.

NASTRAN and LS-DYNA Conversion using Template System

The initial template system used for LS-DYNA was drastically expanded to cover not only both NASTRAN and LS-DYNA, but also the deviation of both codes was supported through a Conversion Table (see Appendix for LS-DYNA to NASTRAN conversion, Appendix B for NASTRAN to LS-DYNA conversion).

Knowing the drastically different data structures and intended applications of two codes, the conversion from LS-DYNA to NASTRAN is quite easy, because there is more definition on LS-DYNA side than on the NASTRAN side. In contrast, it is almost impossible to convert completely from NASTRAN to LS-DYNA. In particularly, when non-linear definition is required which is absent from the linear definition of NASTRAN. In order to get the best possible conversion, default values and assumptions are made to get as close as possible for commonly used applications.

For example; the RBE2 is NASTRAN could be used for spot welds, general-purpose rigid body and connection of any kind. In LS-DYNA, RBE2 is converted to the Nodal Rigid Body, as a general-purpose approach.

Supporting LS-970 Capabilities

As features and capabilities are added to the LS-DYNA PC software, FEMB 28 will be updated to support each of the features. The template system is periodically updated to allow users the ability to access these features, at major release dates. The current template for FEMB version 28 supports all of LS-DYNA 960, with additional items currently being implemented to support all of LS-DYNA version 970 shortly after the official release.

Conclusions

The template system is the key infrastructure capability that enables FEMB to support with LSTC's rapid development efforts. The advantage of such capability is shown in this release of FEMB which supports most of the solver capabilities in a synchronized fashion.

The NASTRAN interface and the ability of FEMB in delivering effortless conversions will encourage NASTRAN users to take advantage of LS-DYNA's implicit solver for their static and dynamic linear applications.

The topology automesher is a major enabler to reduce the need for manual work to repair CAD surface data prior to meshing and provides a state of the art tool to boost meshing productivity.

The availability of the topology meshing technology makes tetra automeshing a practical meshing tool to support solid element generation.

It is ETA's commitment to continuously develop pre and post-processing capability to support LS-DYNA usage and in the next release. Users may look forward to many exciting features which will be implemented, such as direct CAD interface with UG, CATIA V4/V5, Solid Works and Pro-E, and other popular CAD packages.

With respect to post-processing, a Macro system will be implemented to allow automated post-processing of all LS-DYNA output data. A web based "Reporter" software will help the user assemble and publish an internet-ready report.

As a major release, users may expect to see the Unix and Windows versions unified by the 4^{th} Quarter of 2003. This release, FEMB/PC 29 will bring all PC capabilities to that level found the UNIX/LINUX versions of the software.

Appendix A. Conversion from LS-DYNA to NASTRAN

```
============== ELEMENTS =================
*ELEMENT_SOLID         =>      CHEXA/CPENTA/CTETRA
*ELEMENT_SHELL         =>      CTRIA3/CQUAD4
*ELEMENT_DISCRETE      =>      CELAS1 + CORD2R
*ELEMENT_BEAM          =>      CBAR/CBEAM/CROD/CTUBE
*CONSTRAINED_SPOTWELD  =>      RBAR
*CONSTRAINED_NODAL_RIGID_BODY =>      RBE2
*CONSTRAINED_NODE_SET  =>      RBE2
*ELEMENT_TSHELL        =>      CHEXA
*ELEMENT_INERTIA       =>      MASS
*ELEMENT_MASS          =>      MASS
*ELEMENT_SEATBELT      =>      Discard
*CONSTRAINED_JOINT     =>      Discard
*ELEMENT_SPH           =>      Discard

============== MATERIALS ================
All materials  =>      MAT1

============== PROPERTIES ===============
*SECTION_SOLID         =>      PSOLID
*SECTION_SOLID_ALE     =>      PSOLID
*SECTION_TSHELL        =>      PSOLID
*SECTION_SHELL         =>      PSHELL
*SECTION_SHELL_ALE     =>      PSHELL
*SECTION_BEAM          =>      PBAR
*SECTION_BEAM(CST=1 or ELFORM = 5) => PTUBE
*SECTION_DISCRETE      =>      PELAS
*SECTION_SPH           =>      Discard
*SECTION_SEATBELT      =>      Discard

================= LCS ===================
*DEFINE_COORDINATE_NODES      =>      CORD2R
*DEFINE_COORDINATE_SYSTEM     =>      CORD2R
*DEFINE_COORDINATE_VECTOR     =>      CORD2R

================= SPC ===================
*BOUNDARY_SPC_NODE     =>      SPC
*BOUNDARY_SPC_SET      =>      SPC

============ Initial Velocity ===========
*INITIAL_VELOCITY_NODE        => TIC
*INITIAL_VELOCITY_SET         => TIC
*INITIAL_VELOCITY_GENERATION  => TIC

================= Force =================
*LOAD_NODE_POINT       =>      FORCE
*LOAD_NODE_SET =>      FORCE

================ Pressure ===============
*LOAD_SHELL_ELEMENT    =>      PLOAD4
*LOAD_SHELL_SET        =>      PLOAD4
*LOAD_SEGMENT          =>      Discard
*LOAD_SEGMENT_SET      =>      Discard
```

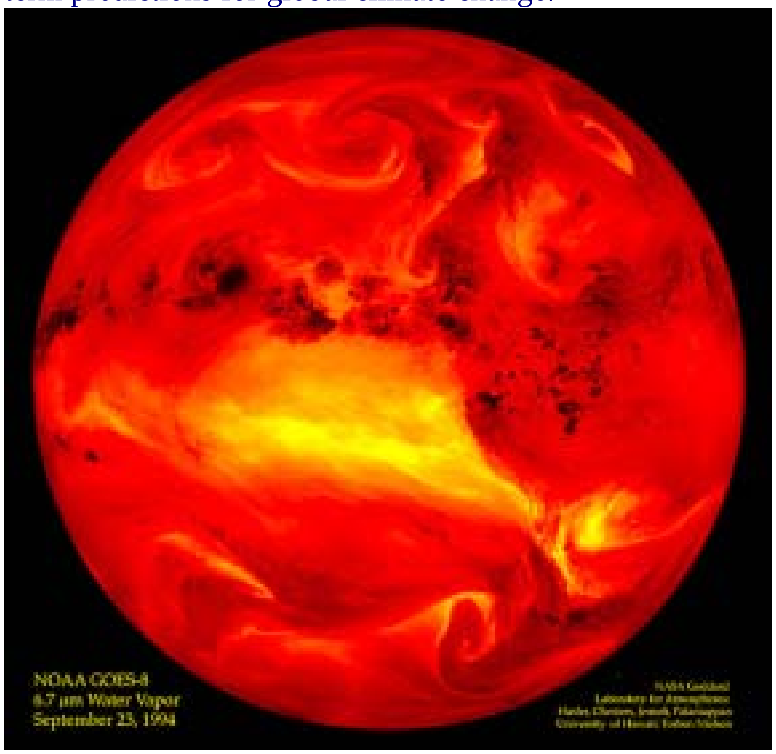
Appendix B. Conversion from NASTRAN to LS-DYNA

```
============== ELEMENTS =================
CHEXA/CTETRA/CPENTA    =>      *ELEMENT_SOLID
CQUAD4/CTRIA3          =>      *ELEMENT_SHELL
CQUAD8/CTRIA6          =>      *ELEMENT_SHELL
CBAR/CBEAM/CROD/CTUBE  =>      *ELEMENT_BEAM
RBAR    =>      *CONSTRAINED_SPOTWELD
RBE2    =>      *CONSTRAINED_NODAL_RIGID_BODY (Depends on DOF) +
                *SET_NODE
RBE2    =>      *CONSTRAINED_NODE_SET (Depends on DOF) +
                *SET_NODE
CELAS1  =>      *ELEMENT_DISCRETE +
                *DEFINE_SD_ORIENTATION
CONM2 =>        *ELEMENT_MASS
CGAP    =>      Discard
RBE3    =>      Discard

============== MATERIALS  ===============
MATS1           =>      Discard
MATT1-MATT9     =>      Discard
Others  =>      *MAT_ORTHOTROPIC_ELASTIC

============== PROPERTIES ===============
PLSOLID =>      *SECTION_SOLID
PSOLID  =>      *SECTION_SOLID
PCOMP   =>      *SECTION_SHELL
PLPLANE =>      *SECTION_SHELL
PSHEAR  =>      *SECTION_SHELL
PSHELL  =>      *SECTION_SHELL
PROD    =>      *SECTION_BEAM(type:TRUSS)
PTUBE   =>      *SECTION_BEAM(type:INTEGD)
PBCOMP  =>      *SECTION_BEAM(type:RESULT)
PBEAM   =>      *SECTION_BEAM(type:RESULT)
PBAR    =>      *SECTION_BEAM(type:RESULT)
PBEND   =>      *SECTION_BEAM(type:RESULT)
PBUSH   =>      *SECTION_DISCRETE
PBUSH1D =>      *SECTION_DISCRETE
PDAMP   =>      *SECTION_DISCRETE
PELAS   =>      *SECTION_DISCRETE
PGAP    =>      Discard

================= LCS ===================
ALL     =>      *DEFINE_COORDINATE_SYSTEM

================= SPC ===================
SPC     =>      *BOUNDARY_SPC_SET
SPC1    =>      *BOUNDARY_SPC_SET

================= TIC ===================
TIC     =>      *INITIAL_VELOCITY

=======FORCE/MOMENT/PLOAD4(F.M.P4)=======
FORCE   =>      *LOAD_NODE_SET +
                       *SET_NODE +
                       *DEFINE_CURVE
PLOAD4  =>      *LOAD_SHELL_SET +
                       *SET_NODE +
                       *DEFINE_CURVE
MOMENT  =>      Discard
```

Computer Modeling Helps Unlock Secrets of Global Climate

Images courtesy of the Laboratory for Atmospheres, NASA Goddard Space Flight Center
Article with enlarged graphics can be viewed at:
www.sgi.com/features/2003/feb/weather/index.html

SGI® high-performance computing systems running advanced computer models are proving to be powerful tools for understanding the intricacies of our planet's climate. Through the computation of massive and complex mathematical relationships, SGI supercomputers are helping to unlock the mysteries of our global environment. By accurately modeling atmospheric and oceanic conditions, scientists can not only dissect what is happening in the current global climate system, but can also make near- and long-term predictions for global climate change.



This image, taken by GOES-8 on September 23, 1994, shows atmospheric water vapor observed at a wavelength of 6.7 microns.

From portraying the current climate more quantitatively to simulating future global warming scenarios, NASA scientists are objectively evaluating the effects of natural and human activities on global climate using two SGI supercomputers that are among the most powerful systems of their kind. The SGI supercomputers driving this breakthrough in global climate research include a 1,024-processor SGI® Origin® 3800 single system image supercomputer at NASA Ames Research Center, Moffett Field, California, and a 512-processor SGI Origin 3800 supercomputer at NASA Goddard Space Flight Center, Greenbelt, Maryland

These two SGI supercomputers have achieved a tenfold improvement running earth science applications using optimization techniques and multilevel parallelism (MLP) software developed at NASA Ames in cooperation with SGI. This increase in performance allows earth scientists to complete climate simulations in days rather than months, providing the performance needed to better understand how human activity has changed climate patterns.

To improve the prediction capabilities of the climate models, the 1,024-processor SGI Origin 3800 supercomputer at NASA Ames assimilates thousands of gigabytes of observational data from the whole Earth to make a database to verify the physics of the computer model. NASA Ames then backs up a few years and runs the climate model to see how good its predictions are. The computer models can then be adjusted to improve their accuracy for future prediction.

Supercomputers are needed to handle the immense volume of data contained in climate models obtained from satellites, aircraft, and ground-based sensors. The primary user of the new SGI Origin 3800 supercomputer is the Data Assimilation Office (DAO) at NASA Goddard, which is responsible for taking global satellite observations and converting the data into NASA's next generation of climate models to better understand the physics of weather.

"With the SGI Origin 3800 system, NASA will more than double the amount of data it ingests to 800,000 observations each day," said Dr. Richard B. Rood, acting chief of the Earth & Space Data Computing Division, NASA Goddard, and former chief scientist at DAO. "We will also integrate assimilation systems for several satellites so that, like the real Earth, the impact of one type of data will be felt by another type of data."

Perspective view of Hurricane Luis on 8 September 1995 at 17:45 UT from GOES-8.

The SGI Origin 3800 system's processing power, along with the MLP software that takes advantage of its unique memory design, will enable the DAO climate models to run more than four times faster and at double the resolution than was previously achieved. Climate models divide the globe into a grid of stacked boxes, solving the relevant equations in each box and then assembling the full results. With a box only a half-degree wide (or 30 miles over the continental United States), the model will more faithfully represent atmospheric conditions worldwide for periods as long as 15 years.

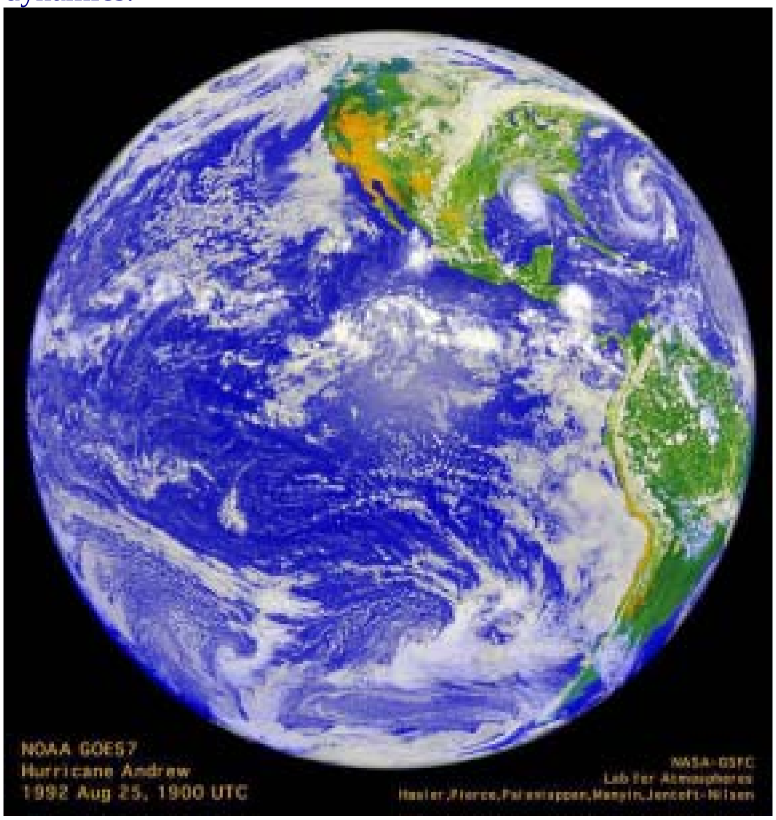
"This more capable computer will allow us to employ more realistic representations of the global climate systems in our attempts to understand climate change that has already occurred and to predict climate change that will occur throughout the 21st century," said Dr. James E. Hansen, director, Goddard Institute for Space Studies. "Our most pressing needs are to represent the full atmosphere--troposphere and stratosphere--with adequate vertical resolution, and to represent the ocean with better horizontal and vertical resolution."

A recently installed 512-processor SGI Origin 3800 supercomputer at Los Alamos National Laboratory, New Mexico, is allowing scientists to complete intricate climate simulations in weeks rather than months and to perform high-resolution ocean model simulation. By providing coherent shared memory across 512 processors, this SGI Origin 3800 supercomputer at Los Alamos National Laboratory has dramatically increased code performance and had a major impact on global climate modeling. Understanding the processes that control the Earth's climate and predicting future changes in climate due to natural and anthropogenic influences require computer models of the climate system that describe the time-evolving circulation and thermodynamics of the atmosphere and oceans, the two main components of the climate system.

The Parallel Ocean Program (POP) was developed at Los Alamos National Laboratory under the Department of Energy's Climate Change Prediction Program (formerly CHAMMP), which was established to rapidly advance the science of decade- and longer-scale climate prediction. Los Alamos National Laboratory has installed the POP model to run on the SGI Origin 3800 machine to perform high-resolution ocean model simulation.

"Six years ago, we did a simulation of the North Atlantic climate over a 10-year period that took five months to complete. Now, with a more powerful, dedicated computer, we're substantially cutting the run time for these models. What used to take two months now takes a week," said Robert Malone, a member of Los Alamos National Laboratory's climate ocean and sea ice modeling team.

POP, which has become a standard model for many climate simulators, can now resolve one-tenth of a degree of the ocean's surface, the equivalent of 10 km at the equator. The finer resolution makes a difference because it enables the researchers to simulate ocean eddies, which play a key role in ocean dynamics.



Full view of the Earth, taken by GOES-7 on August 25, 1992. The image shows Hurricane Andrew approaching Louisiana.

By utilizing the power of supercomputers, numerical model simulation is an alternative to direct observation and experimentation and is a key method for studying various oceanographic and atmospheric phenomena, especially in the research of climate prediction, such as El Niño. Los Alamos National Laboratory's main objective is to optimize the parallel ocean model to run effectively on massively parallel processing supercomputers like the lab's new 512-processor SGI Origin 3800.

Weather forecasting and climate research represent the most demanding tasks in supercomputing today and SGI is the clear leader in the field. Meteorology requires a combination of extraordinary power and extraordinary availability: the system must operate 24 hours a day, 365 days a year--without failure.

Because the amount of data required to run climate models is exploding, the demands for data management computing and system reliability are ever expanding. SGI continues to meet and surpass those demands for more than 35 national and regional sites worldwide, maintaining a staff of highly skilled experts in weather modeling applications who work with national weather centers around the world.

Special Announcements and Highlights of News Pages

Posted on FEA Information in February - Archived on the news page

February 03	**ETA - FEMB 27-PC**
	HP - Superdome
	Distributor: Altair Italy
February 10	**Intel® - Pentium® 4 processor**
	CEI – EnLiten
	Distributor: Altair Western Region, USA
February 17	**Fujitsu – PRIMERGY Server Product line**
	AMD – AMD Athlon™
	Distributor: CAD-FEMB GmbH, Germany
February 24	**ANSYS - ANSYS EMAG™:**
	LSTC – LS-DYNA
	Distributor – LEAP, Australia

Special Announcement: LSTC's optimization program LS-OPT is now included as part of the LS-DYNA License. It is currently available for machines running the Unix and Linux operating systems. Contact Dr. Wayne Mindle for information wlm@lstc.com and please mark the subject line LS-OPT Offer

FEA Information Participants

Headquarters	Company	
Australia	Leading Engineering Analysis Providers	www.leapaust.com.au
Canada	Metal Forming Analysis Corp.	www.mfac.com
France	Cril Technology Simulation	www.criltechnology.com
Germany	DYNAmore	www.dynamore.de
Germany	CAD-FEM	www.cadfem.de
India	GissEta	www.gisseta.com
Italy	Altair Engineering srl	www.altairtorino.it
Italy	Numerica srl	www.numerica-srl.it
Japan	The Japan Research Institute, Ltd	www.jri.co.jp
Japan	Fujitsu Ltd.	www.fujitsu.com
Korea	THEME Engineering	www.lsdyna.co.kr
Korea	Korean Simulation Technologies	www.kostech.co.kr
Russia	State Unitary Enterprise - STRELA	www.ls-dynarussia.com
Sweden	Engineering Research AB	www.erab.se
Taiwan	Flotrend Corporation	www.flotrend.com
UK	OASYS, Ltd	www.arup.com/dyna
USA	INTEL	www.intel.com
USA	Livermore Software Technology	www.lstc.com
USA	Engineering Technology Associates	www.eta.com
USA	ANSYS, Inc	www.ansys.com
USA	Hewlett Packard	www.hp.com
USA	SGI	www.sgi.com
USA	MSC.Software	www.mscsoftware.com
USA	DYNAMAX	www.dynamax-inc.com
USA	CEI	www.ceintl.com
USA	AMD	www.amd.com
USA	Dr. T. Belytschko	Northwestern University
USA	Dr. D. Benson	Univ. California – San Diego
USA	Dr. Bhavin V. Mehta	Ohio University
USA	Dr. Taylan Altan	The Ohio State U – ERC/NSM
USA	Prof. Ala Tabiei	University of Cincinnati
Russia	Dr. Alexey I. Borovkov	St. Petersburg State Tech. University
Italy	Prof. Gennaro Monacelli	Prode – Elasis & Univ. of Napoli, Federico II

EVENTS – CONFERENCES – EXPO's

2003	
May 13	**HP's 14th annual technology trends in engineering symposium** being held in Plymouth, MI
May 19-21	**BETECH 2003** taking place at the Hyatt Regency Dearborn hotel in Detroit, USA - 15th International Conference on Boundary Element Technology
May 22 - 23	**4th European LS-DYNA Conference** will be held in ULM, Germany
June 3-5	**Testing Expo 2003**, Stuttgart, Germany.
June 17-20	**The Second M.I.T. Conference on Computational Fluid and Solid Mechanics**, taking place at Massachusetts Institute of Technology Cambridge, MA.,USA
Oct 29-31	**Testing Expo North America 2003 -** located Novi Expo Center in Detroit, Michigan.
Nov 12-14	**CAD-FEM User Conference 2003** - Dorint Sanssouci Hotel, Berlin Potsdam.

If you have an event you would like posted send it to mv@feainformation.com