

**FEA Information International News
For The
Worldwide Engineering Community**



**1st Anniversary Issue
October, 2001**

**FEA Information Inc.
www.feainformation.com**



Featured Articles

FEA Information International News:

Managing Editor

Trent Eggleston

Feature Editor

Marsha Victory

Technical Writer

David Benson

Art Director

Wayne Mindle

The contents of FEA Information International News is deemed to be accurate and complete. However, FEA Information Company doesn't guarantee or warrant accuracy or completeness of the material contained in this publication.

All trademarks are the property of their respective owners.

FEA Information International News is published for FEA Information Inc..

© 2001 FEA Information Inc. International News. All rights reserved. Not to be reproduced in hardcopy or electronic format without permission of FEA Information Inc..

Pg	Featured Articles
03	<i>Smoothed Particle Hydromatics – Part I of II</i> Jean Luc Lacomme, LSTC
07	<i>Editing LS-DYNA Input Files using Oasys PRIMER Version 8.1</i>
12	<i>History & Future of Computing – FAQ Part 3 of 3</i> Paul Bemis, ANSYS
14	<i>Contact Modeling in LS-DYNA part 3 of 4</i> Suri Bala, LSTC
17	<i>Eta/FEMB27-PC</i> Armando Esteves, ETA
19	<i>FEA Information Inc. World Wide Participant Introduction:</i> The Japan Research Institute, Limited
20	<i>Courses & Events</i>
21	<i>Web Site Summary</i>
22	<i>Monthly Product Showcase</i>

DO NOT OPEN e-mail from:
feainformation.com
FEA Information Inc. ONLY uses
vic@lstc.com
feainfo@lstc.com

INTRODUCTION

Smoothed Particle Hydrodynamics (SPH) is an N-body integration scheme developed by Lucy, Gingold and Monaghan (1977). The method was developed to avoid the limitations of mesh tangling encountered in extreme deformation problems by the conventional finite element method. The uniqueness of SPH is the absence of background grids. Therefore, partial differential equations, such as conservation laws, are transformed into integral equations. The kernel estimate then provides approximation functions to estimate field variables at discrete points. This paper is devoted to the presentation of this method. The first part will introduce the foundational theories of this method. In part II, we will present the development of this new feature in LS-DYNA and the coupling method for SPH and the finite element method.

I - Definitions

The particle approximation of a function is:

$$\Pi^h f(x) = \int f(y)W(x-y, h)dy$$

Where W is the kernel function.

The Kernel function W is defined as:

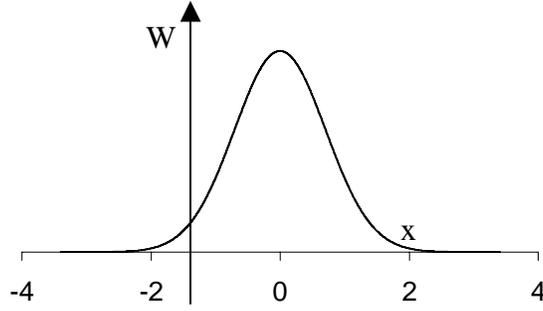
$$W(\mathbf{x}, h) = \frac{1}{h(\mathbf{x})^d} \theta(\mathbf{x})$$

Where d is the number of spatial dimensions and h is called *smoothing length* which varies in time and in space, and x is the location of the particle.

Usually, $W(\mathbf{x}, h)$ is a centrally peaked function. The common smoothing kernel θ used by the SPH community is the cubic B-spline.

$$\theta(u) = C \times \begin{cases} 1 - \frac{3}{2}u + \frac{3}{4}u^2 & \text{for } 0 \leq u \leq 1 \\ \frac{1}{4}(2-u)^2 & \text{for } 1 \leq u \leq 2 \\ 0 & \text{else} \end{cases}$$

Where C is a constant of normalization that depends on the number of spatial dimensions.



The SPH method is based on the quadrature formula for moving particles $((\mathbf{x}_i(t)) \ i \in \{1..N\})$, where $\mathbf{x}_i(t)$ is the location of particle i , which moves along the velocity field \mathbf{v} .

The particle approximation of a function can now be defined by:

$$\Pi^h f(\mathbf{x}_i) = \sum_{j=1}^N w_j f(\mathbf{x}_j) W(\mathbf{x}_i - \mathbf{x}_j, h)$$

Where $w_j = \frac{m_j}{\rho_j}$ is the “weight” of the particle j . The weight of a particle varies proportionally to the divergence of the flow.

The SPH formalism implies the use of a derivative operator. So we need to define a particle approximation for this operator. Before giving the definition of this approximation, we define the gradient of a function as:

$$\nabla f(x) = \nabla f(x) - f(x) \nabla 1(x)$$

Where 1 is the unit function.

Starting from this relation, we can define the particle approximation of the gradient of a function:

$$\Pi^h \nabla f(\mathbf{x}_i) = \sum_{j=1}^N \frac{m_j}{\rho_j} [f(\mathbf{x}_j) A_{ij} - f(\mathbf{x}_i) A_{ij}]$$

where $A_{ij} = \frac{1}{h^{d+1}} \theta \left(\frac{\|x_i - x_j\|}{h} \right)$

We can also define the particle approximation of the partial derivative $\frac{\partial}{\partial x^\alpha}$:

$$\Pi^h \left(\frac{\partial f}{\partial x^\alpha} \right) (\mathbf{x}_i) = \sum_{j=1}^N w_j f(\mathbf{x}_j) A^\alpha(\mathbf{x}_i, \mathbf{x}_j)$$

where A is the operator defined by : $A(x_i, x_j) = \frac{1}{h^{d+1}} \frac{(x_i - x_j)}{\|x_i - x_j\|} \theta \left(\frac{\|x_i - x_j\|}{h(x_i, x_j)} \right)$

A^α is the α -th component of the A vector.

II - Discrete form of conservative equations

We are looking for the solution of the equation:

$$L_v(\phi) + \text{div}F(\mathbf{x}, t, \phi) = S$$

where $\phi \in R^d$ is the unknown, F^β with $\beta \in \{1..d\}$ represents the conservation law and L_v is the transport operator defined by:

$$L_v : \phi \rightarrow L_v(\phi) = \frac{\partial \phi}{\partial t} + \sum_{l=1}^d \frac{\partial(\mathbf{v}^l \phi)}{\partial x^l}$$

- The strong formulation approximation:

For the strong form solution, the equation is kept at its initial formulation. The discrete form of this equation implies the definition of the operator of derivation D defined by:

$$D : \phi \rightarrow D\phi(x) = \nabla \phi(x) - \phi(x)\nabla \mathbf{1}(x)$$

The particle approximation of this operator is:

$$D_h \phi(\mathbf{x}_i) = \sum_{j=1}^N w_j (\phi(\mathbf{x}_j) - \phi(\mathbf{x}_i)) A_{ij}$$

where A_{ij} is defined previously.

Finally, the discrete form of the strong formulation is written:

$$\frac{d}{dt} (w_i \phi(\mathbf{x}_i)) + w_i D_h F(\mathbf{x}_i, t, \phi) = w_i S(\mathbf{x}_i)$$

However this discrete form is not conservative; therefore the strong formulation approximation is not acceptable for the numerical computation. Thus, we are compelled to use the weak form.

- The weak formulation approximation :

In the weak form formulation, the adjoint of the L_v operator is used:

$$L_v^* : \phi \rightarrow L_v^*(\phi) = \frac{\partial \phi}{\partial t} + \sum_{l=1}^d v^l \frac{\partial \phi}{\partial x^l}$$

The discrete form of this operator corresponds to the discrete formulation of the adjoint of $D_{h,s}$:

$$D_{h,s}^* \phi(\mathbf{x}_i) = \sum_{j=1}^N w_j (\phi(\mathbf{x}_i) A_{ij} - \phi(\mathbf{x}_j) A_{ji})$$

A discrete adjoint operator for the partial derivative is also necessary, and is taken to be the α -th component of the operator:

$$D_{\alpha}^* \phi(\mathbf{x}_i) = \sum_{j=1}^N w_j \phi(\mathbf{x}_j) A^{\alpha}(\mathbf{x}_i, \mathbf{x}_j) - w_j \phi(\mathbf{x}_i) A^{\alpha}(\mathbf{x}_j, \mathbf{x}_i)$$

Clearly, the final form of the discrete equation is conservative. Hence, all the conservative equations encountered in the SPH method will be solved using this weak form. It has been proved that this approximation is $P0$, which means that we have a good particle approximation for constant functions and that error estimate between any function and its particle approximation is known.

III - Applications to conservative equations

With the definitions explained above, the conservative equations can now be written in their discrete form.

- Momentum conservation equation :

The momentum equation is:

$$\frac{d\mathbf{v}^{\alpha}}{dt}(\mathbf{x}_i(t)) = \frac{1}{\rho_i} \frac{\partial(\sigma^{\alpha\beta})}{\partial x_i}(\mathbf{x}_i(t))$$

where α, β are the space indices, v is the velocity and σ is the stress tensor.

The particle approximation of the weak form of this equation is:

$$\boxed{\frac{d\mathbf{v}^{\alpha}}{dt}(\mathbf{x}_i) = \sum_{j=1}^N m_j \left(\frac{\sigma^{\alpha,\beta}(\mathbf{x}_i)}{\rho_i^2} A_{ij} - \frac{\sigma^{\alpha,\beta}(\mathbf{x}_j)}{\rho_j^2} A_{ji} \right)}$$

- Energy conservation equation :

The energy equation is:

$$\frac{dE}{dt} = -\frac{P}{\rho} \nabla \mathbf{v}$$

where E is the internal energy, P is the pressure and ρ is the density. The particle approximation of the weak form of this equation is:

$$\boxed{\frac{dE}{dt}(\mathbf{x}_i) = -\frac{P_i}{\rho_i^2} \sum_{j=1}^N m_j (v(\mathbf{x}_j) - v(\mathbf{x}_i)) A_{ij}}$$

Editing LS-DYNA Input Files using *Oasys PRIMER Version 8.1*
 © Copyright Oasys, 2001

PRIMER is a pre-processor dedicated to editing LS-DYNA input files. While there are many pre-processors available for mesh-building and general modeling, their support for non-linear input data is often incomplete: specialist data such as load-curves, joints, complex material models and so on can be lost when an existing analysis is taken back into them for modification, or when models are merged. In addition there are several specialist functions peculiar to particular types of analysis, such as occupant positioning, which are not provided satisfactorily by general-purpose pre-processors. *Oasys PRIMER* has been designed to solve these problems.

Primer is capable of reading, processing and writing out the entire LS-DYNA keyword input deck, with no exceptions or omissions: no information is lost during processing. It will also read and write several other common formats. Input decks may be visualized directly, and any number of input models may be merged intelligently into a single output model; with the additional ability to translate, rotate, reflect and scale models, parts or individual components in the process.

PRIMER provides several specialist occupant modeling features. It will manipulate occupant (dummy) models intelligently, providing easy interactive positioning of whole models and their constituent parts; it will fold airbags; it will fit seatbelts. This positioning, folding and fitting information is appended to the end of the output deck, and can be reread back in directly for further manipulation if required.

The Purpose of PRIMER: The standard approach to using PRIMER is illustrated in Figure 1. A general purpose pre-processor is used to create the Finite Element mesh from CAD data. Primer imports the mesh where all of the many LS-DYNA specific features can be added. In addition to editing these features, PRIMER aids the user by visualizing them and providing detailed cross-referencing from feature to feature. Both by using the edit menus and by using Find Attached.

Further data can be combined with the model from other sources such as the Bill of Materials, Materials Database and Spotweld file. The required mass can be applied to the model using the Massing function. Other models can be combined at this point. One of the original reasons that PRIMER was developed was to assist in merging Occupant models, handling the issues related to positioning, contact definition and seatbelt fitting. Finally, the model can be checked, with over 1500 separate checks carried out as well as new functions to check contact penetration, mass scaling requirements. PRIMER provides checks of material and section properties using contour plotting.

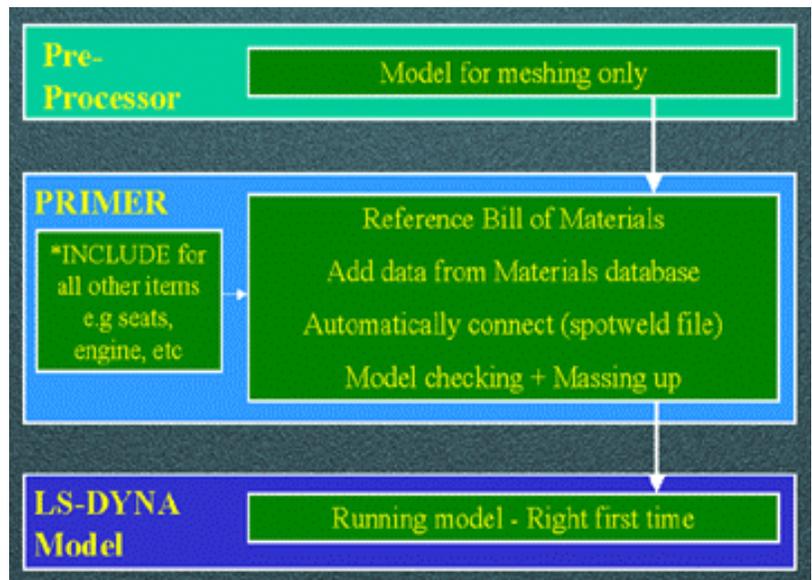


Figure 1 The Purpose of PRIMER

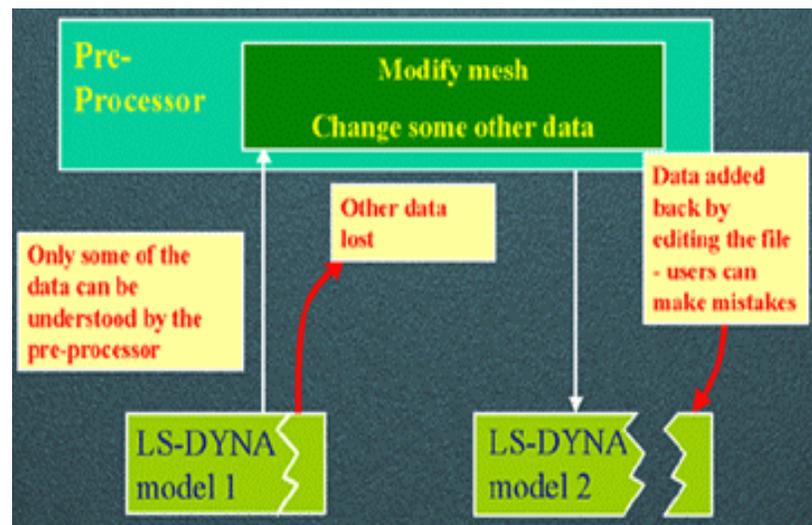


Figure 2 Typical model modification process

While all this is very useful when preparing a new model for analysis, there are more reasons why PRIMER should be used when modifying a model for second and subsequent runs. The typical process to modify an existing model is shown in Figure 2. Most pre-processors lose data on reading the LS-DYNA model because not all features are supported. Although it is sometimes possible to re-stitch this lost data back in after the model is written again (one of the great benefits of the Keyword format of course), mistakes can be made and a lot of time can be wasted.

The procedure with PRIMER is improved, as shown in Figure 3. Mesh modifications are made in the pre-processor as before. The model to be modified is read in by PRIMER along with the new or modified mesh; PRIMER's powerful model merging is then used to combine the two with full checking to avoid clashes. Any further changes can then be made in PRIMER before writing the LS-DYNA model, ready for a new analysis.

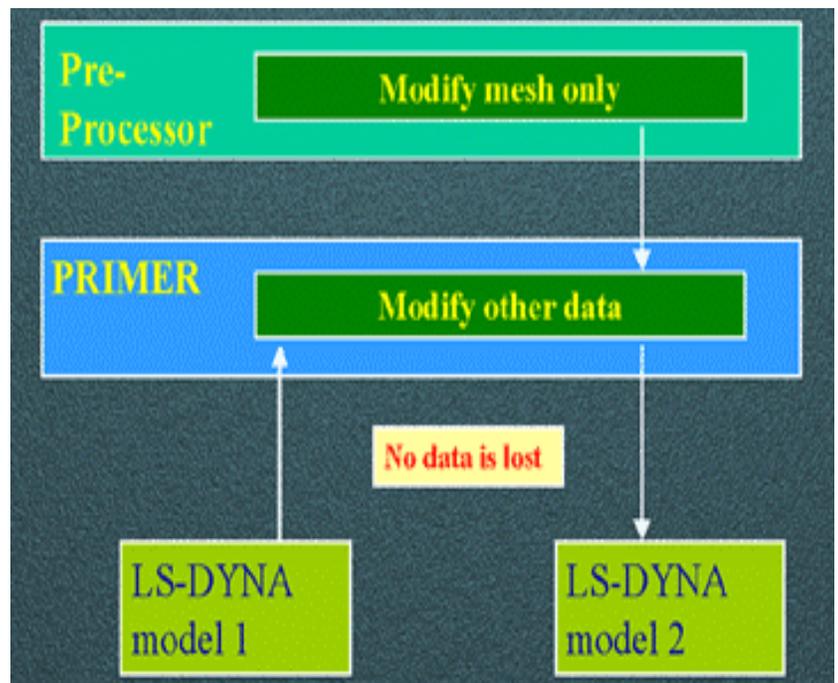


Figure 3 Model modification process with PRIMER

New Modelling Features in Version 8.1

Bill of Materials:

It can be difficult to ensure that a crash model is up to date with the latest product release; material properties and gauge can change for existing parts, and parts can be added or deleted. This information is summarized in the Bill of Materials (BOM) which is usually maintained as a spreadsheet. PRIMER is now able to read this spreadsheet. By matching names between BOM and model PRIMER can be used to ensure that the model is up to date. After reading the spreadsheet, the user must define the meaning of each data column, e.g., Part ID number, material name, gauge, etc. PRIMER then checks the model against the spreadsheet and reports any errors, including parts in the BOM that are not in the model and vice-versa.

Materials Database:

Once the BOM has been read, the materials for each matching part are renamed accordingly. This name can be used to extract the required material properties and values from an existing material database, created by the individual user or system manager. The database is in LS-DYNA Keyword format. As long as some part of the model material name matches the database, a match will occur; e.g., a material name “CR4 Treatment C” will match one named “CR4” in the database. After applying, all matched materials are highlighted; the user may then modify the selection and choose values from the database for unmatched materials. Multiple databases can be created and selected within PRIMER.

Spotweld Definition:

One of the most time-consuming tasks in constructing a vehicle model is the definition of the thousands of spotwelds connecting the body-in-white panels. Up until recently it was necessary for spotweld elements to join nodes in the panel flanges directly, forcing the model to have carefully matched mesh densities on all mating flanges. LS-DYNA, from Version 950, has offered mesh independent spotwelds with the potential to save a great deal of time during mesh creation. Now with the release of PRIMER 8.1, the user can take

full advantage of mesh-independent spotwelds as almost all of the hard work in creating the required nodes, beams, materials and tied contact surfaces has been eliminated. Individual spotwelds can be quickly created by the user. After defining which panels are to be joined, the user only needs to select one node and all the rest is automatic. A second node is projected onto the mating flange and a beam created between them; for a weld through three or more layers, PRIMER will create a chain of beams. The beam material and section properties and the tied contact interface required to stick the projected node to the flange are all easily defined.

Even more powerful, however, is the ability to read a spotweld CAD file. This file, which need only contain the x, y, z coordinates of each weld, can be read from a spreadsheet format. As with the BOM, the user must define which column defines which data. Including the panel part IDs for the weld can also be useful as it can avoid errors later. An example is shown in Figure 4. All possible spotwelds are created in the model by pressing **Apply** once the file is read and the columns are defined. A typical result is shown in Figure 5.

Any spotwelds that cannot be made are reported to the user for individual editing. Individual welds can be isolated and shown; this may reveal that the wrong panels were selected. Other reasons why the welds could not be made can also be displayed. Most often, these will relate to the maximum or minimum length defined for a spotweld, the minimum pitch, the angle to the panel or the maximum number of panels that can be joined with a single weld. Finally, Find Connected and Find Unconnected functions can be used to check that all relevant panels have been successfully spotwelded.

Mass Assign:

Another new useful function for LS-DYNA users is Mass Assign. Often parts in a model are represented by approximate geometry, or a solid body might just be represented by an exterior shell element mesh. PRIMER offers the ability to smear

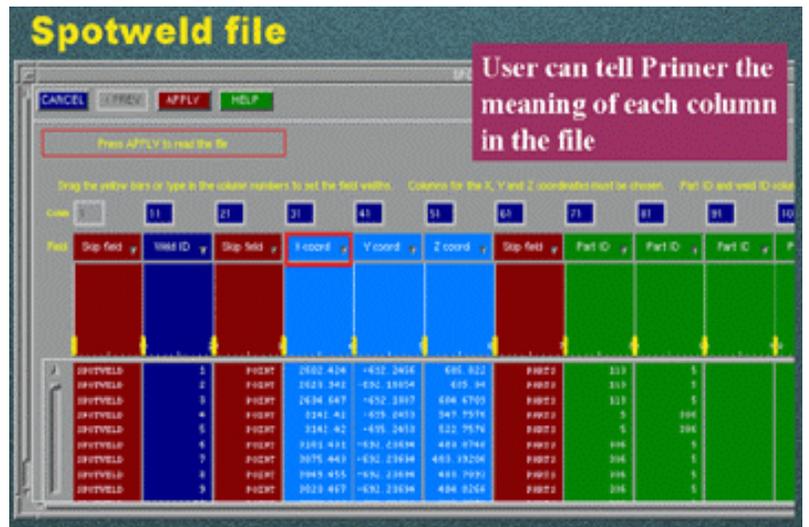


Figure 4 Reading a Spotweld file into PRIMER

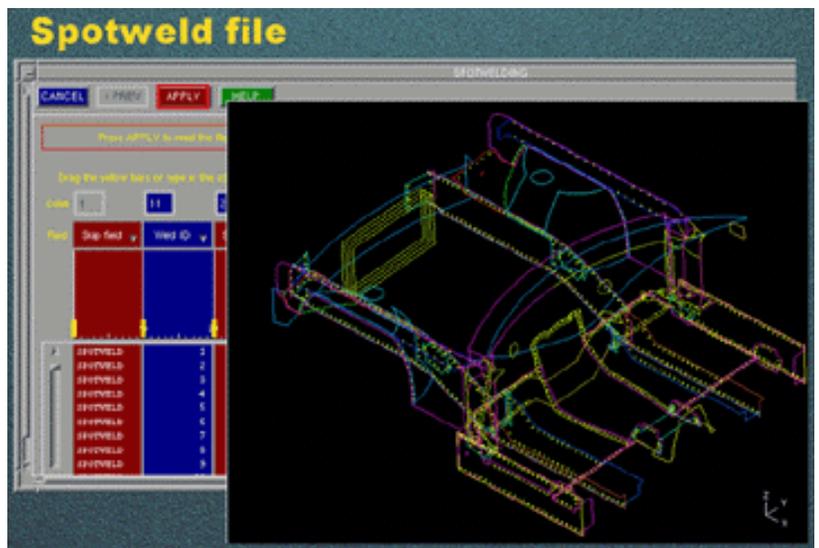


Figure 5 Spotwelds created automatically in PRIMER

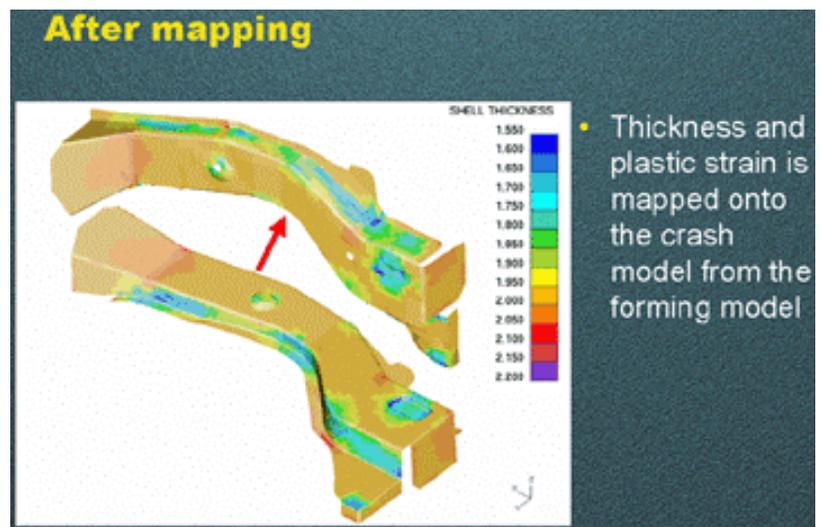


Figure 6 Result of mapping forming data

mass over these parts to obtain the required total mass and centre of gravity.

Effects of Forming:

An area gaining more and more interest is how the forming process affects the performance of the manufactured component. Research has shown that strain hardening and thickness changes can have a significant influence on the crash response of a vehicle, so functions have been added to PRIMER to help users include these effects. Data from a separate forming simulation must be mapped onto the crash model; but often the forming model will have a different mesh density and a different orientation. PRIMER tackles this by defining an orientation based on three nodes in each model and then mapping the thickness and plastic strain data from the forming model to the crash model. A typical result is shown in Figure 6.

Advanced Checking Functions:

PRIMER has always had the capability to check and clean up a model before saving it for analysis. For the Version 8.1 release, even more checks have been added and a number of these now have an Autofix option. Beyond these, several new functions have also been added to visually inspect the model and ensure that the analysis runs “right the first time”.

Find Attached is one of the new visual checking capabilities. This feature allows the user to build the model entity by entity, to ensure that the connections are intended. PRIMER’s ability to visualize all LS-DYNA features gives it a unique capability to visualize the model construction.

Contact Penetration checking is another new addition. Once a contact surface has been defined a check can be made; PRIMER reports the number of penetrating nodes as well as any shells which actually cross each other. The maximum penetration is stated and contour plots of the depth of penetration are available as shown in Figure 7. When the number of penetrations is high, individual nodes can be selected for checking. At present the options to fix penetrations within PRIMER are limited, although null beam elements can be created to help find the problem areas in a meshing pre-processor.

The new contouring capability allows a number of other model characteristics to be checked in PRIMER 8.1. Values that can currently be displayed include shell thickness, material properties such as yield stress, and element timestep and percentage added mass for mass scaling requirements (if a minimum timestep is defined). Vector plots are also now possible. For example, initial velocity can be plotted to check that all entities have the required velocity value (and direction) and any errors due to *PART_INERTIA definitions can be corrected.

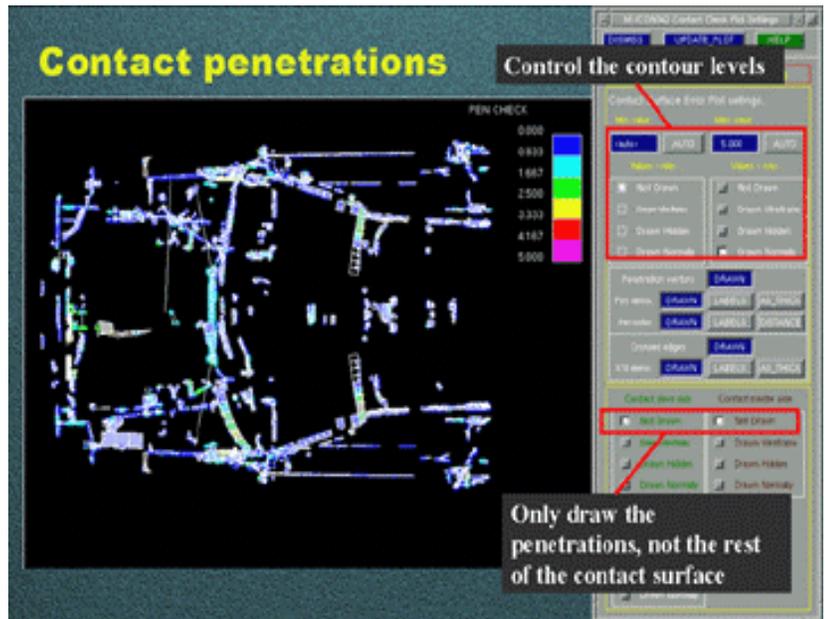


Figure 7 Contour plot of contact penetration

Improvements to General Functionality:

A new header bar has been added to the top of the graphics window giving rapid access to commonly used functions, such as the draw commands (hidden line, shaded image, etc.) and the blanking menu. Some of the new features, such as vector plots of velocity or contour plots of timestep or mass scaling, can also be accessed from here. In addition, new options have been added to modify the appearance of the graphics display and a bitmap output option has been added.

More options for handling the ***INCLUDE** keyword have been added in the latest release. More often, modellers are using ***INCLUDE** to assemble large models from a library of smaller sub-models – this provides a flexible way to manage a complex family of vehicle crash models. PRIMER now offers the option not to read included models when the main model is imported, and offers three alternative ways of handling the included models when writing the main file.

Finally, a new HTML help facility has been added for with all the details of the original online PDF manual but with powerful search and jump functions; this is in addition to the text help menus in the previous version. There is now a **MANUAL** button on every help window which links directly to the relevant page of the HTML manual. Here you will find detailed explanation with diagrams and screen captures to demonstrate the use of the function of interest. Full search and navigation functions allow the user to quickly find related information.

Part 3 of 3 – FAQ on Web Based Distributed Simulation

© Copyright Paul Bemis, 2001

ANSYS, Inc.

Web Based Distributed Simulation

ANSYS has been a pioneer in the area of delivering complete engineering solution systems for over 30 years. Over the past ten years, these solutions have focused primarily on the Engineering Workstation and Personal Computer model of computing due to their outstanding cost effectivity and market demand. However, the trend towards highly complex “real world” engineering simulation often exceeds the capacity of these personal systems. As web based computing becomes more popular and cost effective, the opportunity to use remote computational servers to augment the desktop systems becomes practical. The ANSYS e-Sim strategy is focused on delivering engineering simulation solutions that can be deployed on web based infrastructures for optimal cost effectivity and efficiency in product design environments. The primary benefit of this strategy is by creating ANSYS products that are “web enabled” the deployment of these solutions can be done on LANS, WANS, Intranets, Extranets, or the worldwide web. Furthermore, they can be deployed either within one company, or across the supply chain that services a company.

The following FAQ covers Security, Support and Performance.

Security

Generally, how secure is the e-cae.com infrastructure?

All communications between the users system and e-cae.com utilize state of the art Secure Socket Layer SSL technology. Communication is encrypted and secure. User names and passwords are randomly generated and unique. All operations are continuously monitored for possible misuse. The technology used on e-cae.com provides the highest level of security in the market today. For those users who require excellent security, private VPN or leased lines can be provided, as well as dedicated computational systems. The hosting provider for e-cae.com is Genuity Inc (www.genuity.com) and a tour of the Genuity facility can be arranged for customers who desire to see the security infrastructure “first hand”.

What kinds of security does e-cae.com use to protect customers data?

All customer data is protected using a scheme of unique user and password ID. Advanced security features are used extensively throughout the site and data within e-cae.com is protected using unique naming conventions. Under this method there is no ability for one customer to view another customers data, unless explicitly permitted.

How can I be sure the e-cae.com solution will work with my current corporate firewall?

Provided the customer can “see” the internet, e-cae.com will be operative. There is no incremental work required from the customers IT organization. The e-cae.com solution will use either port 80 or 443 depending whether the customer is operating in “secure” SSL or utilizing an “open” connection.

What precautions are in place to prevent an attack of the e-cae.com site?

Continuous traffic monitoring is in place on a 7x24x365 basis providing the highest degree of pro-active protection available today. Procedures used leverage state of the art technologies and methods to provide extensive security capability.

Will e-cae.com provide a virtual private network (VPN)?

Yes, please ask ANSYS Inc reseller or partner for pricing. VPN’s are available for an added level of security in most geographies worldwide.

Support

How does the support model work?

Support for ANSYS users is provided in the same model as today. ANSYS customers typically purchase through an ANSYS Support Distributor in their local region. This same organization is responsible for the support of the e-cae.com offerings.

Does e-cae.com supply clear reports with detailed application access and usage statistics?

Yes, the reporting capabilities of e-cae.com are quite extensive. Users can access reports online that include parameters such as number of file transfers, number of CPU hours, number of logins, and other data of interest. Customer site administrators have more reporting features including the above statistics for multiple users and/or multiple sites within the customers company.

How does e-cae.com manage the ANSYS version or release migration?

Yes, e-cae.com completely manages application migration for the customer. This includes the rapid “same day” introduction of new ANSYS releases and patches, as well as the operation of older versions of ANSYS for users requiring revision compatibility with installed base application revisions.

What happens when a job that has been submitted hangs, or crashes?

ANSYS has developed a customer support process that will determine the reason for the problem, and appropriate resolution. In cases where it is determined to be an issue with the ANSYS program, a proportionate amount of CPU time will be credited to the customer account.

Performance

How fast is e-CAE.com compared to my local computer?

The answer to this will largely depend on the configuration of the local computer, as well as the size and type of simulation being run. However, in test comparisons between the e-CAE.com service and a local PC operating at 800Mhz, the e-CAE.com server is delivering solutions nearly 3x faster for static analysis in the range of 100K DOF using one CPU. Additionally, when operating ANSYS in parallel mode on e-CAE.com, the time to solution will decline proportionately as processors are applied to the solution.

For pricing and further information contact: Paul Bemis, [paul.bemis@ansys.com] the author of this series article and the manager of The ANSYS, Inc. e-CAE.com ASP Program that provides:

- A mechanism for running **ANSYS** simulations and/or **LS-DYNA** simulations on large parallel compute servers, at a remote data center site using the internet.
- A system that has been developed to allow engineers the ability to run remote simulations with specific controls on job execution parameters.
- A solution that uses “state of the art” security and systems infrastructure technology from providers including Sun, Hewlett Packard, Silicon Graphics, Cisco, and others.
- A service that is ideal for engineers and companies requiring occasional “surge” capacity for time critical simulations, or periodic simulations of large models.

Key e-CAE.com security features:

- HTTP or HTTPS (Secure) access
- Strict account & file controls.
- Full data communication encryption.
- Secure Socket Layer (SSL).
- Continuous monitoring on all operations.

Part 3 - Modeling Guidelines For Full Vehicle Contact

Upcoming issue Part 4: Airbag Contact, Edge-to-Edge Contact, and Rigid Body Contact

7.0 Modeling Guidelines For Full Vehicle Contact

Crash analysis involving a full vehicle incorporates contact interactions between all free surfaces. This is quite expensive since 20-30 percent of the total calculation CPU time is used by the contact treatment. One of the challenging aspects of contact modeling in crash analysis is the handling of interactions between structural metallic parts and non-structural components typically made from foam and plastic. This is especially important when occupants are included in the model. Another challenge is handling contact at corners or edges of geometrically complex parts. Guidelines should be followed to achieve stability in contact as well as reasonable contact behavior. Some of the modeling practices based on experience are discussed below.

7.1 Global or Local Contact

Historically, many individual contact definitions were used for the treatment of contact. The development and implementation of a robust single surface type of contact has changed the way engineers model the contact today. From the standpoints of simplicity in preprocessing, numerical robustness, and computational efficiency, it is now usually advantageous to forsake the use of numerous contact definitions in favor of ONE single-surface-type contact that includes all parts which may interact during the crash event. We often casually refer to this single contact approach as a global contact approach.

This, however, does not mean that one should always avoid local contact definitions. Frequently, there exist certain areas of the vehicle that require special contact considerations where the global contact definition is observed to fail. In such instances the user is encouraged to define local contact interfaces with non-default parameters that would best suit the contact condition.

7.2 AUTOMATIC_SINGLE_SURFACE or AUTOMATIC_GENERAL

Though both contact algorithms belong to the single surface contact type, several key parameters distinguish these two contact types. Table 7.1 highlights the important differences.

Parameters	AUTOMATIC_SINGLE_SURFACE	AUTOMATIC_GENERAL
PENMAX	0.4	100
BSORT frequency	Every 100 cycles	Every 10 cycles
SEARCH DEPTH	2	3
Shell Exterior Edge Treatment	No	Yes
Beam to Beam Contact	No	Yes

Table 7.1 Difference Between AUTOMATIC_SINGLE_SURFACE (13) and AUTOMATIC_GENERAL (26)

Of the two single surface contact types listed in Table 7.1, *AUTOMATIC_GENERAL is computationally more expensive owing to its additional capabilities and its more frequent and thorough contact search.

The AUTOMATIC_SINGLE_SURFACE contact option is recommended for global contact. To treat special contact conditions where shell edge-to-edge or beam-to-beam contact is anticipated, the additional use of the AUTOMATIC_GENERAL contact in localized regions is recommended. AUTOMATIC_GENERAL contact should be used sparingly and only where conditions dictate its use. One advantage of the AUTOMATIC_SINGLE_SURFACE contact starting with LS-DYNA version 950d is in its more rigorous treatment of interior sharp corners within the finite element mesh and in the handling of triangular contact segments; consequently, the AUTOMATIC_SINGLE_SURFACE contact is usually superior for parts meshed from triangular and tetrahedron elements. In future version of LS-DYNA, the AUTOMATIC_GENERAL option will also include these improvements.

7.3 Standard Penalty-Based or Soft Constraint Stiffness Method

When several parts of dissimilar mesh sizes and/or dissimilar material properties are included into one global slave set for AUTOMATIC_SINGLE_SURFACE, the soft constraint stiffness method (SOFT =1) is recommended. The soft constraint method seeks to maximize contact stiffness while also maintaining stable contact behavior. The interacting nodal masses and the global time step are used in formulating the contact stiffness. The segment-based contact method, invoked by setting SOFT=2, calculates contact stiffness much like the soft constraint method but otherwise is quite different. Segment-based contact can often be quite effective where other methods fail at treating contact at sharp corners of parts.

In contrast to a soft constraint approach, the standard penalty-based contact stiffness (SOFT=0) is based on material elastic constants and element dimensions. In foam and plastic materials, the contact stiffness given by the two methods can differ by one or more orders of magnitude. The primary disadvantage of choosing the soft constraint method is its dependence on the global time step. Occasionally, the global time step must be scaled down using the TSSFAC parameter in *CONTROL_TIMESTEP to avoid numerical instabilities in the contact behavior. This results in an increased run time for the entire simulation. As an alternative to reducing the global time step the soft constraint scale factor, SOFSCL, in the *CONTACT definition can be reduced from the default value of 0.1 to 0.04-0.07.

If the standard penalty-based approach is used in a global contact definition, the soft constraint approach can be used locally to handle dissimilar materials in contact. The following are examples where contact behavior may benefit from use of the soft constraint method:

- Airbag to Steering Wheel
- Airbag to Occupant
- Front Tire to SIL
- Spare tire to neighboring components
- Foam to structural components

Using a combination of both contact stiffness methods may promote good contact behavior without having to reduce the global time step.

7.4 Definition of Slave Set

There are several ways to define the slave set for the global contact definition. These include: all parts (this is the default), a set of included parts, a set of excluded parts, or a set of segments. The default, which includes all parts, can sometimes result in obvious instabilities at the beginning of a simulation unless great care is taken in setting up the model to avoid such things as initial penetrations and nonphysical intersections of parts. The option to ignore penetrations on the *CONTROL_CONTACT keyword (set IGNORE equal to 1) is recommended if care is not taken to eliminate initial penetrations. Many models run perfectly with just one

interface definition; others, however, will not run until changes are made to the input, usually by excluding parts or by modifying the finite element mesh to more accurately reflect the physical model.

To reiterate, the following methods can be used for defining the global contact definition:

- All parts (default)
- Included parts by *SET_PART
- Excluded parts by *SET_PART. Non-Excluded parts will be considered for contact
- Segments by *SET_SEGMENT

In addition to the above slave sets, a three-dimensional box, defined using *DEFINE_BOX, may be used to restrict the contact to the parts or segments that lie within the box at the start of the calculation. This will reduce the extent of the contact definition leading to a reduction in contact-associated cpu time.

7.5 Friction

When using one global contact that includes several components of the vehicle, a uniform friction coefficient (possibly zero) may be acceptable for initial analyses. However, the use of *PART_CONTACT keyword to specify friction coefficients on a part-by-part basis is recommended when friction is expected to play a significant role. Friction coefficients specified in *PART_CONTACT will override friction coefficients specified elsewhere if and only if FS in *CONTACT is set to -1.0. Please note that the dynamic friction coefficient FD will have no effect unless a nonzero decay coefficient DC is provided.

7.6 Contact Thickness

To reduce the number of initial penetrations, the contact thickness can be changed from the default element thickness by using the global SST and MST parameters in *CONTACT. The OPTT parameter in *PART_CONTACT can be used to override SST and MST on a part-by-part basis. The user is cautioned against setting the contact thickness to an extremely small value as this practice will often cause contact failure. In fact, for treating contact of very thin shells, e.g., less than 1 mm, it may be necessary to increase the contact thickness to prevent contact failure.

If a contact surface is comprised of tapered shell elements, then a uniform contact thickness should always be specified. The contact assumes that the segment thickness is constant, which can result in thickness discontinuities between adjacent segments. As a node moves between segments of differing thickness, the interface force will either suddenly drop or increase as a result of the discontinuous change in the penetration distance. This can result in negative contact interface energies.

eta/FEMB27-PC
Finite Element Model Builder, version 27 for PC © Copyright, 2001
Armando Esteves, ETA

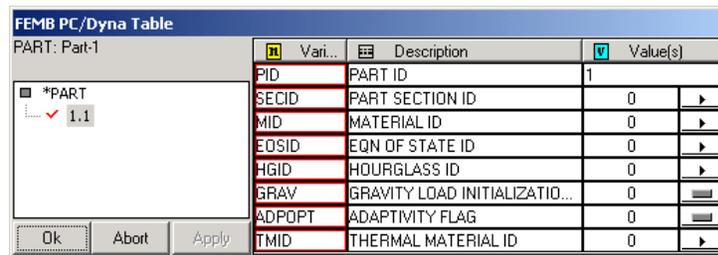
eta/FEMB27-PC is the newly released version of ETA's pre-/post-processor for LS-DYNA for personal computers using Windows 98/NT4/2000. More than a regular upgrade, eta/FEMB27-PC was redesigned from the ground up based on the Unix version of eta/FEMB which is 100% compatible with LS-DYNA.

Complete LS-DYNA Interface: The major goal in the development of eta/FEMB27-PC was to fully support LS-DYNA in a non-text-editing environment. To reach this goal, the following requirements were defined:

- support all LS-DYNA cards and keywords (graphical and numerical);
- set default values as in the LS-DYNA User's Manual and input validation check;
- total graphical interface;
- remain user-friendly and intuitive;
- maintain efficiency and high performance;
- remain the most cost-effective solution available to users.

During development of eta/FEMB27 for unix workstations, ETA designed an "LS-DYNA Template" which allows our developers to easily keep pace with the evolution of LS-DYNA and the release of new keyword commands. A simple addition to our text-based template enables almost immediate compatibility with new or enhanced features of LS-DYNA. In eta/FEMB-PC Version 26, LS-DYNA capabilities were hardcoded into the software, making it difficult to keep up with solver development. To respond effectively to user needs, it was essential to port the LS-DYNA Template to eta/FEMB-PC. This was the primary motivation for a completely new interface. With the advent of the LS-DYNA template, we can also provide the default values as defined in the User's Manual and perform an input validation check according to the expected value range and numerical format.

The introduction of the LS-DYNA Template generates a graphical entity named "LS-DYNA Data Table/Collector" (Fig. 1).



Vari...	Description	Value(s)
PID	PART ID	1
SECID	PART SECTION ID	0
MID	MATERIAL ID	0
EOSID	EQN OF STATE ID	0
HGID	HOURGLASS ID	0
GRAV	GRAVITY LOAD INITIALIZATIO...	0
ADPOPT	ADAPTIVITY FLAG	0
TMID	THERMAL MATERIAL ID	0

Fig. 1: LS-DYNA Data Table/Collector

The LS-DYNA Data Collector is a graphical and intuitive version of the command descriptions in the LS-DYNA User's Manual. It graphically provides the keyword command, the number of cards associated with it, the variable name on each card, a brief description, and a value field for data input and default value (if any). It provides the user with an electronic version of the LS-DYNA User's Manual embedded in the pre-processor interface. Sample windows with the extensive list of Material Keywords (Fig. 2) and the available Contact Keyword selection (Fig. 3) are shown on the next page.

Redesign of the eta/FEMB27-PC started by determining the best way to handle the large amount of data necessary to build an LS-DYNA model. We borrowed from the UNIX version to design the new GUI. We feel that the new design has made the GUI even more organized and intuitive. The fact that the new GUI resembles the UNIX version allows for a smooth transition to the new PC version.

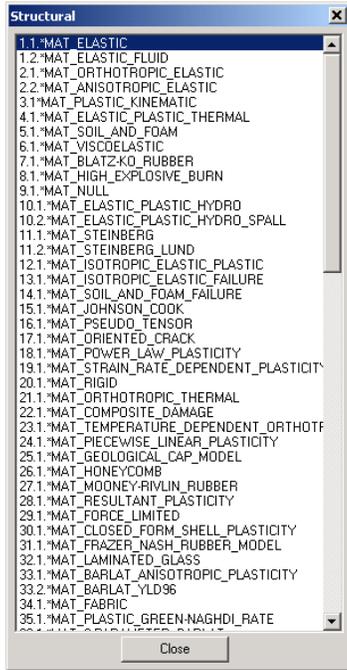


Fig. 2: Structural Material Keyword list



Fig. 3: 3D Contact Keyword List

Productivity is improved by easy command access through function keys and model handling is enhanced with mouse and keyboard combinations. Model handling is also better due to the introduction of mouse drag zoom. The menus are reorganized in a logical way (File / CAD / Model / Properties) and the tools icon menus are preemptive. The simple interface eliminates the steep learning curve normally associated with a full featured program.

eta/FEMB27-PC has extensive CAD capabilities which cover the line and surface definition needs of most power users. Also, more robust IGES and VDA input translators are now available. A powerful new feature is the ability to generate curves (or, load curves) directly from a new graphic plot handler. Curves can be input or inserted (read from a table) in various ways and are immediately associated with the loading case. eta/FEMB27-PC contributes to model data discussion by providing direct export of model images to JPEG format which can be easily inserted into documents, presentations or posted on the Internet. eta/FEMB27-PC allows direct export of the LS-DYNA model to virtual reality VRML format, allowing anyone to manipulate the 3D model using an internet capable browser.

Eta/FEMB27-PC, Compete LS-DYNA Solution Package: Although “FEMB” has historically been perceived as only a pre-processor, eta/FEMB27-PC is in reality a complete LS-DYNA solution package that includes:

- FEMB, pre-processor;
- PostGL, post-processor (d3plot);
- eta/Graph, plot post-processor (ASCII and binary).

Cost-Effective Solution: ETA foresaw the current trend and began several years ago to make the transition from Unix platforms to more cost-effective personal computers. Our goal has been to achieve the same features and ease of use in eta/FEMB27-PC that have been the hallmark of our workstation version.

With our 1st anniversary issue of FEA Information News we are pleased to Showcase one of our participating companies



The Japan Research Institute, Limited

President Masahiko Koido

Message from the President

From Dreams to Reality through "Knowledge Engineering"

www.jri.co.jp

(excerpt from the website: The Japan Research Institute, Limited)

Information technology (IT) is at the heart of a broad process that is radically transforming industry and society not only in Japan but on a worldwide scale. To survive this age of rapid change requires sound business management based on a clear vision of the future and well-thought-out strategies for information utilization. Since its establishment, The Japan Research Institute, Limited (JRI) has kept pace with its clients and their changing needs by basing our operations on the underlying principle of "creating new value for clients." That is, by identifying problems hindering companies and offering concrete, practical proposals for solving those problems, JRI generates new arenas of value for its clients to explore. At JRI, these efforts toward comprehensive problem-solving are understood as "knowledge engineering," a concept which forms the foundation of all our activities.

Offices	Head Office	16 Ichiban-cho, Chiyoda-ku, Tokyo 102-0082, Japan TEL: (81)(3)3288-4700
	Osaka Head Office	5-8 Shinmachi 1-chome, Nisi-ku, Osaka 550-0013 TEL: (81)(6)6534-5111
	Branch Offices	Sapporo, Nagoya, Fukuoka, New York, Singapore
	Representative Office	Hong Kong
	Computer Centers	Tokyo, Higashi Nihon, Gokokuji, Yotsubashi, Kinyu-kyodo

Products for Engineering	
LS-DYNA	Nonlinear Dynamic Analysis of Structures
JOH/NIKE 2D/3D	An implicit Finite-Deformation Finite Element Code
JMAG-Works – JMAG Studio	Electromagnetic Analysis Software
JVISION	Pre-post Processor
JSTAMP-WORKS	A sheet metal forming simulation system

FEA Information Web Sites Monthly Summary

September articles are archived on the News Page at www.feainformation.com

Articles from September on FEA Information website

Sept. 03	New Site	High Performance Computing Servers (www.hpcservers.com)
	Software	PostGL 1.0 from Engineering Technology Associates
	Software	OASYS T/HIS from OASYS, Ltd
	Software	LMS Roadrunner from LMS Int'l
Sept 10	Hardware	hp 9000 unix servers from Hewlett-Packard
	Software	EASI-SEAL® from EASi Engineering
	Distributor	Engineering Research AB [ERAB] located in Sweden
Sept. 17	Product	LifeBook notebook series from Fujitsu Ltd
	Software	LS-POST free post processor from Livermore Software Technology Corp
	Distributor	THEME Engineering, Inc. located in Korea
Sept 24	Software	MSC.Linux version August 2001 from MSC Software
	Software	JOH/Nike 2D/3D from The Japan Research Institute
	Distributor	Metal Forming Analysis Corp. [MFAC] located in Canada

I would like to take this opportunity to thank everyone for helping us publish a monthly publication for the past year.

A special thanks to **Desktop Engineering Magazine, a Helmers Publishing Inc. publication, www.deskeng.com** for allowing FEA Information News to reproduce their articles relating to FEA Information Participants.

In coming issues we will have articles in reference to our engineering websites:

- **High Performance Computing Servers:** www.hpcservers.com - has information online and is scheduled to be fully operational February 2002.
- **Meshless Methods:** www.meshlessmethods.com - will be operational February 2002
- **Fluid-Structure Interaction:** www.fluid-structureinteraction.com - has information online and is scheduled to be fully operational February 2002

Future articles in the news will correlate with our numerous engineering applications websites such as:

- **Heat Transfer Analysis** – www.heattransferanalysis.com
- **Implicit FEA** – www.implicitfea.com
- **Linux for PC** – www.linuxforpc.com

Marsha Victory,
President, FEA Information Inc.

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

Events/Conferences

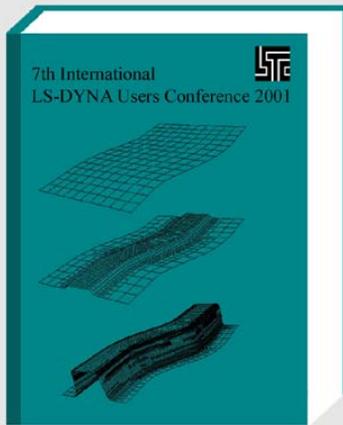
2001	
Oct 30-31 Japan	LS-DYNA Users Conference, sponsored by Japanese Research Institute (JRI) - to be held at the Sheraton Grande Tokyo Bay Hotel.
Nov. 13-14 France	LMS 2001 Conference for Physical and Virtual Prototyping, to be held at Hotel New York, Disneyland Paris, Paris, France. If you have any questions, contact Kirsten Cabergs, via e-mail (kirsten.cabergs@lms.be) or phone (+32-16-384-200).
April 22-24	ANSYS Users Conference & Exhibition 2002 - Pittsburgh Hilton, Pittsburgh, PA. For information visit: ANSYS, Inc.
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 Hyatt Regency Hotel & Conference Center

**Due to the amount of travel/classes/conferences being rescheduled
I have listed the websites of our FEA Information Participants
for you to visit for up to date information**

Headquarters	Company	website
USA	Livermore Software Technology	www.lstc.com
USA	Engineering Technology Associates	www.eta.com
UK	OASYS, Ltd	www.arup.com /dyna
Japan	The Japan Research Institute, Ltd	www.jri.co.jp
USA	ANSYS, Inc	www.ansys.com
USA	Hewlett Packard	www.hp.com
USA	SGI	www.sgi.com
USA	MSC Software	www.mscsoftware.com
Japan	Fujitsu Ltd.	www.fujitsu.com
USA	EASi Engineering	www.easiusa.com
Belgium	LMS, International	www.lmsintl.com
Sweden	Engineering Research AB	www.erab.se
USA	DYNAMAX	www.dynamax-inc.com
Korea	THEME Engineering	www.lsdyna.co.kr
Korea	Korean Simulation Technologies	www.kostech.co.kr
Australia	Leading Engineering Analysis Providers	www.leapaust.com.au
Canada	Metal Forming Analysis Corp.	www.mfac.com
France	Dynalis	www.dynalis.fr
India	GissEta	www.gisseta.com
Russia	Strela – Russia	www.ls-dynarussia.com

FEA Information Showcase

7th International LS-DYNA Users Conference May 19-21, 2002 Dearborn, Michigan, USA



Second Call for Papers

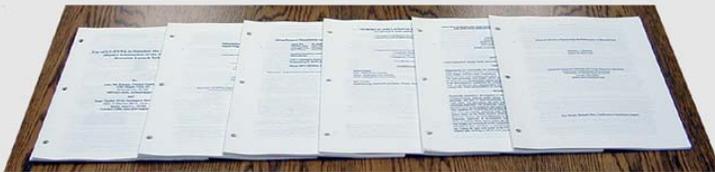
Abstract Deadline: December 19, 2001
Notification of Acceptance: January 19, 2002
Final Paper Deadline: March 19, 2002

Specification for Paper Submission:

Abstract Length: Approximately 300 words; please include figures if possible. Paper Length: Maximum of 3000 words, single-spaced, on 8 1/2" x 11" paper. Papers must be in MS Word format with imbedded graphics.

Please mail, fax or email your abstracts to:

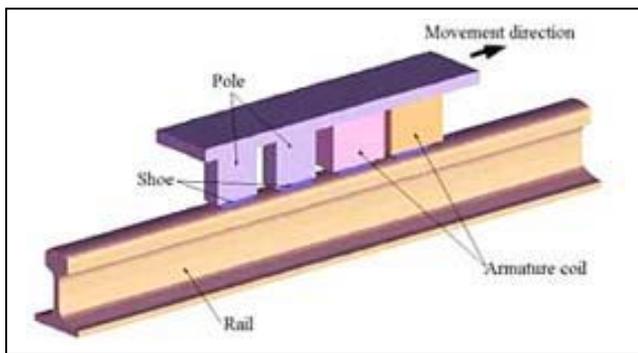
Ms. Pat Kulzer
Livermore Software Technology Corp.
7374 Las Positas Road
Livermore, CA 94550
Tel: 925-245-4546
Fax: 925-449-2507
Email: papers@lstc.com



The Japan Research Institute Limited
www.jri.co.jp/pro-eng/jmag/e/jmg/index.html

JMAG-Studio A magnetic field analysis program

This example shows how JMAG can be used for the analysis of an eddy current brake for railcars



eta / FEMB27-PC



No more text editing! FEMB27-PC (*Finite Element Model Builder*) is a complete pre/post-processing package designed from the ground up for LS-DYNA compatibility. FEMB-PC's new graphical interface supports all LS-DYNA keyword cards through version 950+.

New Features:

- Complete LS-DYNA Compatibility
- New Graphic User Interface
- Improved Model Handling Controls
- CAD Capabilities
- Robust IGES & VDA translator
- Load Curve Manipulation
- Presentation Ready (JPG, AVI, VRML)
- OpenGL Supported
- Windows 98/NT4/2000

eta
FEMB
27

FEMB

eta
Post
GL

PostGL

eta
Graph

Graph

eta/FEMB IS A FULL-FEATURE, FULL-FUNCTION GENERAL PURPOSE, PRE / POST PACKAGE FOR USE WITH LS-DYNA