FEA Information International News

www.feainformation.com

Issue 2001-02
Fifth Issue

Welcome to Issue 2001-2 of our newsletter. Our goal is to bring you a monthly synopsis of the additions/revisions to the FEA Information web.

When available, we'll bring you information from our participating companies, engineers, professors, consultants, and students of FEA Information. Feel free to have your associates send me an e-mail to be added to our FEA Information mailing list mv@feainformation.com

A LIST OF NEW CAPABILITIES IN VERSION 960 OF LS-DYNA

Most new capabilities work on both MPP and SMP computers; however, the capabilities that are implemented on the SMP version, which were not considered critical for this release, are flagged below. These SMP unique capabilities are being extended for MPP calculations and will be available in the near future in updates to the 960 releases.

Version 960 consists of a single source code, but separate executables, for MPP and SMP. The implementation of new capabilities in the MPP option requires the addition of message passing code that can require considerable manpower to write and debug. Due to the fact that older, more widely used SMP capabilities have higher priority for porting, we sometimes delay the MPP implementation of the newer capabilities.

Single and double precision versions of the UNIX release will be available. Double precision is necessary for implicit problems. On the PC platform a double precision version will be available as soon as possible.

ADAPTIVITY

- Adaptive mesh coarsening can be done before the implicit springback calculation in metal forming applications.
- Two-dimensional adaptivity can be activated in both implicit and explicit calculations. (SMP version only)
- Adaptive calculation can now be done on the MPP version.
- A three-dimensional adaptive option for solid elements is under development in version 960. This new development will be discussed in next month's newsletter.

AIRBAGS

- Airbag option "SIMPLE_PRESSURE_VOLUME" allows for the constant CN to be replaced by a load curve for initialization. Also, another load curve can be defined which allows CN to vary as a function of time during dynamic relaxation. After dynamic relaxation CN can be used as a fixed constant or load curve. This new option is currently used for tire inflation.
- Hybrid inflator model utilizing CHEMKIN and NIST databases is now available. Up to ten gases can be mixed.

CONSTITUTIVE MODELS

- Damage mechanics is available to smooth the post-failure reduction of the resultant forces in the constitutive model *MAT_SPOTWELD_DAMAGE.
- Finite elastic strain isotropic plasticity model is available for solid elements. *MAT_FINITE_ELASTIC_STRAIN_PLASTICITY.
- A shape memory alloy material is available: *MAT_SHAPE_MEMORY.
- Reference geometry for material, *MAT_MODIFIED_HONEYCOMB, can be set at arbitrary relative volumes or when the time step size reaches a limiting value. This option is now available for all element types including the fully integrated solid element.
- Non orthogonal material axes are available in the airbag fabric model. See *MAT_FABRIC.
- New constitutive models include, for the beam elements:
  - *MAT_MODIFIED_FORCE_LIMITED
  - *MAT_SEISMIC_BEAM

*MAT_CONCRETE_BEAM

for the discrete beam elements:

*MAT_ELASTIC_SPRING_DISCRETE_BEAM
*MAT_INELASTIC_SPRING_DISCRETE_BEAM
*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM
*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM

for the shell and solid elements:

*MAT_ELASTIC_VISCOPLASTIC_THERMAL

for the shell elements:

*MAT_GURSON
*MAT_GEPLASTIC_SRATE2000
*MAT_ELASTIC_VISCOPLASTIC_THERMAL
*MAT_COMPOSITE_LAYUP
*MAT_COMPOSITE_LAYUP
*MAT_COMPOSITE_DIRECT

for the solid elements:

*MAT_JOHNSON_HOLMQUIST_CERAMICS
*MAT_JOHNSON_HOLMQUIST_CONCRETE
*MAT_INV_HYPERBOLIC_SIN
*MAT_UNIFIED_CREEP
*MAT_SOIL_BRICK
*MAT_DRUCKER_PRAGER
*MAT_RC_SHEAR_WALL

and for all element options a very fast and efficient version of the Johnson-Cook plasticity model is available:

*MAT_SIMPLIFIED_JOHNSON_COOK

- A nonlocal failure theory is implemented for predicting failure in metallic materials. The keyword *MAT_NONLOCAL activates this option for a subset of elastoplastic constitutive models.
- A new option allows the use of deviatoric strain rates rather than total rates in material model 24 for the Cowper-Symonds rate model.

CONSTRAINTS

- Attachment nodes can be defined for rigid bodies. This option is useful for NVH applications.
- CONSTRAINED_POINTS tie any two points together. These points must lie on a shell element.
- CONSTAINED_INTERPOLATION option for beam to solid interfaces and for spreading the mass and loads. (SMP version only).
- Moving SPC's are optional in that the constraints are applied in a local system that rotates with the 3 defining nodes.
- Acceleration time histories can be prescribed for rigid bodies.
- Prescribed motion for nodal rigid bodies is now possible.

CONTACT

- Torsional forces can be carried through the deformable spot welds by using the contact type: *CONTACT_SPOTWELD_WITH_TORSION (SMP version only with a high priority for the MPP version if this option proves to be stable.)

- Tiebreak automatic contact is now available via the *CONTACT_AUTOMATIC_..._ TIEBREAK options. This option can be used for glued panels. (SMP only)
- *CONTACT_RIGID_SURFACE option is now available for modeling road surfaces (SMP version only).
- Interference fits can be modeled with the INTERFERENCE option in contact.
- Option to track initial penetrations has been added in the automatic SMP contact types rather than moving the nodes back to the surface. This option has been available in the MPP contact for some time. This input can be defined on the fourth card of the *CONTROL_CONTACT input and on each contact definition on the third optional card in the *CONTACT definitions.
- A new contact option allows the inclusion of all internal shell edges in contact type *CONTACT_GENERAL, type 26. Adding _INTERIOR after the GENERAL keyword activates this option.
- Soft constraint is available for edge-to-edge contact in type 26 contact.

DAMPING

- Relative damping between parts is available, see *DAMPING_RELATIVE (SMP only).
- Rayleigh damping input factor is now input as a fraction of critical damping, i.e. 0.10. The old method required the frequency of interest and could be highly unstable for large input values.

DATABASES

- The command "sw4." will write a state into the dynamic relaxation file, D3DRLF, during the dynamic relaxation phase if the D3DRLF file is requested in the input.
- Added mass by PART ID is written into the MATSUM file when mass scaling is used to maintain the time step size, (SMP version only).
- Upon termination due to a large mass increase during a mass scaled calculation, a print summary of 20 nodes with the maximum added mass is printed.
- If the average acceleration flag is active, the average acceleration for rigid body nodes is now written into the D3THDT and NODOUT files. In previous versions of LS-DYNA, the accelerations on rigid nodes were not averaged.
- Fixed rigid walls PLANAR and PLANAR_FINITE are represented in the binary output file by a single shell element.
- The interface force binary database now includes the distance from the contact surface for the FORMING contact options. This distance is given after the nodes are detected as possible contact candidates. (SMP version only).
- Interface frictional energy is optionally computed for heat generation and is output into the interface force file (SMP version only).
- The CADFEM option for ASCII databases is now the default. Their option includes more significant figures in the output files.
- When using deformable spot welds, the added mass for spot welds is now printed for the case where global mass scaling is activated. This output is in the log file, D3HSP file, and the MESSAG file.
- A database option has been added that allows the output of added mass for shell elements instead of the time step size.
- Initial penetration warnings for edge-to-edge contact are now written into the MESSAG file and the D3HSP file.

ELEMENTS

- A layered shell theory is implemented for several constitutive models including the composite models to more accurately represent the shear stiffness of laminated shells.
- A fully integrated version of the type 16 shell element is available for the resultant constitutive models.
- A discrete Kirchhoff triangular shell element (DKT) for explicit analysis with three in plane integration points is flagged as a type 17 shell element. This element has much better bending behavior than the C^0 triangular element.
- A discrete Kirchhoff linear triangular and quadrilateral shell element is available as a type 18 shell. This shell is for extracting normal modes and static analysis.
- A C^0 linear 4-node quadrilateral shell element is implemented as element type 20 with drilling stiffness for normal modes and static analysis.
- An assumed strain linear brick element is available for normal modes and static's.
- The fully integrated thick shell element has been extended for use in implicit calculations.
- A fully integrated thick shell element based on an assumed strain formulation is now available. This element uses a full 3D constitutive model, which includes the normal stress component and, therefore, does not use the plane stress assumption.
- The 4-node constant strain tetrahedron element has been extended for use in implicit calculations.
- Type 14 acoustic brick element is implemented. This element is a fully integrated version of type 8, the acoustic element (SMP version only).
- Preload forces are can be input for the discrete beam elements.
- Objective stress updates are implemented for the fully integrated brick shell element.
- Second order stress updates can be activated by PART ID instead of globally on the *CONTROL_ACCURACY input.
- Finite length discrete beams with various local axes options are now available for material types 66, 67, 68, 93, and 95. In this implementation the absolute value of SCOOR must be set to 2 or 3 in the *SECTION_BEAM input.
- A moving local coordinate system, CID, can be used to determine orientation of discrete beam elements.

INPUT

- Generalized set definitions, i.e., SET_SHELL_GENERAL etc. provide much flexibility in the set definitions.
- A capability to initialize the thickness and plastic strain in the crash model is available through the option *INCLUDE_STAMPED_PART, which takes the results from the LS-DYNA stamping simulation and maps the thickness and strain distribution onto the same part with a different mesh pattern.
- A capability to include finite element data from other models is available through the option, *INCLUDE_TRANSFORM. This option will take the model defined in an INCLUDE file: offset all ID's; translate, rotate, and scale the coordinates; and transform the constitutive constants to another set of units.

MISCELLANEOUS

- Incompressible flow solver is available. Structural coupling is not yet implemented.
- An internally generated smooth load curve for metal forming tool motion can be activated with the keyword: *DEFINE_CURVE_SMOOTH.
- A flooded surface option for acoustic applications is available (SMP version only).

- Eigenvalue analysis of models containing rigid bodies is now available using BCSLIB-EXT solvers from Boeing. (SMP version only).
- Modal superposition analysis can be performed after an eigenvalue analysis. Stress recovery is based on type 18 shell and brick (SMP only).
- Each compilation of LS-DYNA is given a unique version number.

Implicit Notes III
Dr. Bradley Maker
Livermore Software Technology

Topic E (Revision)

Equation solver options

The sparse ordering options were incorrect in the notes from the January issue. Below are the correct new input options for the *CONTROL_IMPLICIT_SOLVER keyword:

```
$
*CONTROL_IMPLICIT_SOLVER
$   lsolvr   prntflg    negeig     order
        0         0         0         0
$
$ lsolvr = linear equation solver
$   eq. 1: sparse direct auto out-of-core
$   eq. 2: sparse direct incore
$   eq. 3: sparse direct double precision
$   eq. 4: new sparse direct single - auto out-of-core (DEFAULT)
$   eq. 5: new sparse direct double - auto out-of-core
$   eq. 6: BCSLIB-EXT double - auto out-of-core
$   eq.11: iterative, Conjugate Gradient
$   eq.12: iterative, C.G. with Jacobi
$   eq.13: iterative, C.G. with Inc. Choleski
$   eq.14: iterative, Lanczos
$   eq.15: iterative, Lanczos with Jacobi
$   eq.16: iterative, Lanczos with Inc. Choleski
$
$ prntflg = print flag
$    eq.0: no printed information concerning solver
$    eq.1: summary information:  memory, cpu
$    eq.2: solver statistics
$    eq.3: debug level information and checking
$
$ order = sparse ordering option
$    eq.0: ordering chosen by LS-DYNA
$    eq.1: order with Multiple Minimum Degree
$    eq.2: order with METIS
$
```

Case Study: MAN Nutzfahrzeuge AG, Munich, Germany

ONE LESS REASON TO TURN OVER

Riding in overland busses is as safe as air or rail travel even in the event of bus overturning. Through structural optimization, MAN Heavy Trucks and Busses (Nutzfahrzeuge) AG insures that the passenger compartment provides sufficient survival space

The automotive industry was among the first to rely on finite element analysis in product development. Increasing demand for vehicle performance called for a reduction of costly and time-consuming tests with physical prototypes. The finite element method has now become a well-established tool in all areas of engineering. Probably the most spectacular among its many applications are digital vehicle crash tests. Passenger cars that leave the assembly lines in large numbers usually justify a substantial amount of crash tests. Comparatively small quantities and high cost of the individual vehicle, however, characterize bus manufacturing. Thus, only a small number of tests are economically feasible, ideal circumstances for applying numerical methods. Another compelling reason to use simulation techniques in crash testing is a law that allows buses to be certified based on numerical analysis only.

Certification via Numerical Simulation

Munich-based MAN Heavy Trucks and Buses (Nutzfahrzeuge) AG, a subsidiary of the MAN group, successfully works with the software package LS-DYNA which is marketed by CAD-FEM. Comparisons between simulation and test results of bus overturning convinced MAN’s research and development experts of LS-DYNA’s outstanding capabilities. Loading conditions and structural response measured during the tests agreed well with numerical predictions. Hence MAN is now using LS-DYNA for structural optimization and certification of its entire product line.

In order to limit efforts for structural modeling, it is imperative to optimize all CAE processes. This prompted the bus maker to work with the CAD program CATIA to generate the finite element mesh. This procedure has two key advantages: One is that the designers can participate in preparatory mesh generation tasks. The other is that the error prone transfer of solid structures to finite element preprocessors is no longer needed, since only nodes and elements must be transferred. Using existing interfaces, this procedure is relatively straightforward. The favorite finite element general-purpose software at MAN is ANSYS, which performs both static and dynamic strength or stiffness analyses. In the early stages of product development, such studies insure that vehicle operating performance and expected life span at minimum weight meet the demands. The idea behind comprehensive analyses with ANSYS is to reduce costly and time-consuming fatigue testing to one final check of the optimized structure. Crash analyses addressed in this article also start with ANSYS. Data transfer from ANSYS to LS-DYNA is through the CAD-FEM interface. All it takes to conduct crash simulation is some supplementary input of material and control parameters.

Slow Motion

Bus turnover is a comparatively slow dynamic process that leads to large computation time when analyzed with programs that use explicit equation solvers. Hence, the total duration of nearly 14 seconds for a bus turnover simulation had to be subdivided into several intervals. Using the certification ECE R66 turnover event as an example, the simulation is conducted as follows:

During the first one-second phase, the bus is lowered onto a platform. The vehicle body is considered a rigid body, only the power train is analyzed as an elastic structure allowing for the transition from the construction to the testing positions.

During the second phase (7.5 seconds), the platform rotates by 37 degrees. The entire model is now considered a rigid body such that the computational effort is minimal.

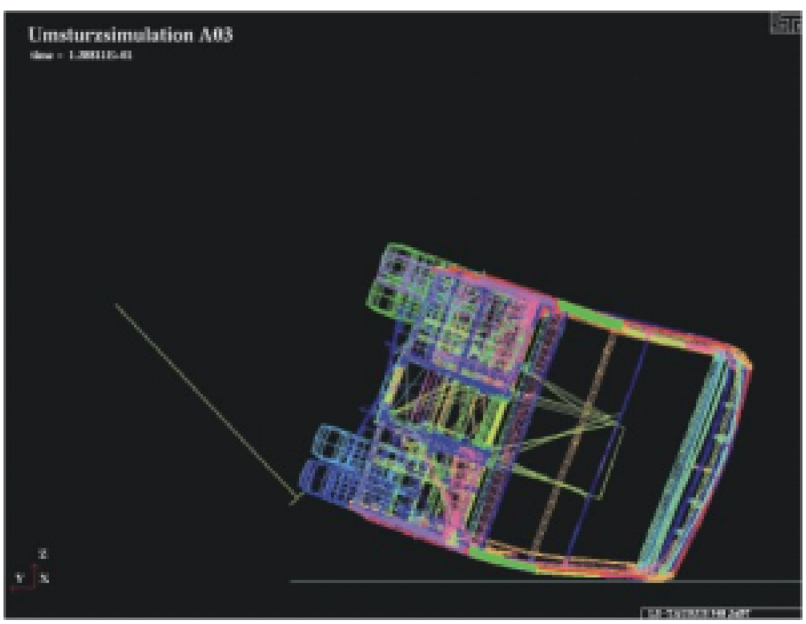
During the third phase of 5.2 seconds, the power train is modeled elastic, allowing an accurate representation of the vehicle body's overturning and swaying motion.

Ground impact of the structure launches the forth and final phase, which is the actual crash analysis. The model realistically accounts for the elasto-plastic, rate-dependent constitutive behavior of the structure. Modeling the complete overturning process pivots on LS-DYNA's special feature of "Rigid/Deformable Material Switching". Thanks to this option, it was not only feasible to model the complete overturning process but also achieve very good agreement between numerically predicted and measured response.

General information:

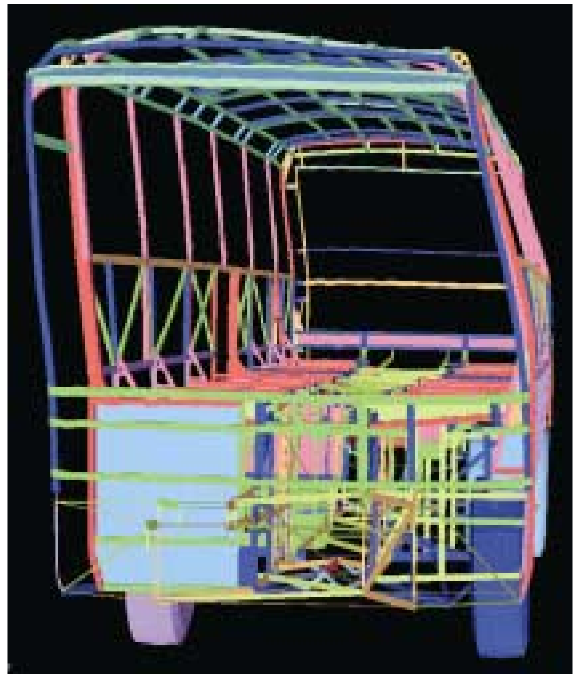
MAN Speeds up

When it comes to product development, MAN Heavy Trucks and Buses (Nutzfahrzeuge) AG obviously has a solid strategy. According to MAN's latest company report for the fiscal year 1998/99, it was able to push up sales of heavy trucks and buses by 12 per cent to 55,000 units. This boosted revenues by 9 percent to 4.93 billion (10^9) Euros. MAN also scored on a different front. Despite stiff competition, the company won the prestigious award "Bus of the Year 1999" for its new low floor bus.



Bus Ground Impact Launches Actual Crash Analysis

Experimental vs. Simulation Result



Case study is an excerpt of an article previously published in the CAD-FEM Infoplaner.

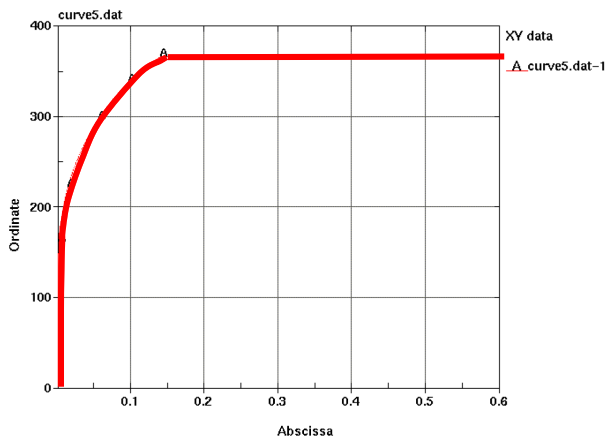
Motivation for Simulation Metal forming Process Notes #I:

- Common mistakes made by several different users
- Bad simulation results
- Better predictions could be obtained

Stress-Strain Curve (1)

- Observations: early failure (or necking) in sheet metal simulation
- Findings: the stress-strain curve is not reasonable (too low in the higher strain range)

Effective Stress



Effective Strain

Bad stress-strain curve

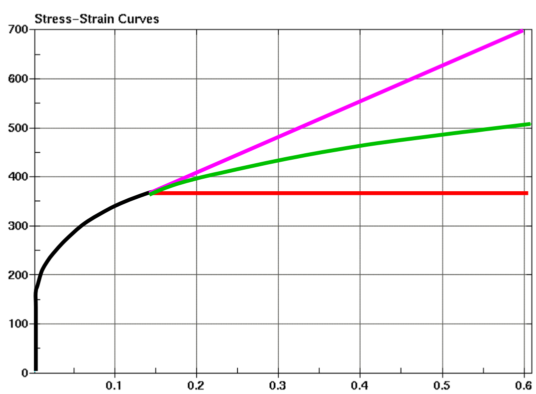
Stress-Strain Curve (2)

Cause:

- Stress-strain curve can only be obtained within a certain range from experiment, and extrapolation is needed.
- There are several ways to extrapolate

Effective

Stress



Constant slope (too high stress in large strain

Power Law (more reasonable fit)

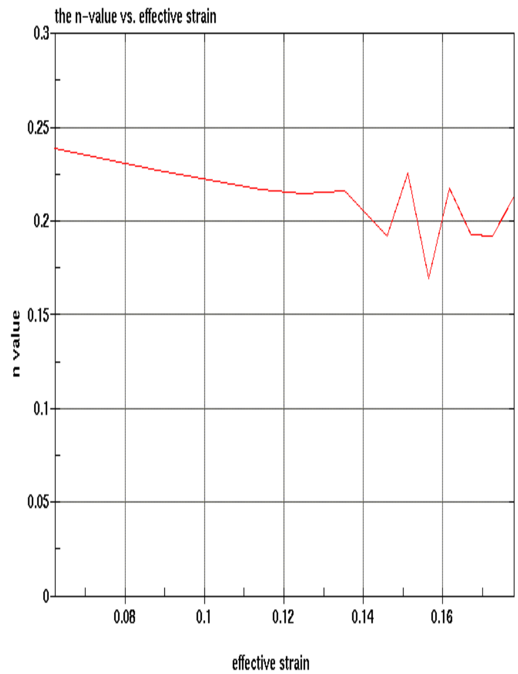
No hardening (result in earlier necking

Effective strain

Commonly extrapolated stress-strain curves

Stress-Strain Curve (3)

In using the power law to fit the curve, the k and n values need to be selected carefully (fitted from Numisheet'99 benchmark data).



Variations of n value with respect to strain

Stress-Strain Curve Conclusions

- There are some errors in the extrapolation of stress-strain data
- Power law is not very accurate, because the n and k values are not constant with respect to effective strain
- More studies need to be done.

Constraints in Spring Back Analysis

There are two ways to do spring back analysis:

- IMFLAG = O and use *INTERFACE_SPRINGBACK_SEAMLESS
- IMFLAG = 1

Advantages of the above two methods:

1st: It is easy to apply boundary constraints. We can simply put the necessary constraints under the keyword *interface…seamless
2nd: it can output 'dynain' file

Disadvantages of the two methods:

1st: it can not output 'dynain' file
2nd: extra keyword cards need to be used (*constrain…SPC) to put the constraint

Common mistakes:

- Using IMFLAG = 0 and seamless switch, no 'dynain'file can be obtained
- Using IMFLAG = 1, and putting the constraints under the keyword *INTERFACE_SPRINGBACK_DYNA3D, the constraints will not be applied.

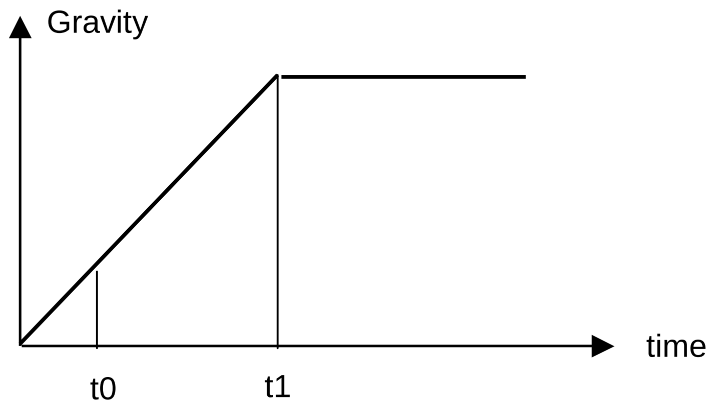
Accordingly, convergence will become a problem. (Constraints will only be output in *NODE section of dynain file)

Gravity in Spring Back Prediction

For big parts, it is important to consider the gravity effect in spring back predictions.

One possible mistake:

- Using a ramped curve for gravity, and the termination time is very small.



t0 – termination time in input deck

- Termination time should be large enough

Adaptive Constraints

There is a problem in using DYNAFORM for multiple-step stamping simulation.

- DYNAFORM reads the dynain file obtained from previous simulation, and generates another .blk file. However, blk file does not contain **'**Initial_stress_shell' and 'constrained_adaptivity' cards.
- Without '*initial_stress_shell", the residual stress and strain will not be considered in the next simulation.
- Without '*constrained_adaptivity', the adapted slave nodes will not be constrained by the master nodes, and then breach will happen

FEA Information Monthly Site Summary
Marsha Victory, FEA Information Co.

Monthly News Summary for January, 2001:

- FEA Information Co. welcomed SGI as a new participant.
- We revised the applications page with graphics.
- We started our additional News Page Sections.
- We revised our Featured Publication Page to showcase a publication for a month from the various sessions of the LS-DYNA User's Conference.
- In our Educational Forum we added a $50 payment to students for graphics.
- We added to our AVI Library an animated graphic #60a – Linear model of a heart valve (courtesy of our participant OASYS, LTD.
- January 15th we started our weekly showcase of software/hardware:
    1. January 15th: JRI - Magnetic Field Analysis With JMAG-Studio
    2. January 22nd: LSTC – LS-OPT
    3. January 29th: SGI – SGI Origin 3400 System

Publications Posted During the Month Of January

C2-1	Analytical Road Loads and Proving Ground Simulation in LS-DYNA Using Component Mode Synthesis – Dave Benson (University of California San Diego); John O. Hallquist (Livermore Software Technology Corporation)
C2-2c	Accurate and Detailed LS-DYNA FE Models of the US- and EUROSID: A Review of the German FAT Project – Ulrich Franz, Oliver Graf (CAD-FEM GmbH): Andreas Hirth, Matthias Walz (DaimlerChrysler AG)
C2-3c	Making FEM Tire Model and Applying It for Durability Simulation – Masaki Shiraishi, Hiroshi Yoshinaga, Naoaki Iwasaki (Sumitomo Rubber Industries, Ltd.)' Kimihiro Hayashi (Japan Research Institute)
D3-4c	Pedestrian Protection: Use of LS-DYNA to Influence Styling and Engineering – Bhavik R. Shah and Richard M. Stuart (Ove Arup & Partners): Aram Kasparian (Design Research Associated Ltd. – ARUP)

February 8, 2001 was FEA Information's one-year anniversary. We are also pleased to announce that FEA Information was listed as "Editor's Choice" on Netscape under the category search Finite Element Analysis.

We will continue to bring you information and develop each of our informational websites. Always feel free to send for posting your presentations, publications or information utilizing any of our participant's software/hardware.

Sincerely,

Marsha Victory
mv@feainformation.com

Participant Product News

Engineering Technology Associates Inc.
1133 E. Maple Rd., suite 200
Troy, MI 48083 USA
Tel: (248) 729-3010, Fax: (248) 729-3020

eta/DYNAFORM 3.2

A perfect Mesh from an imperfect surface DYNAFORM's latest release includes and enhanced mesher with the industry's most powerful corrective solution. Gaps and overlapping surfaces are automatically repaired. This release also offers residual stress and thinning mapping for structural analysis.

DYNAFORM is used to predict the formability of sheet metal products in terms of cracking, wrinkling, thinning, skidmark, and springback effects that occur during the stamping process.