*fea*INFORMATION
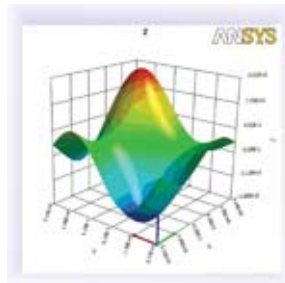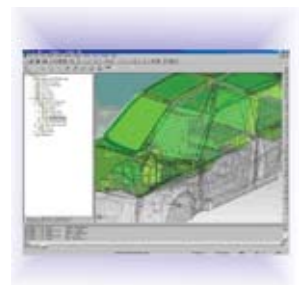
The 10th International
LS-DYNA® Users Conference
June 8-10, 2008
Dearborn, Michigan USA

ANSYS
DesignXplorer

ESI-Group
VAOne
Vibro-acoustic
Analysis and Design

FEA Information Announcements

LSTC's 10th Int'l LS-DYNA® Users Conference
> www.ls-dynaconferences.com

"H-III Dummy Model" mailing list
You will receive all LSTC's new H-III Dummy Releases and Updates.
> If you would like to be on the H-III Dummy mailing list contact Sarba
> sarba@lstc.com

LS-OPT Version 3.3 now available.
   Information is included in this month's news

Dr. Ian Do, of LSTC,
> Presenting LS-DYNA, Its Applications and Potentials
> The 19th Annual HP CAE Symposium.
> Invitation http://www2.lstc.com/pdf/HP_invite.pdf

> **One Day Free Training Seminar** - LSTC Troy, Michigan office.
> April 7th - LSTC and Intel will be offering a joint one day LS-DYNA® and
> INTEL® Cluster Ready Training Seminar. Please contact: wlm@lstc.com

> **One Day Free Symposium at the Meridien Hotel in Pune, India**.
> Oasys Ltd. and nHance Engineering Solutions Pvt. Ltd.
> invite you to find out about current and future LS-DYNA developments
> and FE dummy barrier models. Register by e-mail india.support@arup.com

Sincerely,
> Art Shapiro, Editor art@feainformation.com
> Marsha Victory – President – mv@feainformation.com
> Anthony Giaccana – Business Manager – agiac99@aol.com
> Wayne Mindle – Graphics – wlm@lstc.com
> Trent Eggleston – Consultant/Systems Admin - feaanswer@aol.com

Table of Contents —January 2008

LS-OPT Version 3.3

To get LS-OPT, type in or copy to your browser ftp://user:computer@ftp22.lstc.com

The following new features are available:

1. Radial Basis Function networks are available to create surrogate models. The building of these metamodels is significantly faster than Neural networks.

2. Multi-Objective optimization (MOO) for converging to the Pareto optimal front is supported as both a direct and metamodel-based solver.

3. Robust parameter (Taguchi) design is supported. The variation of a response can be used as an objective or a constraint in the optimization process.

4. Mapping of results to the FE mesh of the base design:
   a. the results are considered at fixed coordinates.
   b. These capabilities allow the viewing of metal-forming
   c. robustness measures in LS-PREPOST.

5. The ANSA morpher is supported as a preprocessor.

6. The truncated normal distribution is supported.

7. Extra input files can be provided for variable parsing.

8. A library-based user-defined metamodel is supported.

9. Interfacing to a Windows Compute Cluster Server (CCS)is supported.

10. Statistics can be calculated for checkpoints using an existing metamodel.

11. The 3-D Metamodel plotting including point plotting has been significantly enhanced.

12. Constraint plotting on 3-D metamodels is available with color-coded feasible/infeasible regions.

13. Regression statistics for metamodels
    a. (Mean, PRESS, R-squared) integrated with metamodel
    b. "Accuracy" plots.

14. The file viewing option (menu-bar) has been significantly
    a. enhanced. "lsopt_report" summary file refined and
    b. expanded to more types of analyses.

Platform availability: This first release is only for all Linux and Windows platforms. UNIX versions will be considered on request. Some UNIX versions are already available as beta versions while others are in preparation.

E N D   O F   R E L E A S E   N O T E S

LS-PrePost Update 2008

LS-PrePost is an advanced pre and post-processor that is delivered free with LS-DYNA. The user interface is designed to be both efficient and intuitive. LS-PrePost runs on Windows, Linux, and Unix utilizing OpenGL graphics to achieve fast rendering and XY plotting.

March 18th
    Updated the Trace Interface to make use of the General Selection panel

March 12th
    Updated the Toggle Menu with a more intuitive "check" system

February 29th
    Added coincident node pricking and made node/element picking significantly faster and easier.

January 23rd
    Added Split Solid option to the ElEdit Interface to allow splitting of hex and penta elements

January 1st
    Started working on LS-PrePost 2.3 Version 2.2 will be updated with bug fixes.

   New features will be added to 2.3

ANSYS DesignXplorer

The ANSYS DesignXplorer solution works from within the ANSYS Workbench interface to perform Design of Experiments (DOE) analyses of any ANSYS Workbench simulation, including those with CAD parameters. Although it requires more analyses to be performed and is typically slower than Variational Technology, which is now also a part of ANSYS DesignXplorer, DOE is not limited in the types of analyses that can be used with it. In fact, ANSYS DesignXplorer software can be used with ANSYS® Parametric Design LanguageTM (APDL)-based files to perform DOE on existing or new ANSYS analyses. ANSYS DesignXplorer allows you to perform optimization and Design for Six Sigma with any application or sequence of applications, including in-house codes, by using the third-party plug-in.

Design of Experiments

ANSYS DesignXplorer has a powerful suite of DOE tools. Automatic Design Points can be generated two ways, Central Composite Design (CCD) or Optimal Space-Filling. CCD provides a traditional DOE sampling set, while Optimal Space-Filling's objective is to gain the maximum insight with the fewest number of points. After sampling, ANSYS DesignXplorer provides four different meta-models to represent the simulations responses; Full Second-Order Polynomial, Kriging, Non-Parametric Regression and Neural Network. Kriging has two variants, pure Kriging and Radial Basis Function. These meta-models can accurately represent highly nonlinear responses. Once you have the simulation's responses characterized, ANSYS DesignXplorer supplies three different types of optimization algorithms: Screening (shifted Hammersley), Multi-Objective Genetic Algorithm (MOGA) and Nonlinear Sequential Quadractic Programming (NLPQL). ANSYS DesignXplorer has a full suite of sampling, modeling and optimization routines to address a wide variety of applications.
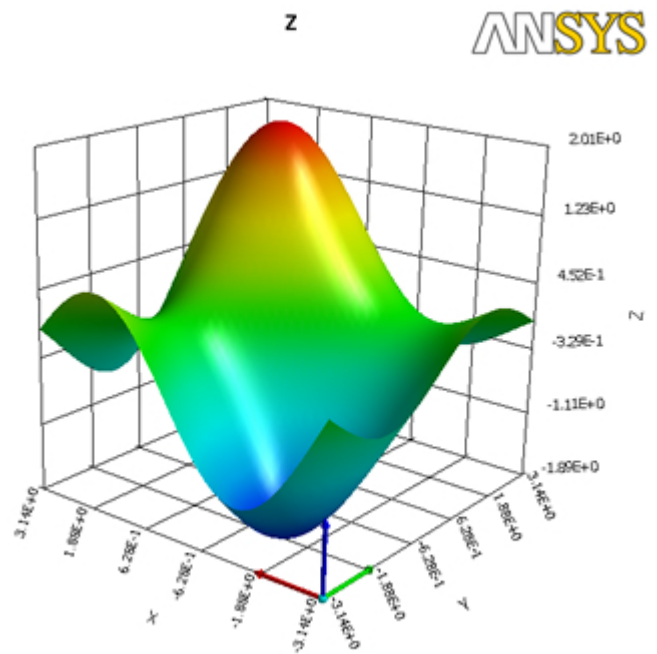
DesignXplorer's powerful sampling techniques and meta-models can capture complicated responses such as the one shown above.

Variational Technology

Look beyond traditional aspects of response simulation by using the Variational Technology method within ANSYS DesignXplorer that gives users a broader view of design concepts providing complete FEA results for every design point. Depending on the analysis

problem, the Variational Technology method can provide acceleration factors up to 100. With Variational Technology (VT), users can approach product design decisions much more efficiently.

Using the VT method to automatically calculate the entire design envelope within a single finite element solution, ANSYS DesignXplorer software allows users to perform quick and accurate what-if scenarios to periodically test design ideas. A traditional Design of Experiments approach requires many solutions to capture the behavioral changes due to parameter variations. For boolean parameters, the practical limit using traditional methods is about 10 boolean parameters, but the VT method handles up to 20 boolean parameters.

The Variational Technology method addresses many kinds of parameters:

- Geometric variations (CAD or ANSYS DesignModeler)
- Boolean variations (active or inactive status of spot welds, stiffening ribs, bolts, etc.)
- Element property variations (shell thickness, spring stiffness, point mass)
- Material variations (engineered plastics, composites, metals)
- Load variations (inertia and surface structural and thermal loads)

Design for Six Sigma (DFSS)

Design for Six Sigma (DFSS) is an analysis technique for assessing the effect of uncertain input parameters and assumptions on your model. A Design for Six Sigma analysis allows you to determine the extent to which

uncertainties in the model affect the results of an analysis. Based on a probabilistic characterization, Design for Six Sigma enables you to quantify the quality of your product addressing issues such as minimizing warranty costs and quantifying the reliability and robustness of the product. DesignXplorer VT supports DFSS by allowing you to define Uncertainty Variables, which are used in a probabilistic analysis.
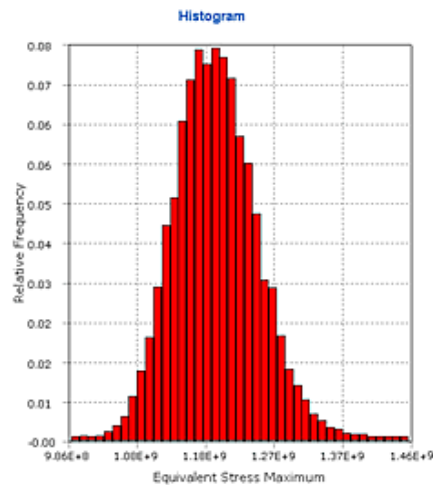
Histogram of the Maximum Equivalent Stress
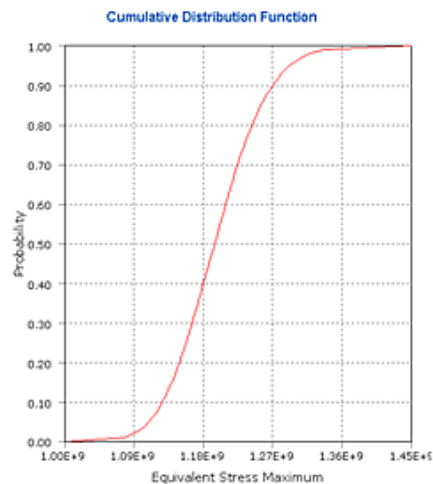
Cumulative Distribution Function of the Maximum Equivalent Stress

Probability	Equivalent Stress Maximum
0.0000693123	1.00097×10^9
0.0001	1.00487×10^9
0.0003	1.02404×10^9
0.001	1.04725×10^9
0.003	1.06112×10^9
0.01	1.07915×10^9
0.025	1.09389×10^9
0.05	1.10943×10^9
0.1	1.12681×10^9
0.3	1.16556×10^9
0.5	1.19462×10^9
0.7	1.22431×10^9
0.9	1.27044×10^9
0.95	1.29152×10^9
0.975	1.31328×10^9
0.99	1.33646×10^9
0.997	1.36758×10^9
0.999	1.39329×10^9
0.9997	1.41302×10^9
0.9999	1.44593×10^9
0.999931	1.44909×10^9

Probability Table showing the value Maximum Equivalent
stress at different Probability Values

The presentation format of results makes it easy to understand how design variation can effect system performance.

TOP CRUNCH Participant Submissions – March 2008

www.topcrunch.org

Vendor/ Submitter Org.	Computer/Interconnect	Processor	#Nodes x #Processors per Node x #Cores Per Processor = Total #CPU	Time (Sec)	Benchmark Problem	Submit Date 2008
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 128 x 2 = **256**	166	neon_refined _revised	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 64 x 2 = **128**	205	neon_refined _revised	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 32 x 2 = **64**	296	neon_refined _revised	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 256 x 2 = **512**	1070	3 Vehicle Collision	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 128 x 2 = **256**	1184	3 Vehicle Collision	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 64 x 2 = **128**	1684	3 Vehicle Collision	03/08/
SGI/Applications Engineering	Altix 4700/ NUMAlink	Intel Itanium DC Montvale 1.669Ghz	1 x 32 x 2 = **64**	2667	3 Vehicle Collision	03/08/
SGI/RDECOM-TARDEC HPC Center	Altix ICE 8200/Infiniband	Intel Xeon Core 2 Duo 3.00GHz	16 x 1 x 2 = **32**	7233	3 Vehicle Collision	03/19/
Self built/ University of Oulu	Lattice/ Gigabit Ethernet	Intel Core 2 duo E6850 3.0GHz	8 x 1 x 2 = **16**	1039	neon_refined _revised	03/20

8

INTERACTIVE MATERIAL MODEL DRIVER:
A LITTLE KNOWN LS-DYNA CAPABILITY

Len Schwer - Len@Schwer.net - January 2008

Abstract

This article provides a brief introduction to the LS-DYNA Material Model Driver (MMD), a utility for verifying the stress-strain response of material models. Some motivation as to why a MMD exists in LS-DYNA, and why users might find this utility useful, is provided. An illustration of results from the MMD, and a single element, stress-strain simulation are presented. Finally, a sample input file for the MMD is provided in an Appendix.

Scanning through the Appendices of the LS-DYNA User Manual reveals one appendix, i.e. Appendix K, that seems to be most overlooked. At a recent LS-DYNA User Conference I asked for a show of hands of those that knew there was a Material Model Driver (MMD) in LS-DYNA. In the room of about 50 users only 4 raised their hands, and I believe most of those had previously learned of the LS-DYNA MMD from me; I did not bother to ask how many had used the MMD.

Why is the LS-DYNA MMD known to so few, and used by even fewer? Probably the main reason is so few LS-DYNA users are truly interested in the material models they use. In my experience, material models are viewed as a 'necessary evil' by many analysts. The 'necessary' part should be obvious, a material model is a mandatory part of any LS-DYNA analysis, even rigid body dynamics. The 'evil' part has at least two components: (1) there are so many material models in LS-DYNA, which is the best for the application at hand, and (2) having selected from the long list of material models, where does the average user obtain the myriad of parameters required by most material models. The LS-DYNA MMD does not help the user select nor parameterize a material model, but it does offer the possibility of verifying how well the model parameters reproduce the laboratory data that describes the constitutive model response.

As stated in Appendix K,

> "The interactive material model driver in LS-DYNA allows calculation of the material constitutive response to a specific strain path. Since the constitutive model subroutines in LS-DYNA are directly called by this driver, the behavior of the constitutive models is precisely that which can be expected in actual (sic) applications."

The first sentence explains how the MMD works, i.e. the user specifies strains as a function of time[1], via *Define_Curves, and the MMD provides the corresponding stresses. The second sentence requires careful reading and bit of thinking about the implications of the phrase "… the constitutive model subroutines … are directly called by this driver, …" The implication of 'direct' is that no ancillary finite element software is involved when the MMD is exercised. What ancillary software? Well my list is not

---

[1] Time in this case can either be a pseudo-time that simply increments the strain, or physical time that defines the strain-rate appropriate for rate dependent materials.

complete, but it includes: nodes, elements, shape functions, time integrators, hourglass control, mass properties, numerous solution dependent settings, etc. None of these items are important to constitutive models, and hence the MMD.

$$\Delta\varepsilon_{ij}$$

$$\varepsilon_{ij}^{n},\sigma_{ij}^{n} \rightarrow \boxed{\textbf{Plasticity Algorithm}} \rightarrow \varepsilon_{ij}^{n+1},\sigma_{ij}^{n+1}$$
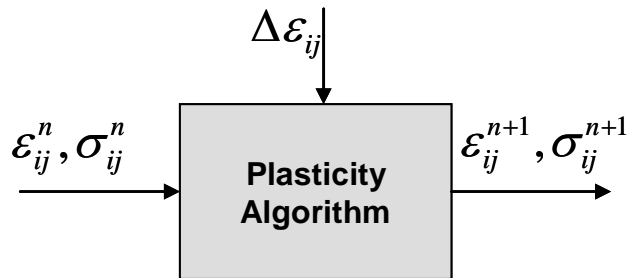
Figure 1 Idealization of a constitutive model with previous stress, strain and new strain increments as input and current stress and strain as outputs.

Figure 1 shows a cycle in the life of a constitutive model – constitutive models only care about a few things: strains, stresses, material parameters, and things called state variables, e.g. the current yield stress. A constitutive model beings its work before time starts, i.e. it is part of the initialization process that occurs before the first time increment is attempted. The constitutive model is sent material parameters and a flag from LS-DYNA instructing it to perform any initializations it may require. Most models only require setting a few state variables to their initial values, e.g. the current yield stress is set to the initial yield stress, and then they are ready to start their real work – converting increments in strains into new stresses for the next time step.

Where do these increments in strains, required by the constitutive models, come from? In a typical explicit analysis the strain increments are obtained at the element integration points by spatially differentiating the nodal displacements, which are interpolated over the element by shape functions. The nodal displacements come from the time integration of the nodal velocity, which in turn are obtained by time integration of the nodal accelerations. The accelerations are obtained from the equations of motion, which balance the external and internal forces. Those internal forces – they come from a spatial integration the stresses (generated by the hard working constitutive model) at the element integration points, and using the shape functions with a volume integration evaluated at the nodes.

Hopefully from the above 'life cycle' description you can see that constitutive models are at the heart of any analysis. What you should also see is that *external* to the constitutive model there are a lot of calculations: differentiation and integration of the shape functions, two time integrations, and solution of the equations of motion, that are not essential to the constitutive model's world.

Frankly, the constitutive model could careless where the strain increments come from, it just processes them to produce new stresses. Kind of the same way you do not care where the potatoes comes from, you just process the French fries. Since the constitutive model does not care where the strain increments come from, they can be supplied directly by the user – and thus is born a Material Model Driver.

OK, if a MMD accepts strains specified by the user, why don't more analysts use the MMD to check the material model parameters by replicating laboratory stress-strain data? The answers to this question include: (1) often analysts do not have the laboratory stress-strain data, e.g. they obtained the model parameters from a publication that did not include the data, and (2) when laboratory data is available, often it is insufficient for the purpose of using the MMD. The most common example of the latter is data from a uniaxial tensile stress test, which usually consists of the axial stress and the axial strain, sufficient to fit a von Mises model for metals, but not sufficient to use with a MMD. The missing parts of the strain specification are the lateral strains from the tension test. For metals, possibly the only test that reports all the required strains, and stresses, is a torsion test, where only the shear strain and stress are non-zero, but torsion tests on any material are rare.

For geomaterials, e.g. soils and concretes, the situation is usually reversed and almost all laboratory tests report all the non-zero stress and strain components. This is the case because these models are more complex than standard constitutive models for metals, and additional strain and stress data are needed to determine all the model parameters.

So how do analysts interested in verifying their constitutive model and parameters go about simulating laboratory results? Most often a single element is used along with prescribed boundary conditions to produce the desired stress-strain results. To compare such a single element approach to the MMD approach, consider as an example an aluminum specimen subjected to torsion with the experimental results reported as effective stress versus effective strain. The effective stress and strain are given by

$$\sigma_{eff} = \sqrt{\frac{3}{2} S_{ij} S_{ij}} = \sqrt{\frac{3}{2} \left(\sigma_{ij} - P\delta_{ij} \right) \left(\sigma_{ij} - P\delta_{ij} \right)}$$

$$\varepsilon_{eff} = \sqrt{\frac{2}{3} \varepsilon_{ij} \varepsilon_{ij}}$$

where the mean stress (pressure) is given by $P = \sigma_{kk}/3$. For the case of pure torsion, where the only non-zero strain and stress are ε_{12} and σ_{12}, the effect stress and strain are given by

$$\sigma_{eff} = \sigma_{12}$$

$$\varepsilon_{eff} = \frac{2}{\sqrt{3}} \varepsilon_{12} = \frac{1}{\sqrt{3}} \gamma_{12}$$

By comparison, for uniaxial tension where σ_{11} is the only non-zero stress and all the shear strains are zero, the effective stress and strain are given by

$$\sigma_{eff} = \sigma_{11}$$

$$\varepsilon_{eff} = \sqrt{\frac{2}{3} \left(\varepsilon_{11}^2 + \varepsilon_{22}^2 + \varepsilon_{33}^2 \right)}$$

Aside: for metal plasticity, where the incompressibility assumption is made, i.e. $\varepsilon_{33}^{p} = \varepsilon_{22}^{p} = -\nu\varepsilon_{11}^{p} = -0.5\varepsilon_{11}^{p}$, the above effective plastic strain has the more familiar form $\varepsilon_{eff} = \varepsilon_{11}^{p}$. This is why the results from a uniaxial stress test are sufficient to fit a von Mises model.

For the material model driver, the user specifies ε_{12} and allows LS-DYNA to compute the stress, effective stress and effective strain.

For the single element, the user specifies a displacement history with appropriate boundary conditions to ensure uniaxial stress, and allows LS-DYNA to compute the strain, stress, effective stress and effective strain. Displacement, or velocity, should be prescribed rather than force as this provides uniform increments in strain which are needed to closely track stress-strain curves with limited strain hardening.
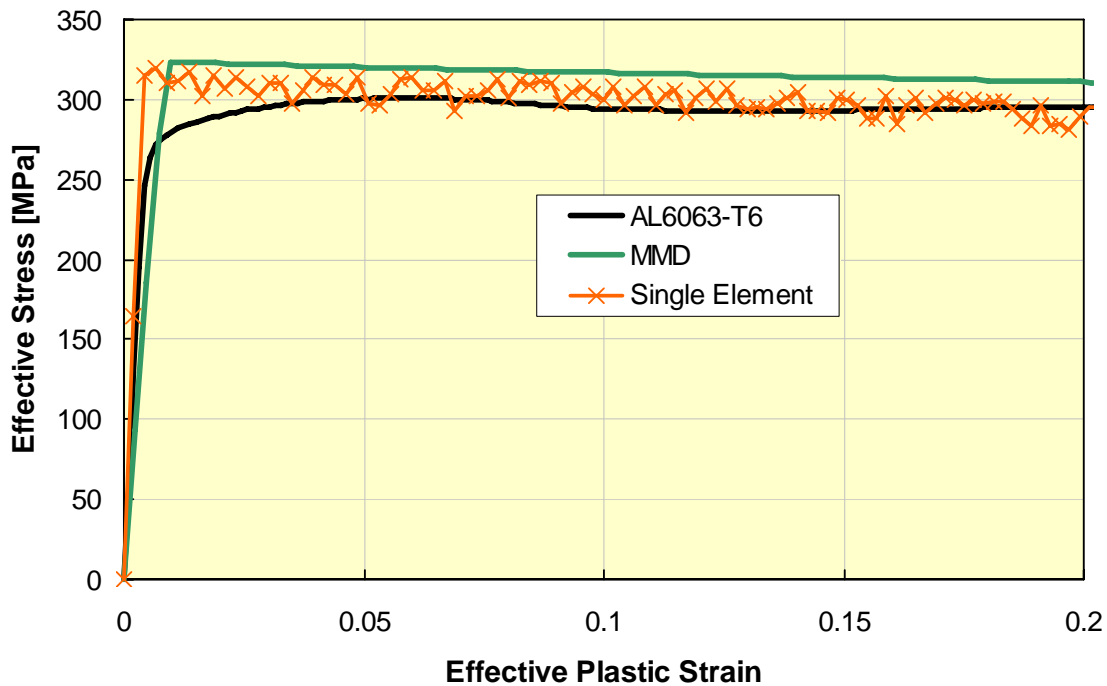
Figure 2 Comparison of effective stress-strain response for Aluminum 6063-T6 laboratory results with MMD pure shear simulation and single element uniaxial stress simulations results.

Figure 2 shows a comparison of the effective stress-strain response for Aluminum 6063-T6 from a laboratory pure shear test, the corresponding MMD pure shear simulation, and the single element uniaxial stress simulation result. Both the MMD and single element simulations used the same Johnson-Cook material model parameters. The effective strain loading rate was 1/second.

The reason for showing this comparison is not to indicate the quality of the model fit to the data, but to illustrate how ancillary items can affect the stress-strain response. Note the stress produced by the MMD is a smooth function of the strain. On the other hand,

the stress from the single element has an oscillatory (noisy) response. The reason for this noisy response is not known, but a good guess is the size selected for the single element has a lateral resonance excited by the axial prescribed motion. This illustrates that option choices, or errors, in the ancillary software will affect the stress-strain interpretation of the single element results.

The Material Model Driver available in LS-DYNA is limited in that it only accepts strains as input. Consider again the uniaxial stress test, as mentioned above the lateral strains are seldom reported, rendering the MMD essentially useless for replicating this test. However, if the MMD includes the option to prescribing either strain or stress components, then the uniaxial stress test could be simulated, as the lateral stress could be prescribed as zero.

Recently, Yvonne Murray of APTEK, has developed a MMD that can accommodate a mixture of prescribed strains or stresses, called a Mixed Mode Constitutive Driver (MMCD); see the article by Yvonne in the August 2007 issue of FEA Newsletter, or visit www.APTEK.com.

In addition to just being a generalization the present LS-DYNA MMD, the MMCD is also integrated with LS-OPT. Why link a MMD with LS-OPT? The combination of a general MMD with LS-OPT allows the user to provide stress-strain laboratory data, from many types of laboratory tests, and use LS-OPT to determine the constitute model parameters that provide the best fit to the laboratory data.

The next time you are considering fitting a material model to laboratory data, or have material model parameters and want to investigate the corresponding stress-strain response, consider a visit to Appendix K and the Material Model Driver.

Appendix – Sample MMD Input

The required keywords are:

- *CONTROL_TERMINATION – end time for MMD,
- *DATABASE_BINARY_D3PLOT –time increment or load step,
- *MAT_JOHNSON_COOK – any material model & EOS parameter definitions,
- *PART – part definition that links the material model and element type,
- *SECTION – element type (solid/shell),
- *DEFINE_CURVE – nine curves defining the nine components of the strain tensor as function of time or load step.

```
*KEYWORD
$
*TITLE
 Pure Shear Test - Material Model Driver
$
$=======1=========2=========3=========4=========5=========6=========7========8
$                                PARAMETER CARDS
$=======1=========2=========3=========4=========5=========6=========7========8
*Parameter
 r PlotOut  120.0e-2 r ASCIOut   25.0e-0  r TimEnd      120.0   r SRate   1.00e-3
$
*Parameter_Expression
  r SHStrn    0.5*TimEnd*sqrt(3)*SRate
  r TEndLC    TimEnd+ASCIOut
$
```

```
$ ........................................................................
$
$            E:\LS-DYNA\Version971\LS971-D-R4-40625.exe
$               i=Mat015-VP0-T300-1E0-ALL-MMD.k
$
$  units are
$                     length -> millimeters  = 10^-3 m
$                     time   -> milliseconds = 10^-3 s
$                     mass   -> gram         = 10^-3 kg
$                     force  -> m*a = g*(mm/ms^2)
$                             = kg-m/s^2 = N
$                     stress -> F/A = N/mm^2 = 10^6 N/m^2 = MPa
$ ........................................................................
$
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                              CONTROL CARDS
$=======1=========2=========3=========4=========5=========6=========7=========8
$
*CONTROL_TERMINATION
&TimEnd,0,0.000E+00,0.000E+00,0.000E+00
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                              DATABASE CARDS
$=======1=========2=========3=========4=========5=========6=========7=========8
*DATABASE_BINARY_D3PLOT
&PlotOut,0
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                              MATERIAL CARDS
$=======1=========2=========3=========4=========5=========6=========7=========8
$
$
$ DEFINITION OF MATERIAL    15                                    Johnson-Cook
$                                  6063-T6
*MAT_JOHNSON_COOK
$ MID    RO        G        E*    PR*  DTF*  VP  RATEOPT    (* shells only)
  15, 2.70E-3,  25.8E3,   0.0, 0.0,  0.0,   0.0, 0.0
$  A       B       N        C       M     TM      TR      EPS0
$ 293.1,  0.00,  1.00,  8.1E-3,  1.5937, 888.7, 293.0, 2.00E-7
  293.1,  0.00,  1.00,  0.0124,  0.8313, 888.7, 293.0, 2.00E-7
$  Cp     PC    SPALL   IT    D1      D2       D3        D4
  900.0, -1.0e6, 1.0,  1.0,  0.000,  0.000,   -0.000, 0.000E-3
$ 900.0, -1.0e6, 1.0,  1.0,  0.2631, 1.2423, -1.899, -2.5245
$  D5     C2
  0.00, 0.000E-4
$ 1.9567, 0.000E-4
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                               EOS CARDS
$=======1=========2=========3=========4=========5=========6=========7=========8
$
*EOS_Linear_Polynomial
$ EOSID  C0    C1        C2   C3   C4   C5  C6
   11 , 0.0, 67.28e3, 0.0, 0.0, 0.0, 0.0, 0.0
$ e0     v0
0.0, 0.0
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                              Section Cards
$=======1=========2=========3=========4=========5=========6=========7=========8
*SECTION_SOLID
$ SID  ElForm   ATE
  1,     1,      0
$=======1=========2=========3=========4=========5=========6=========7=========8
$                              Part Cards
```

```
$=======1=========2=========3=========4=========5=========6=========7=========8
*PART
 Pure Shear Test - Material Model Driver
$ PID  SECID MID  EOSID HGID GRAV ADAPT  TMID
  1,    1,    15,   11,  0,    0
$
$=======1=========2=========3=========4=========5=========6=========7=========8
$                                 DEFINE CURVE CARDS
$=======1=========2=========3=========4=========5=========6=========7=========8
$
$ du/dx=eps_xx
*DEFINE_CURVE
1,0,1.,1.0,0.,0.
0,0
&TimEnd, 0.0
&TEndLC, 0.0
$
$ dv/dy=eps_yy
*DEFINE_CURVE
2,0,1.,1.0,0.,0.
0,0
&TimEnd, 0.0
&TEndLC, 0.0
$
$ dw/dZ=eps_zz
*DEFINE_CURVE
3,0,1.,1.0,0.,0.
0,0
&TimEnd, 0.0
&TEndLC, 0.0
$
$ du/dy=eps_xy
*DEFINE_CURVE
4,0,1.,1.,0.,0.
0.0,0.0
&TimEnd, &SHStrn
&TEndLC, &SHStrn
$
$ dv/dX=eps_yx
*DEFINE_CURVE
5,0,1.,1.,0.,0.
0,0
&TimEnd, &SHStrn
&TEndLC, &SHStrn
$
$ du/dZ=eps_xz
*DEFINE_CURVE
6,0,1.,1.,0.,0.
  0.00, 0.000
&TimEnd, 0.0
&TEndLC, 0.0
$
$ dw/dX=eps_zx
*DEFINE_CURVE
7,0,1.,1.,0.,0.
  0.00, 0.000
&TimEnd, 0.0
&TEndLC, 0.0
$
$ dv/dZ=eps_yz
*DEFINE_CURVE
8,0,1.,1.,0.,0.
  0.00, 0.000
&TimEnd, 0.0
&TEndLC, 0.0
$
```

```
$ dw/dY=eps_zy
*DEFINE_CURVE
9,0,1.,1.,0.,0.
  0.00, 0.000
&TimEnd, 0.0
&TEndLC, 0.0
$
$
*END
```

Appendix – Sample PC Version Command Window

The PC version of the MMD does not include the interactive graphics feature. Users can still use the interactive MMD, but need to write the information to files for subsequent plotting with other applications. The following is an example of the input required to write time histories of two stress-strain components to files, and their cross plot, i.e. with mutual time parameter removed. The lines indicated in RED write components 9 and 109, i.e. effective stress and effective strain, respectively, to files labeled comp9 and comp109 respectively, and then the file, x9109, for cross plotting effective stress versus effective strain.

initialization completed

**.print
.enter filename for data output:comp9
.time 9
.file comp109
.time 109
.file x9109
.cross 9 109
.quit**
    101 t 1.2120E+02 dt 1.00E+06 write d3plot file

N o r m a l   t e r m i n a t i o n

March Featured Paper

Simulation of Masonry Wall Failure and Debris Scatter

Authors:
  Stuart C. McCallum, BAE Systems – ATC (Filton
Paul M. Locking, BAE Systems – Land (Shrivenham)
Steve R. Harkness, BAE Systems – Land (Shrivenham)

ABSTRACT:

This paper outlines a methodology for the simulation of masonry wall failure and debris scatter. The aim of this work is to develop simulation techniques which can be used to assess and improve the design of the building structures subject to high explosive loading. Masonry walls are constructed from bricks that are modeled as individual parts with tiebreak and single surface contact types.

A key requirement of this work is to accurately predict the final landing position and scatter pattern of any bricks. If the acceleration of a separated is significantly high, the trajectory and final landing position of the brick will be influenced by air-drag. In this work we simulate the air drag force using a user FORTRAN subroutine and demonstrate its accuracy with comparison to theory.

The strength of the mortar-masonry brick bond is validated by comparison with laboratory experiments conducted in previous work showing close agreement. A series of simulations are then presented which demonstrate the failure and debris scatter of a simplified building structure.

KEYWORDS: Blast modeling, Masonry bricks, Tiebreak contact, Single-surface contact, Drag

MD Nastran R2, 2.1 Summary

MD Nastran R2 and R2.1 bring powerful new analysis features and enhancements in the areas of contact; nonlinearity; noise, vibration, and harshness (NVH); acoustics; automotive powertrain; crash and occupant safety; high performance computing; assembly modeling; optimization; rotor dynamics; and aeroelasticity. The combined enhancements in MD Nastran R2 and R2.1 are summarized as follows.

Contact in Linear and Nonlinear Solutions

MD Nastran introduces contact in linear and nonlinear solutions, with the ability to easily convert from linear contact in SOL 101 to nonlinear contact in SOL 400.

- Nonlinear Contact in SOL 400. Provides general multibody contact under full nonlinear conditions of large contact motion, nonlinear materials, large rotation, and large strain. SOL 400 features deformable-deformable contact, deformable-rigid contact, and contact with friction.
- Linear Contact in SOL 101. Provides an algorithm similar to SOL 400 for use in SOL 101 linear analysis. A SOL 101 model with linear contact can be converted to simulate nonlinear behavior in SOL 400 without having to redefine contact.
- Permanent Glued Contact in Linear Solution Sequences. Provides a special type of contact model to be used when the contacting surfaces have no relative normal or tangential motion. Permanent glued contact is available in most of the linear solutions (except SOL 200), and provides a convenient way to join dissimilar meshes. Glued, linear, and nonlinear contact make use of the same user interface.
- (New in R2.1) Simplified Contact Definition. A new ALLBODY contact option saves significant time in preparing the input definition.
- (New in R2.1) Improved Performance. Contact performance has been greatly improved, particularly for large models.
- (New in R2.1) (Prerelease) Glued Contact in Optimization. A prerelease version of glued contact is available in SOL 200.

Advanced Integrated Nonlinear Analysis

SOL 400 has been extensively enhanced for multistep nonlinear analysis of large strain and material behavior, with or without contact.

- Nonlinear Contact Modeling. A new contact algorithm has been implemented in SOL 400 to perform general multibody contact for large contact motion, nonlinear materials, large rotations, and large strains. It is available in both nonlinear static and nonlinear transient analysis.
- Nonlinear Elements and Materials. Enhancements include orthotropic material properties for three-dimensional and plane strain

18

behavior, nonlinear gasket material properties for compression behavior, elasto-plastic material properties for use in large deformation analysis, and failure model properties for linear elastic material. Where appropriate, most elements in MD Nastran have been extended to include the effects of finite strain. Also, layered axisymmetric, shell, and solid composite elements are now available. The CQUADR and CTRIAR elements have been extended to nonlinear analysis.

- Nonlinear Procedures. New analysis chaining procedures have been added to conveniently define multiple load steps, run multiple independent cases, and specify multiple and mixed types of analyses in one job. For example, the user can pretension a bolt structure as a first load step in a multistep nonlinear analysis. Adaptive stepping schemes have been introduced that automatically modify load increment or time step size to accelerate convergence.

- Crack and Delamination. A new virtual crack closure technique (VCCT) calculates energy release rates for sharp cracks. In addition, a new library of interface elements (cohesive zone modeling) can be used to simulate the onset and progress of delamination.

- Kinematic Elements. New rigid elements with Lagrange formulations are now available for use in geometric nonlinear analysis involving large rotations. These elements have been implemented in both nonlinear static and nonlinear transient analysis.

- (New in R2.1) Differential Stiffness. Improved nonlinear differential stiffness with expanded options to

selectively turn off differential stiffness (NLDIFF) for improved convergence is now available.

- (New in R2.1) Relative Boundary Conditions. A new relative enforced displacement capability (see the SPCR Bulk Data entry) is now available; this is particularly useful for bolt modeling.

Implicit Nonlinear Analysis (SOL 600)

- Heat Transfer. Advanced thermal analysis is now available in SOL 600, including an efficient hemi-cube view factor calculation method, and automated procedures for thermal stress analysis based upon a prior heat transfer simulation. In addition, thermal analysis of composites includes calculation of the thermal gradient through the thickness.

- Modeling Enhancements. Connector technology has been enhanced to include large deformation formulations of the CFAST, CWELD, and CBUSH elements. Enhancements for facture mechanics include the calculation of stress intensity factors using the VCCT or Lorenzi methods, and the prediction of delamination.

- Performance Improvements. A new streaming input option is available that eliminates transfer files. Streaming input will be automatically turned off for analysis scenarios that are not supported. The analysis of composite shells has been improved such that assembly time has been reduced-often by a factor of ten-and memory requirements have been substantially reduced as well. Other performance improvements include an iterative solution that

runs out-of-core, and direct PLOAD4 support.

- (New in R2.1) Glued Contact. Permanent glued contact is now supported for conveniently joining dissimilar meshes.
- (New in R2.1) Solver Options. A mixed derivative solver has been added as an option.

Explicit Nonlinear Analysis (SOL 700)

Several new SOL 700 analysis capabilities are introduced in this release.

- Airbags and Occupant Safety. Fluid structure interaction (FSI) is now available in SOL 700 to simulate the complex behavior of multicompartment airbags and their interaction with anthropomorphic test devices (ATDs) during crash scenarios. Airbag inflation is based on full gas dynamics and finite volume (Eulerian) technology with adaptive meshing capability. The airbag surface acts as a coupling surface which adapts itself and follows the expanding Eulerian elements as the gas jet flows inside the bag. In addition to full gas dynamics, SOL 700 also supports the conventional uniform pressure method to simulate airbag behavior.
- Time Domain Noise, Vibration, and Harshness (NVH). This technique is based on the fast Fourier transform (FFT) method to compute frequencies and mode shapes of highly nonlinear and dynamic systems such as a vehicle running over a rough road. The advantage of time domain NVH is its ability to capture the nonlinear effects of contact, spring, bushing, tire, and suspension behavior.

- Prestress. Prestressing is now performed by a double precision version of the implicit solver, and the results are used as the prestate for explicit simulations such as bird strike and blade-out applications.
- Nastran Native Output. This release produces native MD Nastran output files for seamlessly postproces sing explicit events with MD Patran.
- New Material and Element Formulations. Thirteen new material models and five element formulations are added in SOL 700.
- Contact Features. SOL 700 now supports many new contact capabilities such as glued contact, tied nodes to surface, tied shell edge to surface, tied surface to surface, spot weld contact, single edge contact, and force transducers.

NVH and Acoustics

- Frequency Response Functions (FRFs) and FRF Based Assembly (FBA). A powerful new capability is available in MD Nastran R2 for frequency response analysis based on representations of components by their FRFs, and on assemblies of such components by the FBA process. FRF-based assemblies provide an effective means for revealing how excitations on one component affect responses at other components in the assembly.
- Exterior Acoustics. MD Nastran R2 is the production release for the exterior acoustics capability. It extends acoustics applications to unbounded regions such as radiated acoustics from an engine. A new sparse solver is available for the efficient solution of the unsymmetric matrices

encountered in exterior acoustics frequency response analysis.

Numerical Enhancements

- Sparse Solvers. Two new sparse solvers have been introduced: TAUCS (statics) and UMFPACK (unsymmetric). The UMFPACK solver provides scalable performance for exterior acoustics. In addition, the Lanczos method has been enhanced to take advantage of available memory. Automatic optimal reordering selection has been implemented for solid models to eliminate requiring the user to set flags.
- Iterative Solvers. Restrictions to the CASI iterative solver have been relaxed for statics of large solid models (e.g., engines). This includes an expanded list of supported element types.
- ACMS. Automated component modal synthesis (ACMS) has been extended to external superelements to provide significant reductions in computational time, I/O, and scratch space. One typical case study demonstrated improvement by an order of magnitude.
- Other HPC Enhancements. MD Nastran R2 has been ported to the Microsoft Compute Cluster. The compute kernels for x86_64 platforms have been optimized for both Intel and AMD-based systems. Improved user diagnostics provide a pivot ratio bar chart to help localize model singularities.
- (New in R2.1) Iterative Solver for Nonlinear Analysis. An iterative solver option is now available for nonlinear contact analysis of large solid models, as well as linear contact. Up to five-fold increases in speed have been achieved.

- (New in R2.1) Memory Management. Users can now control the amount of memory available for the UMFPACK solver to ensure adequate memory at runtime.
- (New in R2.1) (Prerelease) Sparse solver information output (MDTSTATS). Matrix diagonal term statistical output is available via the new MDTSTATS feature for use in determining model quality.

Elements and Connectors

- Connectors. A new seam weld connector element (CSEAM) is now available for assembly modeling. It features extended capabilities for connecting higher order elements; mesh-independent connections to top/bottom shell patches defined either by property IDs or element IDs; tailored parts connection; and support for anisotropic material properties. For spot weld elements (CWELD, CFAST), end point displacement output can now be obtained to view the relationship between the spot weld and the connecting shells. A new connector type, RBE2GS, is introduced to optionally search and connect independent grid points of the two closest RBE2 elements within a specified search radius.
- Composite Beam Using Variable Asymptotic Method (VAM). The arbitrary beam cross section capability has been enhanced to support composite materials. A new three-noded composite beam element (see the CBEAM3 Bulk Data entry) represents the cross-section and layup of plies, and accounts for interaction among plies. It provides an efficient alternative to conventional 3-D

modeling for beam-like composite structures such as rotor blades and shell stiffeners.

Optimization

- Topology Optimization. This release provides combined simultaneous topology, sizing, and shape optimization to find potentially better designs. Different mass targets can now be applied on multiple design parts of the structure. Symmetry constraints have been extended to cyclical applications such as car wheels. An adjoint design sensitivity analysis method has been implemented for inertia relief sizing optimization. Significant performance enhancement with minimum member size control has been achieved, particularly for large numbers of sizing design variables.
- Automatic External Superelement Optimization (AESO). This new feature automatically partitions the model into designed and nondesigned (external superelement) parts for efficient optimization. An order of magnitude increase in speed can be achieved without requiring user knowledge of superelements.
- Randomization (Prerelease). This randomization capability provides a way to stochastically introduce uncertainty into a model, such as tolerances in connectivity, properties, and loads. The user
-

selects the outputs to monitor. This beta capability is a first step in developing a multirun environment to spawn multiple jobs, collect the results, and perform statistical postprocessing.

Rotor Dynamics and Aeroelasticity

- Rotor Dynamics. Unbalanced loading can now be used for frequency response analysis with the rotor dynamics option. Frequency response case control can be used directly in SOL 146, since multiple RGYRO subcases can now be used in rotor dynamics. The specification of damping has been simplified, and now allows for new damping formulations such as hybrid damping. In addition, as a prerelease capability, the effects of rotor stiffness, mass, and damping can be included in SOL 200 optimization.
- Aeroelasticity. Monitoring points can now be updated and summed. A new type of monitoring point (MONCNCM) has been introduced for monitoring stripwise aerodynamic results such as lift and pitching moments. Other enhancements include various splining techniques for aerodynamic structural applications.
- (New in R2.1) Spline Application Programming Interfaces (APIs). Support has been added for user-defined splines (SPLINEX) via an API.

VA One : The ONE simulation environment for vibro-acoustic analysis and design

http://www.esi-group.com/products/vibro-acoustics/va-one

VA One is a complete solution for simulating the response of vibro-acoustic systems across the full frequency range. VA One seamlessly combines Statistical Energy Analysis (SEA), Finite Elements and Boundary Elements in ONE model. It is the only simulation code on the market today that contains the complete spectrum of vibro-acoustic analysis methods within ONE common environment.
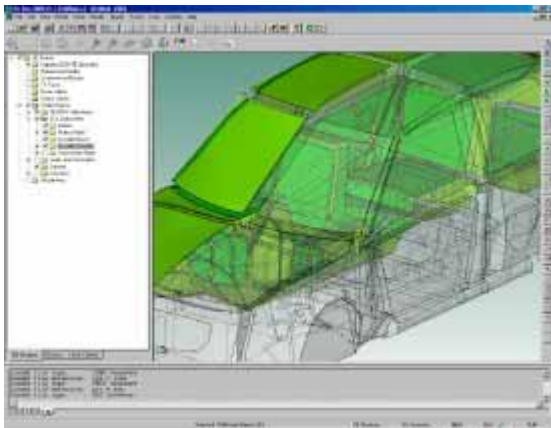
The core functionality of VA One is contained in 6 main modules:
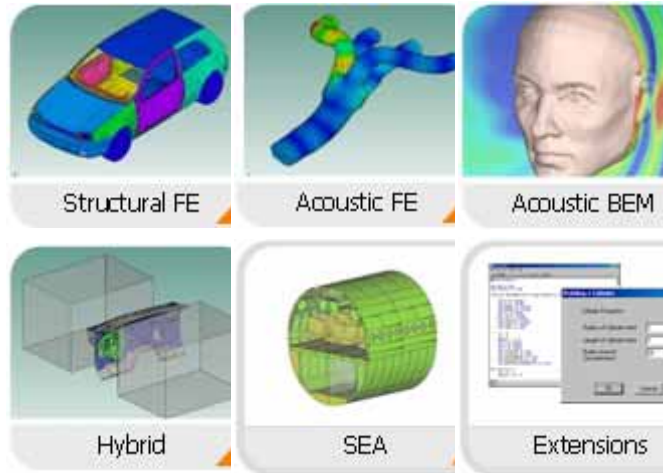
Whether you are simulating the vibro-acoustic response of an automotive firewall or creating a system level model of an entire launch vehicle, VA One has the flexibility to meet your analysis needs!

Yahoo Group Yammerings

Jim Kennedy
KBS2 Inc.
jmk@kbs2.com

Len Schwer
Schwer Engineering & Consulting Services
Len@Schwer.net

The LS-DYNA Yahoo Group archive contains a wealth of information that can be helpful to any LS-DYNA user. We suggest you review the archives when you are seeking help on any topic related to LS-DYNA. *NOTE: Questions and responses may have been edited for clarity & brevity.*

This installment of "Yahoo Yammerings" features several questions and responses from the past month of postings to the LS-DYNA Yahoo Group:
1. *Residual stresses after forming?*
2. **Section_Shell_Elform versus *Control_Shell_Theory?*
3. **Load_Blast and AIR3D pressure?*

Residual stresses after forming?

When sheet metal is plastically deformed, it undergoes strain hardening. Most of the time, I have seen the strain hardening considered after forming. After forming one stage, people consider the effect of strain hardening while analyzing the second stage. However, during forming the material undergoes bending, and bending near the radius. So the unbending which follows the bending is a tensile stress after localized strain hardening. How is this taken into account and will this affect the residual stress remaining in the component after forming?

Reply by Chris Galbraith - MFAC, Canada, www.mfac.com

Maybe I am missing the questioner's point, but it is absolutely clear that LS-DYNA tracks the work hardening, and the effective plastic strain, during all phases - the original forming, second stage, etc. The process is as follows: LS-DYNA first solves for nodal accelerations by calculating the external and internal forces and equating them to the nodal mass times the accelerations. Then it twice integrates the accelerations to get the nodal displacements using the central difference method. From that point, it calculates the strain increments at the element integration points. For shell elements, the bending and unbending determined from nodal rotations leads to strain differences through the thickness. At each integration point, the strain increments are combined to produce an increment in the effective plastic strain. Note that increments in effective plastic strain are by definition all positive. Once the effective plastic strain is calculated, the flow stress is obtained from the stress-strain curve (constitutive model). The effective plastic strain and the flow stress are stored as history variables for each integration point in the element, and work hardening is stored, and updated, at each time step. In this way work hardening is tracked throughout the calculation.

If you could return an element to its original shape, the code would still have an increase in effective plastic strain, and in the flow stress. This is known as redundant work.

Note that with a few exceptions, material models in LS-DYNA do not keep track of strain components. To get strain components you need to use the post-processor. But all plasticity models have a history variable that tracks the effective plastic strain, and another one that tracks the flow stress for each integration point. Note that if a kinematic hardening material model is used, there is also a history variable which tracks the back stress.

Effective plastic strain can only increase, never decrease. Stress can of course fluctuate up and down. The flow stress can only increase, never decrease.

Finally, the stress-strain curve that is input is supposed to be an effective true stress, effective logarithmic plastic strain curve, i.e. the elastic strain components are subtracted out. Whether this comes from a biaxial test or from a uniaxial test is irrelevant. In practice, it usually comes from a uniaxial test, but the stress- strain curve is, in theory, a function of the material and is independent of specimen geometry or strain path used.

All finite element codes that have plastic material models have got to handle this properly, or there would be huge negative implications. LS-DYNA has been used for modeling sheet metal forming since at least 1989, and it has proven to be extremely capable. If you need to prove to yourself that LS-DYNA correctly tracks work hardening, create a tensile test specimen, fix one end, and impose a cyclic displacement on the other end, and then plot the effective stress against the effective plastic strain for an element in the gauge length. You will see that it behaves properly.

*Section_Shell_Elform versus *Control_Shell_Theory?

What is the difference between the setting of shell type under *SECTION_SHELL_ELFORM and the shell theory setting under *CONTROL_SHELL_THEORY? Which one is controlling and what is the point of having both of them?

Reply by Jim Kennedy

I believe that the *CONTROL_SHELL (THEORY) entry provides the default value for all shell parts defined in your simulation while *SECTION_SHELL (ELFORM) allows you to select whatever shell formulation you desire for that particular part.

Reply by Denis Corkery

When you define *SECTION_SHELL but do not define an entry for the ELFORM switch, i.e. it is left blank or has a zero entry, this will revert to a default value which is determined by the THEORY switch on *CONTROL_SHELL. This is normally set to Type 2 BT element as you have already noted. You then have the option of changing the element type that the *SECTION_SHELL will default to by changing the THEORY switch to point to an alternative element type. The ELFORM switch on the *SECTION_SHELL card should always override the element type selected by the THEORY switch, if it points to an alternative or non-default element type.

If this sounds a little confusing, the simplest answer is to ignore the THEORY switch altogether and ensure that you explicitly specify an ELFORM type at all times unless you are happy to default to the Type 2 BT that the THEORY switch redirects provides.

LS-DYNA Yahoo Groups

There are over 2500 subscribers from all over the world. The group currently averages about 270 messages per month. You can subscribe to the group by sending an email request to LS-DYNA-subscribe@yahoogroups.com, or by visiting the Yahoo Groups web site http://groups.yahoo.com

Generally, the quickest/best responses are to those questions posed with the most specifics. General questions such as "How do I use XXX feature?" either go unanswered, or are answered by Jim Kennedy with links to appropriate references in the growing LS-DYNA related literature, e.g. see the archive of LS-DYNA Conference proceedings at www.dynalook.com

Effective plastic strain is a monotonically increasing scalar value which is calculated incrementally as a function of (Dp)ij, the plastic component of the rate of deformation tensor. In tensorial notation, this is expressed as...

epspl=integral over time of (depspl)=integral[sqrt(2/3(Dp)ij*(Dp)ij)]*dt

Effective plastic strain grows whenever the material is actively yielding, i.e., whenever the state of stress is on the yield surface.

In contrast, the tensorial strain values, written by LS-DYNA when **STRFLG** is set to 1 in ***DATABASE_EXTENT_BINARY**, are not necessarily monotonically increasing as they reflect the current, total **(elastic+plastic)** state of deformation. To fringe the tensorial strains in LS-PrePost, click **Fcomp > Strain**.

Effective strain, expressed in tensorial notation, is **sqrt(2/3(eps)ij*(eps)ij)**; (See p. 461 of LS-DYNA Theory Manual 2006). This is NOT the same thing as effective plastic strain.

Other measures of strain can be fringed in LS-PrePost but these are calculated by LS-PrePost from nodal displacements, e.g.,

FCOMP > Infin ; (infinitesimal or engineering strain)
FCOMP > Green
FCOMP > Almansi

Effective stress, also known as von Mises stress, is defined as follows:

sigvm=1/sqrt(2)*sqrt[(sigx-sigy)^2+(sigy-sigz)^2+(sigz-sigx)^2+6*sigxy^2+6*sigyz^2+6*sigzx^2]

Comments regarding these notes can be directed to jday@lstc.com

ESoCAET - European School of Computer Aided Engineering Technology

The internationally recognized and accepted academic master's degree is rapidly gaining significance in Germany, as well as Europe for career development. In the traditional fields, such as Business Administration, many courses leading to a master's degree are offered by diverse organizations and universities.

In the growing field of Computer Aided Engineering (CAE) there is an increasing demand for highly qualified CAE users focusing on a career in the CAE business.

The master's degree in Applied Computational Mechanics, offered as a part-time course, provides students with a wide range of CAE knowledge. They are kept up-to-date with the latest CAE developments and taught how to solve complex simulation problems. The master's degree also includes soft skills, such as management issues and presentation techniques, consequently graduates are fully-trained to work as technical managers or specialized engineers in R & D.

Universities

The master's degree course is held by two Bavarian Universities of Applied Sciences: Fachhochschule Ingolstadt in the home town of AUDI, and Fachhochschule Landshut in the vicinity of BMW. Both universities offer students the very best study conditions and place strong emphasis on scientifically based application. Last generation equipment is available to all students both in laboratories and on campus.

Lectures

Course contents is mainly conveyed by interactive seminars and practical exercises, and completed by intensive practice on the computer. Lectures are held by experienced academic staff from the Ingolstadt and Landshut Universities of Applied Sciences. Additionally, students will benefit from the know-how of renowned professors from other universities, and CAE industry experts.

Public Private Partnership

The Master's degree course was developed by CADFEM GmbH, Ingolstadt University of Applied Sciences and Landshut University of Applied Sciences, a partnership which ensures the course is geared to the demands of industry. CADFEM is responsible for organizational issues and contracting lecturers.

International standards & title

The curriculm is organized into modules, matched with an internationally approved credit point system (ECTS). The program has a total of 90 credits, and the resulting qualification is a Master of Engineering (M. Eng.) awarded by both universities.

For complete Information:

www.esocaet.com

Participant LS-DYNA® Resource Page (alpha order)
Fully QA'd by Livermore Software Technology Corporation

SMP and MPP Hardware and OS

FUJITSU

FUJITSU Prime Power	SUN OS 5.8
FUJITSU VPP	Unix_System_V

HP

HP PA-8X00	HP-UX 11.11. and above
HP IA-64	HP-UX 11.22 and above
HP Opteron	Linux CP4000/XC
HP Alpha	True 64

IBM

IBM 4/5	AIX 5.1, 5.2 5.3
IBM Power 5	SUSE 9.0

INTEL

INTEL IA32	Linux, Windows
INTEL IA64	Linux
INTEL Xeon EMT64	Linux, Windows 64

NEC

NEX SX6	Super-UX

SGI

SGI Mips	IRIX 6.5X
SGI IA64	SUSE 9 w/Propack 4 Red Hat w/ Propak 3

SUN

SUN Sparc	5.8 and above
SUN Opteron	5.8 and above

Participant LS-DYNA® Resource Page (alpha order)
Fully QA'd by Livermore Software Technology Corporation

MPP and Interconnect MPI

FUJITSU

	O/S	HPC Interconnect	MPI Software
FUJITSU Prime Power	SUN OS 5.8		
FUJITSU VPP	Unix_System_V		

HP

	O/S	HPC Interconnect	MPI Software
HP PA8000	HPUX		
HPIA64	HPUX		
HP Alpha	True 64		

IBM

	O/S	HPC Interconnect	MPI Software
IBM Power 4/5	AIX 5,1, 5.2, 5.3		
IBM Power 5	SUSE 9.0		LAM/MPI

INTEL

	O/S	HPC Interconnect	MPI Software
INTEL IA32	Linux, Windows	InfiniBand (Voltaire), MyriCom	LAM/MPI, MPICH, HP MPI, SCALI
INTEL IA64	Linux		LAM/MPI, MPICH, HP MPI
INTEL Xeon EMT 64	Linux	InfiniBand(Topspin, Voltaire), MyriCom, PathScale InfiniPath	LAM/NPI, MPICH, HP MPI, INTEL MPI, SCALI

NEC

	O/S	HPC Interconnect	MPI Software
NEX SX6	Super-UX		

SGI

SGI Mips	IRIX 6.5 X	NUMAlink	MPT
SGI IA 64	SUSE 9 w/Propack 4 RedHat w/Propack 3	Numalink, InfiniBand(Voltaire)	MPT, Intel MPI, MPICH

SUN

Sun Sparc	5.8 and above		
Sun Opteron	5.8 and above		LAM/MPI

LS-DYNA® Resource Page - Participant Software

Interfacing or Embedding LS-DYNA - Each software program can interface to all, or a very specific and limited segment of the other software program. The following list are software programs interfacing to or having the LS-DYNA solver embedded within their product. For complete information on the software products visit the corporate website.

ANSYS - ANSYS/LS-DYNA
ANSYS/LS-DYNA

Built upon the successful ANSYS interface, ANSYS/LS-DYNA is an integrated pre and postprocessor for the worlds most respected explicit dynamics solver, LS-DYNA. The combination makes it possible to solve combined explicit/implicit simulations in a very efficient manner, as well as perform extensive coupled simulations in Robust Design by using mature structural, thermal, electromagnetic and CFD technologies.

AI*Environment:

A high end pre and post processor for LS-DYNA, AI*Environment is a powerful tool for advanced modeling of complex structures found in automotive, aerospace, electronic and medical fields. Solid, Shell, Beam, Fluid and Electromagnetic meshing and mesh editing tools are included under a single interface, making AI*Environement highly capable, yet easy to use for advanced modeling needs.

ETA — DYNAFORM

Includes a complete CAD interface capable of importing, modeling and analyzing, any die design. Available for PC, LINUX and UNIX, DYNAFORM couples affordable software with today's high-end, low-cost hardware for a complete and affordable metal forming solution.

ETA — VPG

Streamlined CAE software package provides an event-based simulation solution of nonlinear, dynamic problems. eta/VPG's single software package overcomes the limitations of existing CAE analysis methods. It is designed to analyze the behavior of mechanical and structural systems as simple as linkages, and as complex as full vehicles.

MSC.Software - MSC.Dytran LS-DYNA

Tightly-integrated solution that combines MSC.Dytran's advanced fluid-structure interaction capabilities with LS-DYNA's high-performance structural DMP within a common simulation environment. Innovative explicit nonlinear technology enables extreme, short-duration dynamic events to be simulated for a variety of industrial and commercial applications on UNIX, Linux, and Windows platforms. Joint solution can also be used in conjunction with a full suite of Virtual Product Development tools via a flexible, cost-effective MSC.MasterKey License System.

MSC.Software - MSC.Nastran/SOL 700

The MSC.NastranTM Explicit Nonlinear product module (SOL 700) provides MSC.Nastran users the ability access the explicit nonlinear structural simulation capabilities of the MSC.Dytran LS-DYNA solver using the MSC.Nastran Bulk Data input format. This product module offers unprecedented capabilities to analyze a variety of problems involving short duration, highly

dynamic events with severe geometric and material nonlinearities.

MSC.Nastran

Explicit Nonlinear will allow users to work within one common modeling environment using the same Bulk Data interface. NVH, linear, and nonlinear models can be used for explicit applications such as crash, crush, and drop test simulations. This reduces the time required to build additional models for another analysis programs, lowers risk due to information transfer or translation issues, and eliminates the need for additional software training.

MSC.Software — Gateway for LS-DYNA

Gateway for LS-DYNA provides you with the ability to access basic LS-DYNA simulation capabilities in a fully integrated and generative way. Accessed via a specific Crash workbench on the GPS workspace, the application enhances CATIA V5 to allow finite element analysis models to be output to LS-DYNA and then results to be displayed back in CATIA. Gateway for LS-DYNA supports explicit nonlinear analysis such as crash, drop test, and rigid wall analysis.

Oasys software for LS-DYNA

Oasys software is custom-written for 100% compatibility with LS-DYNA. Oasys PRIMER offers model creation, editing and error removal, together with many specialist functions for rapid generation of error-free models. Oasys also offers post-processing software for in-depth analysis of results and automatic report generation.

EASi-CRASH DYNA

EASi-CRASH DYNA is the first fully integrated environment for crashworthiness and occupant safety simulations with LS-DYNA, and covers the complete CAE-process from model building and dataset preparation to result evaluation and design comparisons.

EASi-CRASH DYNA can be used for concept crash, FE crash and coupled rigid body/FE crash simulations in conjunction with MADYMO.
Full capability to handle IGES, CATIA V4, CATIA V5, UG and NASTRAN files.

APTEK

The MMCD is a graphics-based and menu-driven program that interfaces with the LS-DYNA library of material models and the LS-OPT optimization code. The core of the MMCD is the driver, which calculates the stress-strain behavior of material models driven by combinations of strain increments and stress boundary conditions, i.e. pure shear stress, and combinations of uniaxial, biaxial, and triaxial compression and tension. MMCD input and output is accessed via pre- and post-processors; graphical user interfaces (GUIs) for easily selecting the material model parameters and load histories, and for plotting the output in both two (stress-strain curves) and three (yield surfaces) dimensions. The pre-processor, driver, and post-processor are combined into a web downloadable software package that operates seamlessly as a single code.

FEA Information Participants –
Company name takes you directly to Website

OASYS Ltd: Markets engineering software products. Consulting engineers, planners and project managers working in all areas of the built environment.

JRI Solutions Ltd.: Specializing in Research & Consulting; System Consulting, Frontier Business, System Integration and Science Consulting.

Hewelett Packard: Personal computing, mobile computing, network management, 3-D graphics and information storage.

ANSYS Inc.: Develops, markets, supports and delivers collaborative analysis optimization software tools.

SGI: Silicon Graphics, Inc., is a leader in high-performance computing, visualization, and storage.

MSC.Software: Information technology software and services provider.. Products & services used to enhance & automate the product design/manufacturing process.

NEC: A history of more than 100 years of leadership/innovation in the core high-technology sectors of communications, computers/electronic components

---

INTEL: For more than three decades, Intel Corporation has developed technology enabling the computer and Internet revolution that has changed the world.

Engineering Technology Associates, Inc.: Provides engineering & IT services & has created the streamlined simulation software packages DYNAFORM and VPG

ESI Group: A software editor for the numerical simulation of prototype and manufacturing process engineering in applied mechanics.

Microsoft: For customers solving complex computational problems, Microsoft Windows Compute Cluster Server 2003 accelerates time-to-insight.

BETA CAE Systems S.A.: Specialized in the development of state of the art CAE pre- and post-processing software systems.

FEA Information Participants –
Company name takes you directly to Website

SUN Microsystems Inc.:   Provides network computing infrastructure solutions that include computer systems, software, storage, and services.

Detroit Engineered Products: A Michigan based engineering consulting and software products firm specializing in the area of Product Development products and solutions.

APTEK: Among the software developed APTEK develops and licenses an interactive program for driving LS-DYNA material models - the Mixed Mode Constitutive Driver (MMCD).

PANSAS: High performing Parallel Storage for scalable Linux clusters. Delivering exceptional scaling in capacity and performance for High Performance Computing (HPC) organizations.

Intelligent Light: A a world leader in the development and delivery of software for computational fluid dynamics (CFD) users. We help the world's best engineering and research organizations maximize the productivity and impact of their CFD capabilities

---

LS-DYNA® Software Distributors - Alphabetical order by Country

Australia	**Leading Engineering Analysis Providers**
Canada	**Metal Forming Analysis Corporation**
China	**Arup**
Germany	**CAD-FEM**
Germany	**DynaMore**
India	**Oasys, Ltd.**
India	**Cranes Software International Limited (CSIL),**
Italy	**EnginSoft Spa**
Japan	**The Japan Research Institute**
Japan	**ITOCHU Techno-Solutions Corporation**
Korea	**Korean Simulation Technologies**
Korea	**Theme Engineering**
Netherlands	**Infinite Simulations Systems B.V.**
Russia	**State Unitary Enterprise - STRELA**
Sweden	**Engineering Research AB**
Taiwan	**Flotrend Corporation**
USA	**Engineering Technology Associates, Inc.**
USA	**Dynamax**
USA	**Livermore Software Technology Corp.**
UK	**ARUP**

Consulting and Engineering Services

Australia **Manly, NSW**	**Leading Engineering Analysis Providers (LEAP)** **Greg Horner info@leapaust.com.au** **02 8966 7888**
Canada **Kingston, Ontario**	**Metal Forming Analysis Corp. - (613) 547-5395** **Chris Galbraith galb@mfac.com**
Germany **Alzenau**	**CARHS - 49 6023 96 40 60** **info@carhs.de**
Italy **Firenze**	**EnginSoft Spa - 39 055 432010** **info@enginsoft.it**
UK **Solihull,** **West Midlands**	**ARUP - 44 (0) 121 213 3317** **Brian Walker brian.walker@arup.com**
USA **Austin, TX**	**KBEC L.C - (512) 363-2739** **Khanh Bui kdbui@sbcglobal.net**
USA **Windsor, CA**	**SE&CS - (707) 837-0559** **Len Schwer len@schwer.net**
USA **Troy, MI**	**Engineering Technology Associates, Inc:** **(248) 729-3010**
USA **Corvallis, OR**	**Predictive Engineering - (1-800) 345-4671** **George Laird george.laird@predictiveengineering.com**
USA **Troy, MI**	**Detroit Engineered Products**
USA **Troy, MI**	**Engineering Technology Associates, Inc.** **(248) 729-3010** **sales@eta.com**
USA **Austin, TX**	**Friedman Research Corporation - (512) 247-2277**

Educational & Contributing Participants
Alphabetical Order By Country

China	Dr. Qing Zhou	Tsinghua University
India	Dr. Anindya Deb	Indian Institute of Science
Italy	Professor Gennaro Monacelli	Prode – Elasis & Univ. of Napoli, Frederico II
Russia	Dr. Alexey I. Borovkov	St. Petersburg State Tech. University
USA	Dr. Ted Belytschko	Northwestern University
USA	Dr. David Benson	University of California – San Diego
USA	Dr. Bhavin V. Mehta	Ohio University
USA	Dr. Taylan Altan	The Ohio State U – ERC/NSM
USA	Dr. Ala Tabiei	University of Cincinnati

Informational Websites
The LSTC LS-DYNA Support site: www.dynasupport.com

LS-DYNA Support Site	FEA Informationwebsites
LS-DYNA Examples (more than 100 Examples)	LS-DYNA Conference Site
TopCrunch – Benchmarks	LS-DYNA Publications to Download On Line
LS-DYNA Publications	LSTC_LS-PrePost Tutorials
CADFEM GmbH Portal	

Seminars From carhs.training gmbh
Managing Director: Rainer Hoffmann
www.carhs.de

carhs.training gmbh
Siemensstr. 12
63755 Alzenau
Tel. +49 (0) 6023 964063
trainingcenter@carhs.de
http://www.carhs.de/en

Training Center: Alzenau – A few of the many 2008 classes

Advanced Complexity-based Robust Design	Wed, 23 Apr
Introduction to Passive Safety of Motor Vehicles	Wed, 28 May
Euro NCAP and global Tests for Consumer Protection through Active and Passive Safety	Mon, 23 Jun
Frontal-Restraint Systems according to FMVSS 208 and Euro NCAP	Wed, 25 Jun
Side Impact - Requirements and Development Strategies	Thurs, 03 Jul

LSTC Training Classes in California & Michigan

	California	Michigan
Advanced Impact	**Aug 5-8**	
Advanced Option	**June 23-24**	**Dec 15-16**
ALE/Eulerian & FSI		
Blast & Penetration	**Aug 18-19**	
Composite Materials	**June 25-26**	
Concrete & Geomaterial Modeling	**Sept 25-26**	
Contact	**Aug 14-15**	**Sept 16-17**
Heat Transfer & Thermal-Stress		
Implicit	**Aug 12-13**	
Intro to LS-DYNA	**May 06-09** **July 22-25** **Nov 11-14**	**June 17-20** **Sept 09-12** **Dec 09-12**
Intro to LS-OPT	**Sept 09-14**	**Apr 08-11**
Material Modeling Using User Defined Options	**June 30-July01**	
Mesh Free Methods (SPH & EFG)		

LSTC & INTEL® One Day Training Course

Intel® Cluster Ready – Cluster Support Training

This one-day course provides cluster administrators and support technicians with the latest information about the Intel® Cluster Ready program. Using exercises and hands-on labs, attendees will discover how the Intel® Cluster Ready program makes it easier to purchase, deploy, and maintain clusters built with Intel components. Attendees will learn how Intel works with platform vendors, solution integrators, and software developers to create solutions for the end user that are ready-to-use. A comprehensive tool to check and troubleshoot a cluster is provided with every Intel Cluster Ready cluster, and this will be reviewed in detail.

Agenda

Module 1: Intel® Cluster Ready Program Overview

This module will discuss the Intel® Cluster Ready program and how it applies to different program members.

At the end of this module, students will be able to

- Describe the purpose of the Intel Cluster Ready program
- List the tools and resources provided by the program
- Describe what is an Intel Cluster Ready reference implementation
- Explain how to obtain Intel Cluster Ready support

Module 2: Certification & Registration Program

Intel® Cluster Ready certification and registration procedures provide a common, proven method to identify compliance with the Intel® Cluster Ready specification. Documented steps are required to achieve application registration or platform certification.

At the end of this module, students will be able to

- Explain the purpose of the certification and registration program
- List the steps necessary to complete a platform certification
- List the steps necessary to complete an application registration

Module 3: Cluster Design

The Intel® Cluster Ready specification defines the minimum and recommended components and design required for a certified cluster. This module identifies important components of the specification.

At the end of this module, students will be able to

- Defined common cluster terminology
- Identify important requirements and recommendations given in the Intel® Cluster Ready specification

Module 4: Intel® Cluster Tools

Intel® Cluster Tools provide a leading development environment for cluster applications. This module describes these development tools and how they integrate into the Intel® Cluster Ready program.

At the end of this module, students will be able to

- Describe the Intel® development tools in the Intel® Cluster Ready SDK
- Describe how to use installed tools or their runtime libraries

Module 5: Intel® Cluster Checker: Introduction

Intel® Cluster Checker provides an automated utility for analyzing the configuration and performance of Linux clusters. This module describes the architecture and operation of Intel® Cluster Checker.

At the end of this module, students will be able to

- Explain how Intel® Cluster Checker works
- Create a basic cluster configuration file
- Execute Cluster Checker

Module 6: Intel® Cluster Checker: Advanced Configuration

Intel Cluster Checker includes options to customize operation to meet requirements for all certified clusters. It also provides features to automate operation and analyze output. This module explores many of those options.

At the end of this module, students will be able to

- Customize configuration options for different performance metrics and messaging fabrics
- Add and remove modules
- Integrate Cluster Checker into a job scheduler (PBS)
- Create node groups with different configurations

Module 7: Intel® Cluster Checker: Troubleshooting

In addition to analyzing the health and compliance of clusters, Intel® Cluster Checker provides an invaluable tool for locating problems on clusters. In the module, features of Intel® Cluster Checker for debugging and troubleshooting are reviewed.

At the end of this module, students will be able to

- Explain the effects of verbosity on Cluster Checker operation
- Activate debugging on individual modules
- Create custom tests using generic modules

For Information Contact:

Dr. Wayne Mindle,
 LSTC – 925-449-2500
wlm@lstc.com

The 19th Annual HP CAE Symposium

Advancing Innovations with High-Productivity Computing

HP and Livermore Software Technology Corporation invite you to a free, one-day CAE Symposium.

Tuesday April 8th, 2008
8:00 a.m. – 4:00 p.m.
Long Beach Mariott,
4700 Airport Plaza Drive
Long Beach, CA 90815

Complete symposium details, agenda and registration

For Questions contact:
Christine Fronczak
408-447-7783

Keynote Presentations

"The unexpected Challenges of an Evolving Enterprise – New Demands on CAE"
John S. Hurley, PhD.,
Sr. Manager Distributed Computing, Networked Systems Technology,
The Boeing Company

"HPC and CAE: Trends and Paradoxes"
Knute Christiansen, Worldwide HPC Solutions Marketing Manager
Hewlett-Packard

LSTC Presentation

"LS-DYNA – It's Applications and Potentials"
Ian Do, PH.D., Senior Scientist
Livermore Software Technology Corp.

This popular one-day symposium will include over 20 of the industry's leading CAE applications experts, industry speakers and more then 200 engineers and managers to discuss current technology trends in computational engineering.

Among the Industry Experts attending are:

- ANSYS
- ESI Group
  Intelligent Light
- Livermore Software Technology Corp.
- MSC Software
- The Boeing Company

Symposium Tracks

- Impact analysis
- Structural analysis
- Computational fluid dynamics
- Visualization
- Engineering IT technologies

The "1st International Conference on Hot Sheet Metal Forming of High performance Steel, October 22-24, 2008, Kassel, Germany"

The ongoing demand towards the application of high-performance steel and, beyond that, towards steel parts with tailored properties leads inevitably to a demand for improved metal forming technologies. Among these technologies hot sheet metal forming is gaining increasing importance. The technological reason for that can be seen in the ability not only to improve formability at elevated temperatures but especially in the potential of simultaneous control of complex micro structures during thermo-mechanical processing. This of course involves challenges in terms of tailored properties through optimisation of varying microstructure over the overall part geometry. As the first in a series of international conferences, to be alternatively held in Kassel and in Luleå, will highlight these multiple aspects from a scientific as well as from an industrial viewpoint

Website   Conference Invitation - pdf

Conference Secretary Contact Information

Nicolas Saba
University of Kassel
Chair of Metal Forming TechnologyKurt-Wolters-Strasse 3
D-34125 Kassel, Germany
Phone: +49 -(0) 561-804-1976
Fax: +49 -(0) 561-804-2706
e-mail: **saba@uni-kassel.de**

nHance   **Oasys**   **ARUP**
             The software house of ARUP

Oasys LS-DYNA® Update Meeting in India - Wed 2nd April 2008

Oasys Ltd and nHance Engineering Solutions Pvt Ltd are pleased to announce the forthcoming Oasys LS-DYNA Update meeting being held on Wednesday 2nd April 2008 at Le Meridien hotel in Pune, India.

This full day free of charge event covers both LS-DYNA and Oasys software and is a perfect opportunity to find out about current and future developments and how the software is being used in the engineering community.
Presentations will also include developments in FE dummy and barrier models.

We are pleased to welcome Mr. Sarang Kshirsagar - Project Manager(Crash) of TATA Motors who will be speaking at this event

Registration

Please send your registration to this event by email to india.support@arup.com with your name, company/affiliation and telephone number.

Venue

The event will be held at Le Meridien Hotel in Pune which is situated in the heart of the business district.

Raja Bahadur Mill Road, Pune 41001, India
+91 (20) 26050505

If you plan to stay over before or after the event, we are pleased to confirm that we have negotiated a special rate for attendees of the Oasys LS-DYNA Update meeting. Please contact us for assistance.

Contact Details: If you have any queries regarding this event you can contact:

Ms Rafia Sultana, nhance Engineering Solutions(P)Ltd (Part of the ARUP Group)
Plot No. 39, Ananth Info Park, Opposite Oracle Campus
HiTec City-Phase II, Madhapur, Hyderabad-500081, India
Tel: +91 (0) 40 44369797/8   -   Email: india.support@arup.com

Provisional Agenda for Oasys LS-DYNA India Update Meeting
Date: 2nd April 2008 - Place: Le Meridien Hotel, Pune

Time	Session	Length of Presentation	Presenter
09:30AM-10:00AM	*Registration with tea & coffee*		
10:00AM-10:10AM	Welcome Address	10 minutes	Prasad Nadipalli & Lavendra Singh
10:10AM-11:10AM	Session 1: LS-DYNA-1 Oasys PRIMER Introduction	30 minutes 30 minutes	Brian Walker Miles Thornton
11:10AM-11:40AM	*Tea & Coffee*		
11:40AM-1:00PM	Session 2: Oasys Post-processing Introduction Arup Car Project (SLC) FE Models: FAT dummies & Arup Cellbond Barriers	30 minutes 30 minutes 20 minutes	Roger Hollamby Neil Butcher/Neil Ridley Brian Walker/Lavendra Singh
1:00PM-2:00PM	*Lunch*		
2:00PM-3:30PM	Session 3: Oasys PRIMER Advanced Oasys Post-processing Advanced Oasys REPORTER	40 minutes 30 minutes 20 minutes	Miles Thornton Roger Hollamby Miles Thornton
3:30PM-4:00PM	*Tea & Coffee*		
4:00PM-5:10PM	Session 4: Invited Speaker 1 Invited Speaker 2 LS-DYNA-2	20 minutes 20 minutes 30 minutes	Mr.Sarang Kshirsagar, Tata Motors TBC Brian Walker
5:10PM-5:20PM	Closing Address & Raffle		
5:20PM-5:50PM	Time for networking and Oasys LS-DYNA Team available for discussion, questions & software demonstrations		

MSC Software Corporation's 2007 Americas VPD Conference: Gaining Competitive Advantage through Engineering Productivity

www.mscsoftware.com/events/vpd2007/na/?Q=135&Z=341&Y=363&M=459

Marriott at the Renaissance Center, Detroit, Michigan, October 11-12, 2007

This year's VPD Conference brought executives, engineers, designers, and academics together to explore how industry leaders are improving virtual product development processes across engineering disciplines, user communities, and supply chains to accelerate product development and enhance productivity.

In addition, MSC management and experts were available to discuss current and future product development requirements. They added insight into how to improve product performance and reliability through simulation enhancements, testing and manufacturing.

Conference Highlights:

- Provided insight into industry trends
- High profile keynotes
- Rich technical sessions
- Demos
- Opportunity to meet with MSC partners
- MSC NV Symposium (directly preceeding the VPD Conference)

.

- Fellowship amongst technical community
- Entertaining Casino Night

Quotes from the conference

- "Talking to some of your developers was a big plus by giving me insights into both your current software and what is coming in the future of FEA."
- "Very informative in terms of benchmarking CAE software use trends in the North American market. Got introduced to 'SimExpert and SimManager' suite of products. Also to MD Nastran. Which does seem to have a lot of potential for small vehicle testing/simulation houses."
- "To my surprise, MSC products are extensively used in many unexpected areas... very interesting information about SimEnterprise products. I think they will change our approach to design and analysis organization.

Invitation to The 2nd PhilonNet CAE Conference Athens May 08 2008
"Drive Innovation with Simulation"
www.philonnet.gr/events/index.html

"After a well received and successful first CAE conference last year with more than 80 attendees, from various industries and academic institutes, PhilonNet anticipates repeating this success at 8th May 2008 conference, and invites you to participate.

Presentations will include speakers from many countries such as: Denmark, the UK, Italy, the United States as well as from Greece.

Dr. Andreas Vlahinos will present applications of LS-DYNA.

Among the topics being presented:

- computational fluid dynamics (CFD)
- fluid-structure-interaction
- crash impact
- drop test
- process integration and design optimization
- crack propagation and fatigue
- ergonomics and biomechanics
- building safety and comfort
- Metal Forming
- Vehicle Safety and Composites

Many of the speakers are involved in large-scale and multi-national industrial and/or research projects and will share their experience and knowledge of best practices in product development and simulation technologies.

The one-day conference begins with a general session in the morning and continues with parallel sessions and workshops in the afternoon.

For more details please visit PhilonNet web site at: www.philonnet.gr/events"

Best Regards,
Stavroula

PhilonNet
Engineering Solutions
LSTC Direct Distributor in Greece
Kerkiras 140, Kipseli
Athens, 11363
Greece
tel: +302108217235
fax: +30 2108217254
mob: +30 6946468478
email:
stavroula.stefanatou@philonnet.gr
website: www.philonnet.gr

10th International LS-DYNA® Users Conference
Keynote Speakers, Sponsors and Exhibitors
June 8-10th, 2008

www.ls-dynaconferences.com

For Sponsor and Exhibitor information – contact Marsha Victory

Keynote Speakers

Professor Ted Belytschko
   Walter P. Murphy Professor
   Northwestern University

Tsuyoshi Yasuki,
   General Manager of CAE Research,
   Development and Application at Toyota

Stephen T. Kozak
   Safety Chief Engineer, Body
Engineering
   North America Engineering
   Ford Motor Company

Chuan-Tao Wang, Ph.D.
   Chief Die Engineer
   GM Technical Fellow Body
   Manufacturing Engineering
   Die Center
   General Motors

Paul Du Bois
   Consulting Engineer

Rahul Gupta, Ph.D.
   Army Research Laboratory
   Aberdeen Proving Ground

John O. Hallquist
   LSTC

Sponsors (includes a booth)

Platinum
   INTEL
   MICROSOFT
   SGI
Gold
   ETA
   HP
Silver & Bronze
   ANSYS
   ARUP
   NEC
   SUN

Exhibitors

AMD	ANSYS	ALTAIR
APTEK	ARUP	
BETACAE	CEI	DATAPOINT
DEPUSA	ETA	ESI
FEA Information	FTSS	HP
INTEL	JRI	MELLANOX
MICROSOFT	MSC.Software	NEC
PANASAS	QLOGIC	SGI
SUN	SCAI	XTREME