



FEA Information International News

www.feainformation.com

Issue March, 2001

The news is a free e-mail publication

FEA Information would like to welcome Professor Ala Tabiei, Assistant Professor, at the University of Cincinnati, as an Educational Participant.

Our goal:

A monthly synopsis of the additions/revisions to the FEA Information web sites.
The news is mailed during the third week of each month.

Feel free to have your associates notify me to be added to our FEA Information mailing list, Marsha – vic@lstc.com

If you have any non-proprietary information, avi's, animated gifs, etc. for our application web sites or technical information news for our newsletter, please feel free to send them to Marsha – vic@lstc.com

When available:

Information from our participating companies, engineers, professors, consultants, and students of FEA Information.

Page	Article	Submitted By:
2	Three-Dimensional R-Adaptivity	Dr. Torodd Berstad Livermore Software Technology Corporation
4	Implicit Notes	Dr. Bradley Maker Livermore Software Technology Corporation
	FEA Site Summary & Participants	Marsha Victory FEA Information

Three-Dimensional R-Adaptivity
Dr. Torodd Berstad
Livermore Software Technology

In version 950 of LS-DYNA, r-adaptivity for two-dimensional plane strain and axisymmetric geometries was introduced for forging simulations. In version 960 similar capabilities for bulk forming are extended to three-dimensional solids using 4-node tetrahedron elements. Volumetric locking is avoided in the tetrahedron elements by using a nodal pressure formulation. Currently, this option is supported for explicit time integration. Note that in version 960 the two-dimensional r-adaptive option is now supported for implicit calculations with coupled heat transfer. This may be discussed in a future newsletter.

The capabilities for three-dimensional remeshing are limited in version 960 and should be used with care since it is still under development. The remeshing is performed at specific time intervals to obtain a new and better mesh. This is similar to sheet metal stamping with adaptivity.

To activate this option set ADPOPT=7 and define a value of ADPFREQ in the *CONTROL_ADAPTIVE command. Also, set ADPOPT=2 (R-adaptive remeshing) under the *PART command to flag the parts which should be remeshed. In addition a new keyword, *CONTROL_REMESHING, is added to control the element sizes during the mesh generation. The calculation can start with a mesh of either 8-node brick elements or 4-node tetrahedrons. During the first remeshing 8-node brick element will be automatic converted to 4-node tetrahedron elements of formulation type 13 (1-point nodal pressure tetrahedron for bulk forming). Boundary conditions should be applied by the *CONSTRAINED_GLOBAL command. Nodes in parts that are remeshed should not be shared with other parts. Loads and interfaces with other parts should be applied by using automatic contact within LS-DYNA.

Adaptive procedure

At the adaptive step, the following steps are performed automatically:

1. Construct the surface mesh. This is the mesh surrounding the volume, which should be remeshed. The mesh consists of simple three node triangles.
2. Smooth the surface mesh. The element edge length will be within the lengths defined by the *CONTROL_REMESHING keyword. On this control card, the maximum value of the edge length, RMAX, should be in the same order of magnitude as the minimum, RMIN.
3. Generate a new mesh consisting of tetrahedron elements inside the surface mesh.
4. Continue the simulation with the new mesh. LS-DYNA performs a least squares remapping of displacement, velocity, stress, and history variables from the old mesh to the new mesh.

Implementation of this procedure in the future for the MPP computers will be straightforward. A domain decomposition will be added after step three just as is done in metal forming applications.

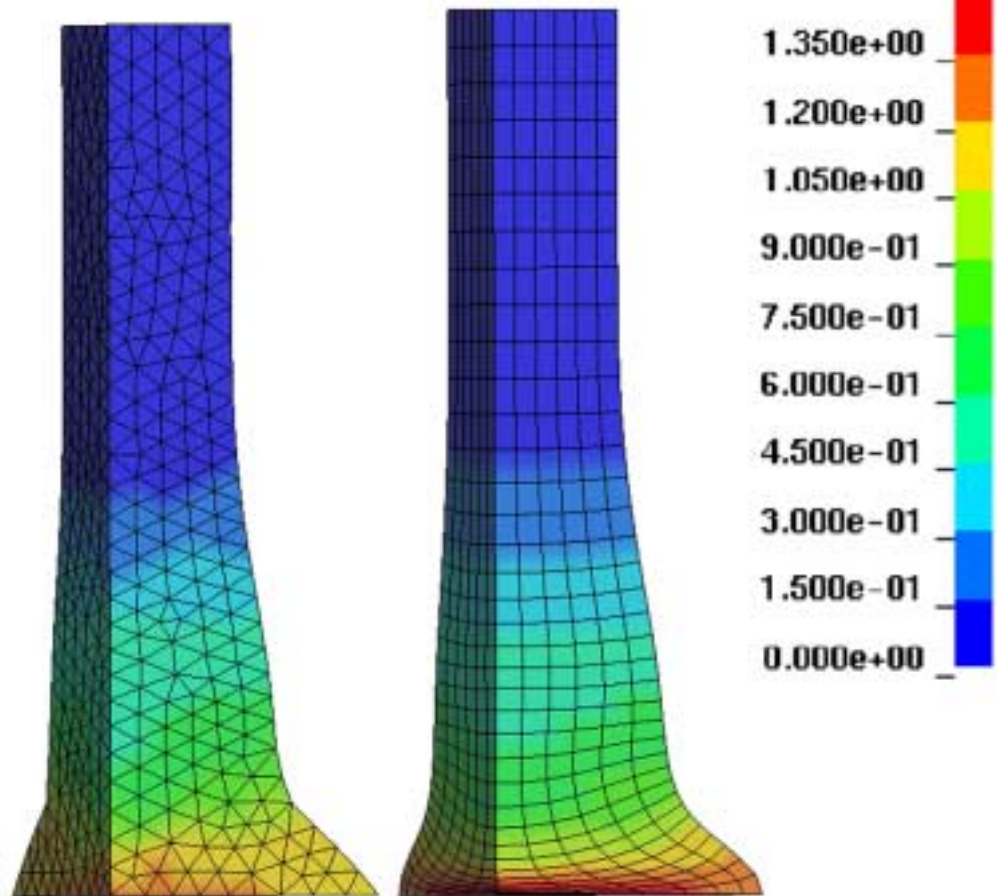
Post-Processing

A sequence of plot files is created which can be read and post-processed by LS-POST. Since the node and element ID's in the remeshed part change after each adaptive step, time history plots by node and element ID are not meaningful. This limitation can be overcome in the future, if the need arises, by adding several new options to LS-POST.

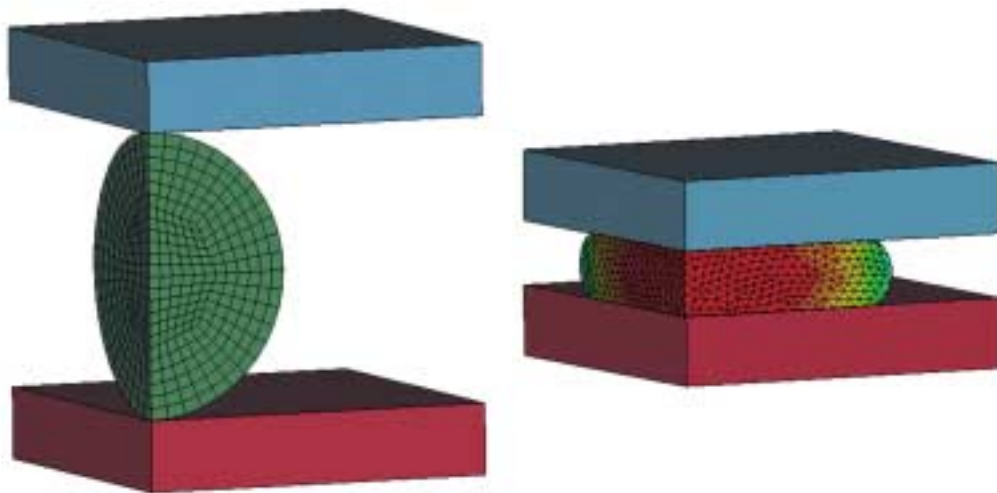
Examples

1. Taylor test: A Taylor bar with remeshing is compared to the results obtained with the constant stress solid element without remeshing. The following figure shows the plastic strain after the problem has completed. In the remeshed bar the maximum plastic strains are somewhat reduced due to the smoothing effect of the larger elements which cannot resolve steep gradients. The deformed shape is very accurate.

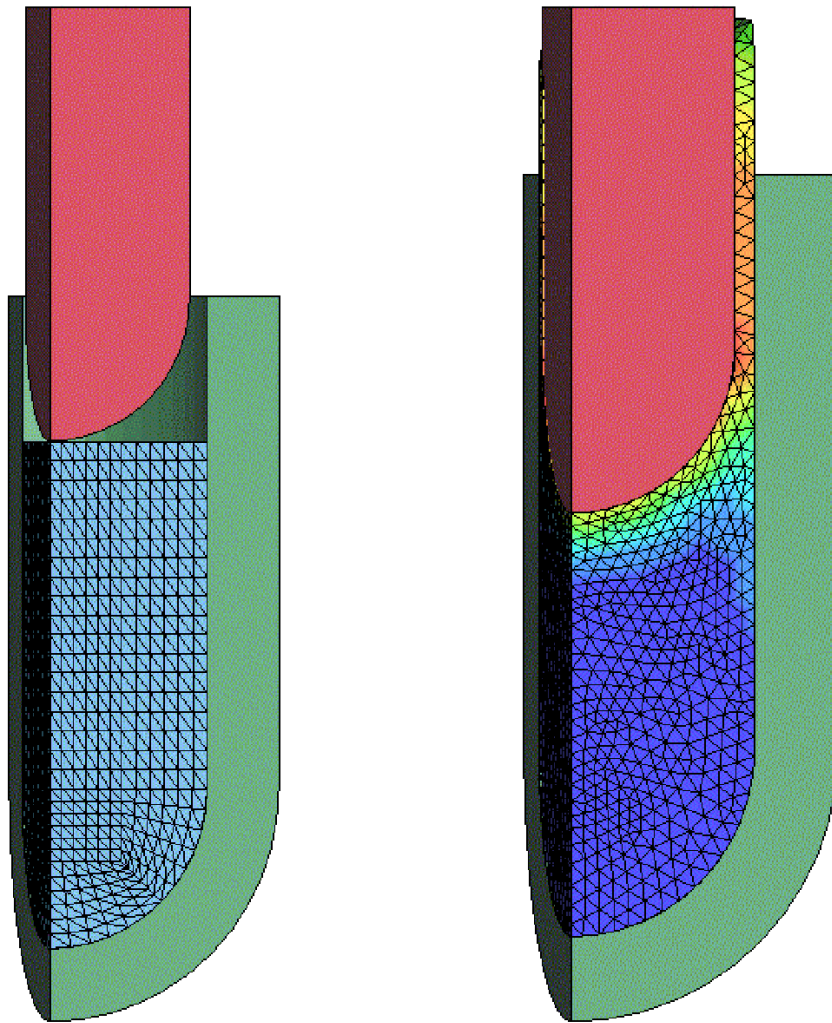
Contours of Effective Plastic Strain
min=2.16413e-12, at elem# 5068
max=1.60869, at elem# 1



2. Simple forging: The figures show a quarter of a circular die before and after forging. The model starts with 8-node solid elements and ends up with 4-node tetrahedron elements



3. Another forging examples: The figures show the initial configuration and a deformed state.



Implicit – LS-DYNA Versions 960
Dr. Bradley Maker
Livermore Software Technology

NEW FEATURES: *CONTROL_IMPLICIT Keywords Renamed

There has been much confusion about the use of keywords for selecting linear and nonlinear implicit analysis, and for controlling options in the linear and nonlinear equation solvers. For this reason, new names have been assigned to these keywords. The old names are still available, but will be undocumented in version 960. The keywords are:

***CONTROL_IMPLICIT_SOLUTION** (replaces ***CONTROL_IMPLICIT_NONLINEAR**):
This keyword is used to select the solution method. Either a linear solution (i.e. NASTRAN) or a nonlinear solution may be chosen. Control parameters for nonlinear and arc length methods are selected here.

***CONTROL_IMPLICIT_SOLVER** (replaces ***CONTROL_IMPLICIT_LINEAR**):
This keyword is used to select options solving the linear algebra problem $[K]\{x\}=\{f\}$.

NEW FEATURES: Dynamic Implicit-Explicit Switching

The implicit-explicit switching feature has been enhanced to allow application to a transient dynamic analysis. Formerly, this method had been applied only for static simulations of metal forming, springback, and roof crush. Now the switching method can be used for dynamic simulations as well.

An example application is the analysis of an impact event, followed by a long duration transient dynamic calculation. The explicit method can be used for the impact portion of the simulation. However, small explicit time steps may be too expensive and provide unnecessary detail for the ensuing low frequency transient response. In this case, the solution can be switched to implicit. The keyword ***CONTROL_IMPLICIT_DYNAMICS** will be used to select either static (default) or dynamic implicit analysis, and the keyword ***CONTROL_IMPLICIT_GENERAL** will be used to specify the implicit time step size. Care must be taken to choose a time step size small enough to resolve the highest frequency of interest, if a dynamic analysis is selected.

As in the static case, the keyword ***CONTROL_IMPLICIT_GENERAL** is used to specify a curve ID which controls the analysis method as a function of time. Switching may occur several times during a simulation, and the simulation may begin and end using either the implicit or the explicit method. At this time, each implicit period of the simulation will run either statically or dynamically.

NEW FEATURES: Implicit-Local SPCs and Sliding Boundary Planes

SPC constraints defined in a local coordinate system, and sliding boundary planes, are now available for use with the implicit method. The input definitions are the same as those used for explicit analysis. The following keywords are used to invoke these options:

***BOUNDARY_SPC_NODE**
***BOUNDARY_SPC_SET**
***BOUNDARY_SLIDING_PLANE**

NEW FEATURES: “Simple” Trimming Option

By specifying a negative value for trimming tolerance on *DEFINE_CURVE_TRIM, a new “simple” trimming feature is activated. In this method, entire elements are deleted during trimming, but the boundary of the resulting mesh is not smoothed. This results in a jagged trimmed edge, but does not alter the shape of elements which remain in the mesh. This mesh produces improved springback results in some cases. More experience with this new feature is needed.

THEORY: Contact gap flag

The contact gap flag IMGAPF has been added as a new input parameter on optional contact card “C”. This parameter selects one of two options for treating situations where gaps are opening:

IGAPF = 1: This option can significantly improve convergence behavior, but can also produce “sticky” interfaces which resist opening of gaps. An indicator of sticking is a fluctuating energy norm during equilibrium iterations, while the displacement norm is well behaved. This option is the default for non-automatic contact interfaces.

IGAPF = 2: This option allows gaps to open freely, but can slow convergence in some problems. This is the default (and only) option for automatic contact interfaces.

The internal function of the gap flag is to control how contact stiffness terms are added to the global stiffness matrix. When IGAPF = 2, stiffness terms are only included for nodes which are in contact. This is a standard implementation, but can suffer from convergence trouble. As an alternative, when IGAPF = 1, stiffness terms are included for nodes which are not yet in contact, but are “close enough”. Of course, contact forces are only generated for nodes which are in contact, regardless of IGAPF.

By default, nodes are “close enough” when they are within two sheet thicknesses of contact. From that distance, contact stiffness is increased linearly with proximity, reaching 100% when contact is actually established. The default gap can be modified using a negative value of IGAPF. In this case, stiffness begins to develop at a gap of $(-IGAPF/10)$ sheet thicknesses. For example, selecting IGAPF = -20 recovers the default value of two thicknesses. Admissible negative values for IGAPF range from -1 to -99, producing values of 0.1 to 9.9 thicknesses.

The “inconsistent” implementation when IGAPF = 1 improves convergence behavior tremendously in many problems. However, this “trick” can corrupt the contact interface force data written to the RCFORC file. The reason this occurs is that stiffness terms for “close” nodes create coupling between the contacting bodies before contact actually occurs. Through this coupling, displacement of one body creates force on the opposing body. These forces occur before contact actually occurs, and are not included in the RCFORC file totals. In extreme cases, when contact stiffness is very high, these forces can be sufficient to deform each body, and equilibrium can be obtained without any true contact.

Figure 1
Three-point bend test simulation, shown before loading (left) and after loading (right)



An example of this case is shown in Figure 1, where a three-point bend test is performed implicitly. The thin, flexible sheet is deformed between rigid rollers under displacement control. For this case, the default penalty stiffness used by LS-DYNA is sufficiently large to deform the flexible sheet due to stiffness coupling alone, and no true contact actually develops. Since there is no true contact, the reaction force from the RCFORC file is zero.

A work-around for this problem is to scale the penalty stiffness to more reasonable values. In the three-point bend problem, the graph in figure 2 shows RCFORC data for several values of stiffness scale factor SLSFAC. No contact forces are output for $SLSFAC = 1.e-2$, reasonable results are obtained with $SLSFAC = 1.e-4$, and excessive penetration develops when $SLSFAC = 1.e-6$.

Figure 2
Reaction force vs. time for the three-point bend test with various values of SLSFAC. Notice that when $SLSFAC = 1.e-2$, zero force is included in the RCFORC file (curve D)

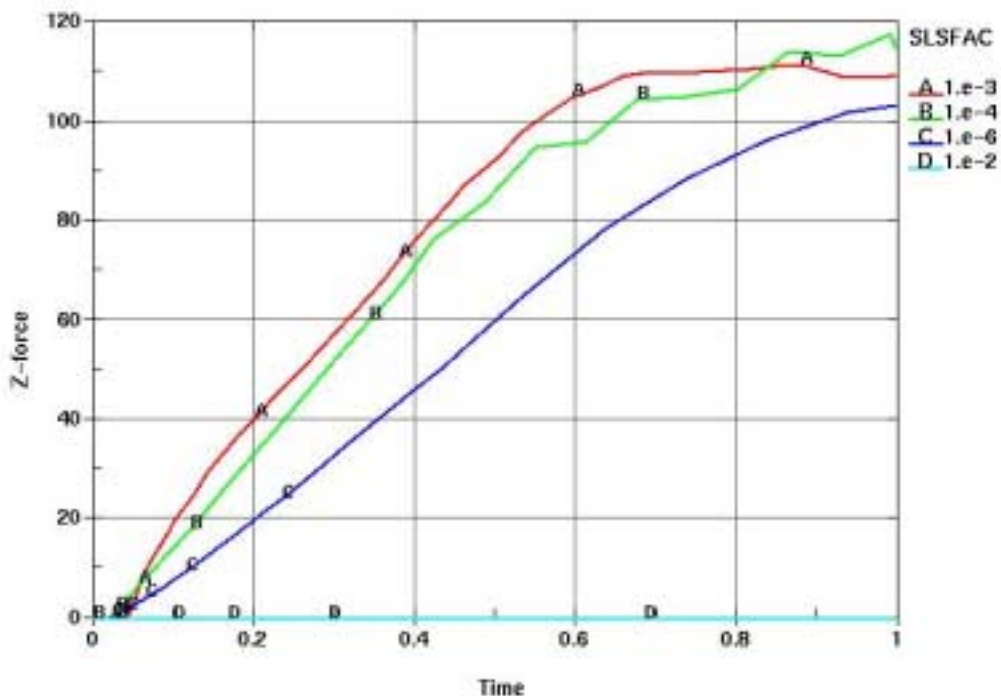


Figure 3
Excessive penetration with SLSFAC = 1.e-6



NEW FEATURES: Implicit Elements and Materials

The following is a list of element formulations and material models currently available for use with implicit mode in LS-DYNA version 960:

Brick element formulations:

1,2,10,15,18

Brick element materials:

1,2,3,4,5,6,7,9,12,13,18,20,24,26,30,49,57,63,91,92,103,104,106,115

Shell element formulations:

1,2,4,6,10,16,17,18,20

Shell element materials:

1,2,3,4,6,9,18,20,24,36,37,49,91,92,103,104,106,116,117,118

Beam element formulations:

1,2,3,4,5,6,7,8,9

Beam element materials:

1,3,4,6,9,18,20,24,100

2-D Solid materials (shell formulation 12, nip=4):

1,2,3,4,6,9,18,20,24,36,37,49,91,92,103,104,106,116,117,118

2-D Solid materials (shell formulation 13 or 15, nip=4):

1,2,3,4,5,6,7,9,12,13,18,20,24,26,57,63

Thick Shell materials (element formulation 2):

1,2,3,4,6,9,18,20,24,36,37,49,91,92,103,104,106,116,117,118

Thick Shell materials (element formulation 3):

1,2,3,4,5,6,7,9,12,13,18,20,24,26,30,49,57,63,91,92,103,104,106,115

Monthly Site Summary February, 2001:

Archived on the News Page is our previous News Letters in Adobe Acrobat PDF Format. We are using Adobe Version 4.0 to convert our news letter into PDF Format.

Event Notice:

- April 19th, 2001 there is a Workshop on FEM Simulation of Tube Hydroforming. It is being held at the Holiday Inn on Lane Avenue, Columbus OH. You are personally invited to attend by Dr. Taylan Altan, Director of The Ohio State Univ. Engineering Research Center for Net Shape Manufacturing.
- Our FEA Information participant Engineering Technology Associates (ETA) will be presenting “*Tube Hydroforming with LS-DYNA/DYNAFORM*” at 10:30 a.m.

Added to our Massively Parallel Computer Website:

- Information on SGITM OriginTM 3400, and SGITM OriginTM 3800.
- A publication: *Scalability of LS-DYNA on SGI Systems – By Guanqye Li* Reprinted with permission from SGI Developer News, Fourth Quarter, 2000 edition.

Showcased on our News Page (available on the News Page Archive) – I try to showcase one week of a participant, one week of a software product, one week of a hardware platform and a week of an article of interest. The News Page is weekly on all the sites.

- **Participant:** CAD-FEM: founded in 1985 by Gunter Muller CAD-FEM is located in Grafing near Munich.
- **Book Review:** Three Dimensional & Dynamic Analysis of Structures A Physical Approach with Emphasis on Earthquake Engineering by E.L. Wilson. Reviewed by Dr. David Benson.
- **Book Review:** Nonlinear Finite Element for Continua and Structures by Ted Belytschko, Wing Kam Liu and Brian Moran. Reviewed by Dr. David Benson.
- **Hardware:** The HP Netserver LP 2000r.
- **Software:** EASi,- developed by EASi Engineering.
- **Software:** DYNAFORM, developed by Engineering Technology Associates (ETA)

Publications Posted During the Month of February:

D3-4c	<i>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 Associates Ltd. – ARUP)</i>
N13-2c	<i>Adaptive Finite Element Simulation of Sheet Metal Forming Processes Using Gradient Based Indicators –Ramin Moshfegh, Xiandong Li, Larsgunnar Nilson (Linkoping Institute of Technology)</i>
K10-2c	<i>A validation Case Study: Steel Billet Drop Tests and Simulation as Reported in NUREG/CR-6608 – Leonard E. Schwer (Schwer Engineering & Consulting Services), James M. Kennedy (KBS2, Inc.)</i>

Marsha Victory
vic@lstc.com

FEA Information Commercial & Educational Participants

Livermore Software Technology:

Develops and supports LS-DYNA, LS-Post, LS-OPT. LS-DYNA, a highly advanced multi-physics simulation code. Implicit, Explicit, SMP and MPP imbedded in one code. Headquartered in Livermore, CA, USA

Engineering Technology Associates, Inc.

An engineering consulting company specializing in automotive Computer Aided Engineering (CAE). Providing services, software, training and technology. Headquartered in Troy, Michigan, USA.

Oasys, Ltd.

Markets engineering software products. A global organization of consulting engineers, planners and project managers working in all areas of the built environment. Headquartered in the UK

CADFEM:

Providing complete services in computer assisted analysis and design. Offering software, application support, training, consulting and development. Headquartered in Grafing near Munich.

Japanese Research Institute, Ltd.

Specializing in Research & Consulting; System Consulting, Frontier Business, System Integration and Science Consulting. JRI is located in Tokyo, Japan.

EASi Engineering:

A global provider of integrated solutions for Fast-to-Market product development. Easi Engineering is headquartered in Madison Heights, Michigan.

ANSYS, INC:

ANSYS, Inc. develops, markets, supports and delivers collaborative analysis optimization software tools. Headquartered at Southpointe in Canonsburg, Pennsylvania.

Hewlett-Packard:

A worldwide leader in personal computing, in such areas as mobile computing, network management, 3-D graphics and information storage. Headquartered in Cupertino, CA., USA.

SGI:

A worldwide leader in high-performance computing technology, from desktop workstations and servers to supercomputers, delivering advanced computing and 3D visualization. Headquartered in Mountain View, CA, USA.

Dr. Ted Belytschko:

Professor Ted Belytschko, W.P. Murphy Professor and Chair at Northwestern University in Chicago, Illinois.

Dr. David Benson:

Associate Professor, University of California, San Diego Research on computational methods for nonlinear, large deformation problems in solid mechanics.

Dr. Bhavin V. Mehta:

Dr. Mehta completed his doctoral work in the Ohio University Individualized Interdisciplinary Program involving chemical engineering, mechanical engineering, and mathematics.

Dr. Taylan Altan:

Professor, Mechanical Engineering - Director, ERC for Net Shape Manufacturing, The Ohio State University.

Professor Ala Tabiei:

Assistant Professor, University of Cincinnati, of Aerospace Engineering and Engineering Mechanics. Director, Center of Excellence in LS-DYNA.