FEA Information International News
For The
World-Wide Engineering Community

November 27, 2001

FEA Information Inc.
www.feainformation.com

Featured Articles

FEA Information International News:

Managing Editor
Trent Eggleston
Feature Editor
Marsha Victory
Technical Writer
David Benson
Art Director:
Wayne Mindle

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

Featured Articles (Contact Modeling by Suri Bala will continue in the December issue)

FEA Information Inc. ONLY uses the following e-mail addresses
vic@lstc.com
feainfo@lstc.com

FEA Information Inc. welcomes:

Flotrend Corporation located in Taipei, Taiwan as a Distributor Participant. Flotrend has expertise in training, consulting, and sales of LS-DYNA and other software.

Dr. Alexey I. Borokov, as an Educational Particpant. Dr. Borokov is the Head of Computational Mechanics Laboratory and Director of CAD/CAE/CAM Dept., St. Petersburg State Technical University, Russia.

THEORY: Shell Element Drilling DOF Constraints

By default, the current version 960 code employs the stiffness method to control drilling rotations in flat shell meshes. The default stiffness introduced in this method is approximately 10x smaller than that used in previous versions of LS-DYNA. This change was made to reduce the overly-stiff behavior seen in some shell models. The change was later found to cause convergence problems on several small, academic test problems which ran successfully using older versions. To recover the old behavior, parameters on the *CONTROL_IMPLICIT_SOLVER card can be modified. In the keyword example below, a drill stiffness scale factor of 1000 is selected, and the AUTOSPC feature is disabled:

```
*CONTROL_IMPLICIT_SOLVER
$   lsolvr   prntflg    negeig     order     DRILL     DPARM   AUTOSPC    ASPCTL
         0         0         0         0         1    1000.0         2         0
$
$ DRILL = shell drilling DOF constraint method
$    eq.1: stiffness method (default)
$    eq.2: constraint method
$    eq.3: none, unless AUTOSPC is active below
$
$ DPARM = drilling DOF parameter
$
$  if DRILL.eq.1 (stiffness method) then
$    DPARM = stiffness scale factor
$       for linear analysis, default =   1.
$       for nonlinear analysis, default = 100.
$       for eigenvalue analysis, dparm = 1.e-8 always
$
$  if DRILL.eq.2 (constraint method) then
$    DPARM = flatness tolerance in degrees (default = 10)
$
$ AUTOSPC = flag to activate AUTOSPC constraints
$    eq. 1: activate AUTOSPC
$    eq. 2: do NOT impose AUTOSPCs
                                        $
$ ASPCTL = AUTOSPC tolerance
$    for single precision, default = 1.e-4
$    for double precision, default = 1.e-8
```

PROGRAMMING - Seamless Springback

The “dyna.str” *structured format* input file now includes implicit data cards for all seamless springback runs, even if all input parameters are default. This change allows initialization of several new implicit features during the input phase, which was previously being skipped during seamless springback jobs, resulting in unpredictable behavior.

APPLICATIONS & BENCHMARKS – Implicit QA Testing

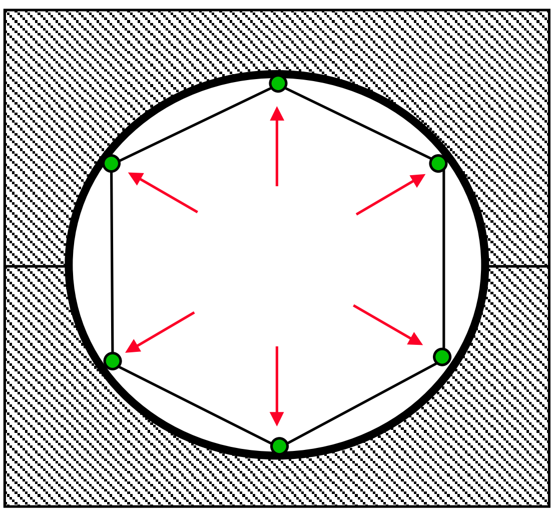
Roger Grimes has helped to assemble a suite of small test problems for QA testing of implicit features in LS-DYNA. The suite currently includes 83 test cases, and is being executed on all platforms.

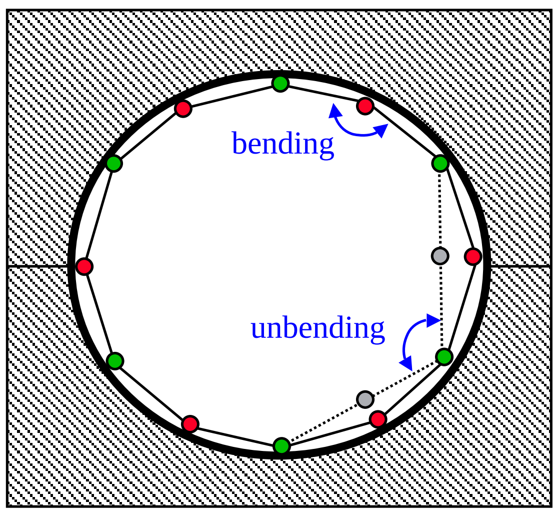
APPLICATIONS & BENCHMARKS – Tube Hydroforming

Tube hydroforming is becoming very popular for automotive applications. A recent LS-DYNA model, developed at the Dana Corporation, showed great promise for the implicit solver. The simple process simulated the forming of a rectangular section in the center of a thick wall tube. Load was applied by internal pressure with simultaneous displacement of the ends of the tube to feed material into the dies.

The explicit solution to this problem, performed under accelerated time scale, required 50 cpu hours. (It is likely that some additional mass scaling could have been applied without significantly changing the results.) The implicit simulation required less than 50 minutes of cpu time. Convergence behavior during this simulation was excellent, with an average of seven equilibrium iterations per time step. This experience suggests that the attributes of the tube hydroforming process - heavy gage material, extensive confinement by tooling, good boundary supports, and relatively small model sizes – may make these processes quite suitable for implicit analysis.

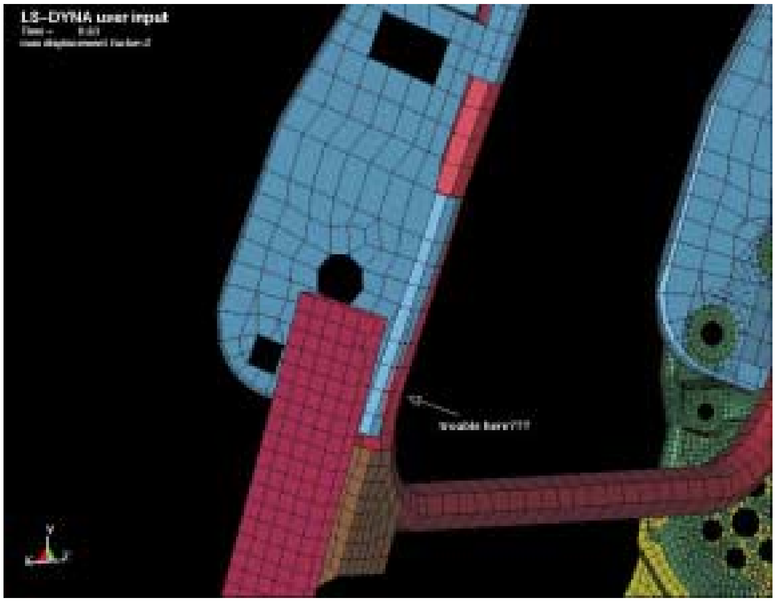
An unexpected error is inherent in the adaptive mesh refinement process when applied to circular tubes, using either the implicit or explicit methods. Since newly created midside nodes are not in contact with the tool, non-uniform deformation must occur to regain equilibrium after mesh refinement, as shown in the diagrams below:





APPLICATIONS & BENCHMARKS – Adding Text in LS-POST

(NO IMPLICIT CONTENT) Text annotations can be easily added to the screen using LS-POST. This makes a convenient method for highlighting particular features in a model. Arrows may be added which point from the text a particular location in the model, as shown in the next picture:



To add text, follow this procedure:

1. Select the ANNO(tation) button to bring up the workspace.
2. Enter your text message in the dialog box, followed by <cr>. The text will appear in the scroll box.
3. Select your text from the scroll box, pick the POSITION button, and click on the screen in the location where you would like the text to be positioned
4. To add an arrow to your text:
5. Pick the SELECT button, and select your text on the screen.
6. Pick the ARROW button, and drag the arrow FROM the text TO the object.

BUG FIXES in version 960

Implicit Adaptive Mesh: Curvature data was being updated during equilibrium iterations, rather than only at the final equilibrium geometry. This caused some extra mesh refinement in areas which did not end up being in contact with tooling.

WARNING: Default hourglass control type. When the *HOURGLASS keyword is used, and zero is entered for the hourglass type, LS-DYNA now ignores the global default value specified on the *CONTROL_HOURGLASS keyword, and instead uses IHG=1. This violates the general convention that *CONTROL_ keywords are used to specify global defaults, which are used when zeros are entered locally. However, the current implementation does follow a strict interpretation of the user's manual. This change has been in effect since version 950d.

Smoothed Particle Hydrodynamics – Part II

This second part of Smoothed Particle Hydrodynamics describes some aspects of numerical implementations in LS-DYNA, and the current capabilities of this solver.

I – Equations: Using SPH, the matter is represented by particles. Each particle has a fixed mass and follows the fluid motion. The equations of conservation governing the evolution of the fluid are expressed as inter-particle forces when written in the SPH form. Each particle carries mass m, velocity v, and other quantities specific to a given problem. The equations governing the evolution of mechanical variables are expressed as the summation of interpolants using a kernel function W with smoothing length h. For example, the equation of conservation of mass is:

$$\frac{d\rho}{dt} = -\rho div(v)$$

Which can be evaluated with the following SPH approximation:

$$\frac{d\rho}{dt}(\mathbf{x}_i) = \sum_{j=1}^{N} m_j (v(\mathbf{x}_j) - v(\mathbf{x}_i)) A_{ij}$$

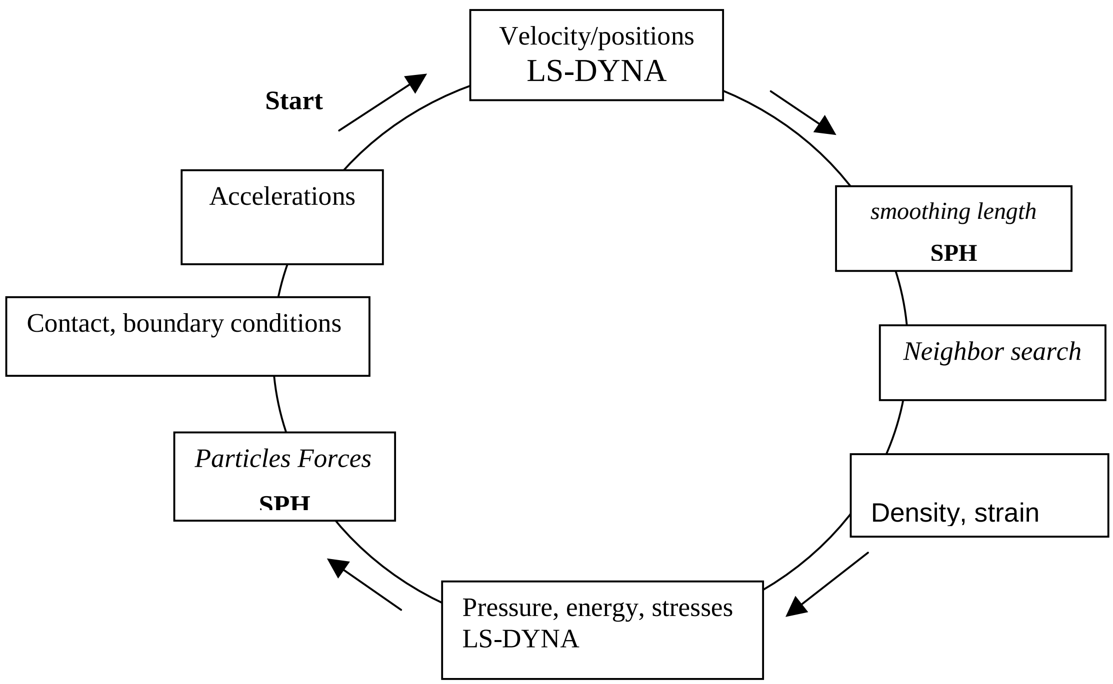
The SPH momentum equation may be written as:

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

And the energy equation is:

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

An integration cycle in time can be represented by the following loop: Figure 1 Loop of SPH Cycle



II – Neighbor search

The neighbor search is a very important step in the SPH computation. It is important to know at any time of the calculation which particle is going to interact with which other. The sphere of influence of each particle is a finite domain of a radius of $2h$. The goal of the neighbor search is to list the particle inside that domain at each time step. In a direct search for a set of N particles, the number of distance comparisons required is $N-1$. Since this comparison needs to be made for each particle, the total number of comparisons is $N(N-1)$ which can dominate the total CPU time for large models. The idea of the search for neighbors is to use the same algorithm as the one used for the contact search: the bucket sort. The domain covered by the particles is split in several boxes of a given size. Then for each particle, we search for neighbors inside the main box and also the neighbor boxes contained in the domain of influence of the given particle (figure 1). Then, when the list of hypothetical neighbors is known, we compute the distance between each couple of particles and check if it is smaller than twice the smoothing length. The total cost of the grouping operations and comparisons, needed to form the buckets, can be nearly linear with the number of particles N.

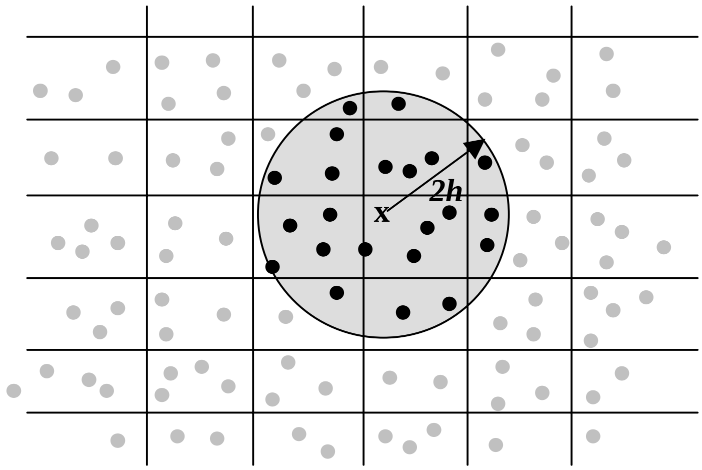


Figure 2 : Bucket sort and Neighbor search

III – Variable smoothing length:

In order to avoid problems due to compression and expansion of material, W. Benz developed at the end of the 80's an idea based on variable smoothing length. The idea is to keep enough particles in the neighborhood to validate the particle approximations of continuum variables. It allows h to vary in time and space. In expansion for example, a constant smoothing length can lead to numerical fracture, which means that no particle is located in the neighborhood of a given particle. In compression, a lot of particles will be located in the sphere of radius $2h$, which can significantly slow down the calculation.
The option of a variable smoothing length is the default in LS-DYNA. The equation of evolution is the following:

$$\frac{dh}{dt} = \frac{1}{3} h div(v)$$

It comes from the following idea: keeping the same number of particles in the neighborhood means keeping the same mass of particles in the neighborhood. The total mass M of n particles enclosed in a sphere of radius $2h$ is:

$$M = n * m = n * \rho * V = n * \rho * \frac{4}{3} \Pi 8h^3$$

and we want to keep that mass constant during the calculation. So, we derive in time and we have:

$$\frac{dM}{dt} = n * \frac{d\rho}{dt}\frac{32}{3}\Pi h^3 + n * \rho * \frac{32}{3}\Pi\frac{dh^3}{dt}$$

The left hand side is 0 because the mass is conserved. After simplification, this leads to the following equation:

$$\frac{dh}{dt} = \frac{1}{3} h div(v)$$

div(v) is the divergence of the flow. The smoothing length increases when particles separate from each other and decreases when the particles are condensed. It varies to keep about the same number of particles in the neighborhood during the time marching. For computation efficiency and numerical reasons, a minimum and maximum value are required for the smoothing length. Default minimum and maximum values are 0.2 and 2 times the initial smoothing length. These values apply to most problems. The smoothing length varies between the minimum and maximum values

$$HMIN * h_0 < h < HMAX * h_0$$

Where h_0 is the initial value of the smoothing length. Defining a value of 1 for HMIN and 1 for HMAX will result in a constant smoothing length in time and space.

IV – Material types

Currently the following material types are available for the SPH solver version LS960:

*MAT_ELASTIC
*MAT_PLASTIC_KINEMATIC
*MAT_SOIL_AND_FOAM
*MAT_HIGH_EXPLOSIVE_BURN
*MAT_NULL
*MAT_ELASTIC_PLASTIC_HYDRO
*MAT_STEINBERG
*MAT_JOHNSON_COOK
*MAT_PSEUDO_TENSOR
*MAT_PIECEWISE_LINEAR_PLASTICITY
*MAT_LAW_DENSITY_FOAM
*MAT_CRUSHABLE_FOAM

The definitions of these materials are the same as for brick elements. Other material definitions can be developed upon request since they are compatible with the SPH formalism.

V – Symmetry planes

In standard finite element methods, a symmetry plane is defined by applying nodal constraints on nodes placed on the boundary. However, this can not be done with SPH for two reasons: the first reason is that SPH is a meshfree method and particles are allowed to be mixed together. If a layer of particles is constrained, there is no reason for the neighbor particles not to pass through these particles and penetrate the boundary. The second reason is that a particle, which is close to the boundary will miss some particles in order to have equilibrium of pressure. This will lead to a decreasing pressure near that boundary. A special treatment needs to be developed to treat symmetry planes for SPH elements. The way we did this is to define a symmetry plane by creating a set of ghost particles. The ghost particles are the image of the particles close (within a distance of $2h$) to the boundary. For each particle close to the boundary, a ghost particle is automatically created by reflecting the particle itself. The ghost particle has the same mass, pressure, and absolute velocity than the real particle. The ghost particle is then in the list of neighbors of that particle and contributes to the particle approximation.

This capability is only available in LS970 beta version.

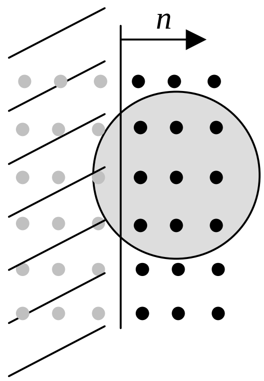


Figure 3 : Ghost particles reflecting real particles

VI – SPH outputs

An ASCII file can be generate by setting the appropriate keywords:

*DATABASE_HISTORY_SPH
*DATABASE_SPHOUT

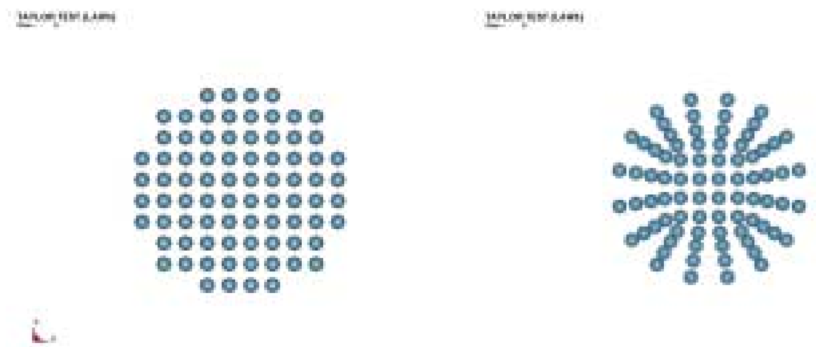
This will cause LS-DYNA to create a SPHOUT file. In this file, particle information can be found. This information is divided into two subset: the stress variables such as stresses, pressure, VonMises and the strain variables such as strain rates, smoothing length, density.

SPH particles can be visualized in the post-processor LS-POST. A SPH menu is available in the **setting** button. User can define the number of facets that draw the sphere of influence of the particles, as well as the scale factor on the smoothing length. All quantities available for brick elements are available for SPH elements: fringes of stresses, velocities, plastic strain, …

VII – SPH use

Initially, a set of particles is defined with two sets of properties: mechanical properties and geometrical properties. By mechanical properties we mean: the mass of the particle, density, constitutive law as defined in the ELEMENT_SPH and the PART cards. The geometrical properties concern the way particles are initially placed. Two discretization parameters are to be set by user: Δx_i lengths (distances between particles) and the CSLH coefficient which is defined in the SECTION_SPH card.

A proper SPH mesh must satisfy the following conditions: it must be as regular as possible and must not contain large discrepancies of the mass of particles.   For instance, if we consider a cylinder SPH mesh, we have at least two possibilities:



Mesh 1 Mesh 2

Figure 4 : Comparison of two possible models for a SPH analysis

The second mesh (Mesh 2) includes too many inter-particle distance discrepancies. Therefore the more uniform mesh (Mesh 1), is better for an SPH analysis. For one-dimensional models, all particles must be aligned in the x-direction. For two-dimensional problems, all particles must be defined in the X-Y plane.

SPH is a meshless method. One major difference between finite elements and particle methods is that we don't have any idea of the topology of the model with SPH. There is no connectivity between the particles, and before the first neighbor search, no one knows which particle is close to which other. During the initialization, we have to find that topology. So, we look for the initial smoothing length for each SPH part and this is what is done by LS-DYNA during the initialization phase. To compute the initial smoothing length, we search, for each particle, the smallest distance to another particle. Then, we take the largest distance of all minimums: this gives L, the maximum of the minimum distance between particles. This value is then scaled by the CSLH coefficient defined in the SECTION_SPH card. After this it is possible to reconstruct the topology of the mesh and to know the neighborhood of each particle.

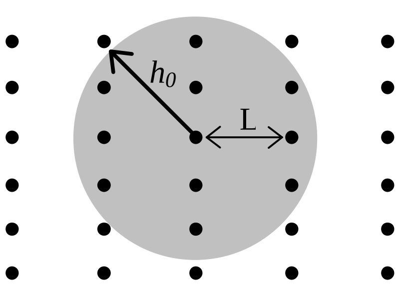


Figure 4 : Initial smoothing length computation

VII – Coupling with LS-DYNA standard elements

When two or more SPH materials are defined in the model, it is not necessary to define a contact between SPH parts. The contact between two SPH materials will occur when particles of the first part will be located in the sphere of influence of particles of other parts.

However in the model, if both SPH parts and shell or brick elements are defined, the user needs to define a contact between particles and other elements. The particles are considered as nodes. Therefore any NODES_TO_SURFACE contact can be used which means:

AUTOMATIC_NODES_TO_SURFACE
CONSTRAINT_NODES_TO_SURFACE
NODES_TO_SURFACE
TIED_NODES_TO_SURFACE

We recommend the use of the soft constraint formulation for the contact. This option is defined on the optional card A of the contact. With the soft constraint option, the interface stiffness is based on the nodal mass and the global time step of the calculation.

Dr. Jean Luc Lacome can be reached at jll@lstc.com

*** ***

FEA Information Educational Participants

Dr. T. Belytschko Northwestern U. USA	**Dr. D. Benson** UCSD USA	**Dr. Bhavin V. Mehta** Ohio University USA
Dr. Taylan Altan The Ohio State U. ERC/NSM USA	**Prof. Ala Tabiei** U. of Cincinnati USA	**Dr. Alexey I. Borovkov** St. Petersburg State Technical. University Russia

To limit the size of FEA Information News graphics will not be included in this series of course notes. The course notes with the graphics are on the website www.feapublications.com

Complete Course Contents:

Part 0: Introduction

History of numerical simulations For Crashworthiness:

- Explicit FE-codes were developed in the 60's and '70s at the DoE Labs in the USA
- All essential algorithms were available : explicit integration, shell element, contact, radial return
- First full vehicle car crash models built and analyzed in the mid '80s, industrial application rendered possible by the introduction of supercomputers (Cray)
- In Germany, a research project was sponsored by FAT to investigate the feasibility of car crash simulations, 2 models were selected : VW-Polo and BMW-300
- Rapid development in the next 10 years, today FE-crash simulation is a fully integrated tool in vehicle design, no competitive development is possible without intensive CAE support

Impact of CAE in crashworthiness engineering :

Evolution of CAE in crashworthiness :

1985 :	• pure research activity • 1 legal safety criterion must be met • about 100 prototypes needed per newly developed carline
1995 :	• CAE part of design process • 4 legal criteria + many in-house criteria must be met • design cycles shortened • average number of prototypes per newly developed carline still around 100

Potential of CAE in crashworthiness :

• avoid building prototypes that fail by virtually optimizing the structure using trend predictions • quickly find the structural fix if problems occur in a test
• reduce the number of development prototypes • allow for more verification & certification testing
• certification needs to be done by testing hardware because of : • legal requirements • limited capacity of numerical models for absolute predictions

Cost and gain of CAE in Crashworthiness engineering:

Example of Ford-Mondeo

Potential savings in testing and prototype hardware :

- about 150 prototypes crashed in Europe and USA
- development cycle of 5 years : 30 prototypes/year
- average prototype cost at least 250000.$
- by conservative estimates, 30% of prototypes can be saved through the use of simulations
- roughly 10 prototypes or $2.5M per year and per carline
- if we need about 20 simulations to eliminate a single protoype, then 200 simulations are needed per year and per carline
- at a rate of 50 CPU hours per simulation, a total of 10000 supercomputer CPU hours are required per carline and per year
- this is equivalent to occupying a single processor year round or a 2 processor machine half time
- the yearly cost is certainly covered by savings in hardware
- in any case, CAE is the only way since numerous regulations create a demand beyond the capacity of the safety testing labs

How could industrial CAE grow so fast ?

- need and opportunity
- need for simulations was regulation driven
- opportunity to deliver results was created by rapid development in hardware and software since 1985, and by buildup of hardware resources in the automotive companies
- software development was inevitably performance driven
- The number of loadcases has increased dramatically :

  (Graphic is not displayed for size reasons)
- Approximate evolution of computer time needed to run 100 milliseconds of crash simulation on a very small vehicle model (8000 elements)

1983	VAX-750 - Single scalar processor	2000 hours
1984	Cray-1 - Single scalar processing	100 hours
1985	Cray-1 - Single vectorized Processing	25 hours
1986	Cray-1 - Single vectorized processing, vectorized gather-scatter	10-12 hours
1998	Cray-T90 - Single processor	30 minutes

- Supercomputing made the difference between research and industrial application
- Car companies have built up considerable computing power, motivated for over 60% by crash analysis :

Ford's supercomputing resources grew by a factor of 500 in about 10 years (1988-1998)

Are we loosing the battle ?

- the need for higher reliability in the simulation work resulted in a continuous increase of size and complexity of the numerical models
- consequently the need for CPU resources has increased dramatically
- Review from Mercedes-Benz :

year	Size (elements)	CPU-time (hours)	Cray
1988	8-10000	5-10	XMP
1990	15-20000	10-20	YMP
1992	30-40000	20-30	YMP
1994	60-80000	30-40	C90
1998	160-180000	60-80	T90
2000	400-500000	120-160	SX-4/5

- Overnight runs are still possible but require parallel calculations
- The potential of parallel computing is remarkable, time needed to run 120 milliseconds of crash analysis on a 500000 element vehicle model :

8 processor Intel	MPP	
8 processor Origin	MPP	
4 processor SX-4	SMP	24 hours
4 processor COMPAQ	SMP	48 hours
16 processor COMPAQ	MPP	12 hours
32 processor COMPAQ	MPP	6 hours

The problem of Reliability :

- Difference between test and calculation results have many different causes, first in the material description of body panels :
- theory and numerical implementation (radial return) of plasticity for mild steel and other metals
- hardening and rate dependency parameters for mild steel and other metals
- initial stresses are difficult to account for when using explicit integration

Material properties for steel sheet :

- Variability is introduced by :
    - Initial anisotropy
    - Forming process (stamping, hydroforming...)
    - Thermal treatment (?)
- Material properties exhibit :
    - Non-homogeneous thickness
    - Initial stresses
    - Initial plastic strain, thus non-homogeneous yield stress

- In LSDYNA v960, these effects can partially be taken into account :
- mesh-independent projection of the results of forming analysis upon the corresponding panel in the crashworthiness model

The problem of Reliability :

- Difference between test and calculation results have many different causes, some are easily solved by mesh refinement :
- numerical methods :
  - use of underintegrated shell elements
  - and penalty based contact algorithms
- too coarse meshes allow only low curvature buckling modes

Required mesh density for the simulation of buckling problems :

- Convergence studies were performed very early
- a good reference is : Mats Larsson, 1989, An assessment of four different shell elements implemented in DYNA3D, a FEM-code used for transient nonlinear dynamic analysis of three-dimensional structures.
- The buckling of a quadratic beam was examined using 6 to 40 shell elements per side
- Mesh convergence can be checked by refining the mesh and comparing results or by judging the smoothness of the deformed mesh
- The study shows 16 elements per side to be adequate to describe the physical behaviour with sufficient accuracy, 12 elements per side gives an approximate solution

The problem of Reliability : (continued)

- Difference between test and calculation results have many different causes :
- functioning (initial stresses) and rupture of connections :
  - bolts
  - screws
  - spotwelds
  - glue
  - etc...
- behaviour of non-steel components
- realistic friction models are missing

Different ways of modeling spotwelds :

For crashworthiness applications, we always model each individual flange with real flange thickness, each spotweld is taken into account as an element

Node-to-node connections :

- nodal rigid body constraints
- welded node sets
- spotweld elements
  - discrete beams

Element-to-element connections :

- beam element type 9, material type 100
- sliding interface type 7 (shell_edge_tied_to shell_surface)

Comparison of both models :

Node-to-node	**Element-to-element**
Spotweld element has no spatial dimension	Spotweld element has a finite size
Allows spreading of flanges	Allows spreading of flanges
Transmits forces and moments between flanges	Transmits forces and moments between flanges
Rotational stiffness is mostly infinite	Material rotational stiffness
Failure possible	Failure possible
No free arrangement of spotwelds, meshes on flanges must match	Free arrangement of spotwelds, flanges can be meshed independently
May promote hourglassing	Does not promote hourglassing
For certain options will not influence the timestep	Beam elements usually influence the timestep
2 elements in the flange allow central positioning of the weld	3 elements in the flange needed to position the weld centrally
Element size influence ?	Element size should be about equal to the weld nugget diameter

More information is in : A.K. Volz, Spot weld modeling for crash computations, requirements and new functionalities in LS-DYNA version 940, 5th international LS-DYNA Users Conference, Southfield, 1998h

- Improved spotweld modeling :
- contact type s_7 to consider torsional stiffness in the weld
- modeling by brick elements (v960)

Modeling of bolts and rubber bushings :

- Subframe and engine mounts should no longer be modeled by single spring or joint elements since this can lead to considerable error in relative rotations between the connected parts
- concentric cylinders with contact or brick elements modeling the rubber are far more accurate
- This can affect the accelerations at B-pillar level

The problem of Reliability : (continued)

- Difference between test and calculation results have many different causes :
  - behaviour of non-steel components
  - realistic friction models are missing

Mass Distribution in vehicle models :

Assume a typical small vehicle :

Total vehicle mass	**1200.kg**	
Car body	200.kg	100000 elts 100000 nodes 2 gram/node
powertrain	200.kg	meshed
Chassis & wheels	200.kg	meshed
components	600.kg	Added mass For 600 masspoints : 1.kg/node

The non-structural components are modeled with a precision that is 2 orders of magnitude lower then the car body itself

Mass distribution in vehicle model : continued :

Additional considerations are excentricity (rotational inertia) of the components and the stiffness of the connection with the car body

A careful consideration of components (other then powertrain) that move relative to the car body is important:

- exhaust system
- fuel tank
- seats
- dummies
- etc...

Fixing components to the car body will lead to an underestimation of acceleration peaks early in the analysis, this can be critical for the investigation of airbag sensor response but usually not for determining occupant kinematics

The problem of reliability :

- Major stumbling block for predictive simulations today is in the material modeling :
- structural use of non-steel materials :
    - lack of test data
    - lack of suitable material models (plastics, foams...)
    - discontinuous cell structures must be approximated as continuous materials (foam, honeycomb...)
    - inhomogeneous composites must be approximated as continuous materials (woodstock, lignotok...)
- lack of models to predict (brittle) rupture (aluminium, magnesium, fiber reinforced composites...) and crack propagation, this is a major problem in castings (engine mounts...)

Trends in crashworthiness simulations :

- The decision to build a prototype is often taken on the basis of a simulation result
- reliability then takes priority over runtime
- reliability is improved by :

** finer meshes
** better quality meshes
** improved numerical algorithms

Size of numerical models :

- full vehicle state-of-the-art model size grew from 10000 to 50000 elements in about 5 years and to 150000 elements in the next 5 years
- the reason is the desire for more reliability and accuracy since :

** finer meshes allow capturing higher curvature buckling modes

** more components can be modeled in detail, including their connection to the car body
** hourglass and contact energy problems are avoided by increasing mesh density

- uniform meshes require fewer assumptions about the response (buckling mode) of the structure
- the tendency is towards a single homogeneous model for all 3 major load cases (front, rear & side)
- fewer restrictions on model size has advantages from a mesh generation point of view : automatic meshing and easy integration of component models because of mesh compatibility

Quality of numerical models :

- car body geometry (CAD) is smoothed to allow an element size of about 5mm, this way reasonable timestep values are obtained initially without mass scaling
- mesh density must allow smooth representation of the deformed geometry
- mesh density must allow for sufficient resolution in terms of integration points : static stress analysis mesh is usually too coarse
- limit number of triangular elements
- regular meshes in order to represent the wave propagation problem with minimal dispersion
- limit warping in BT (Belytschko-Tsay) shell and avoid zones meshed predominantly with warped elements
- mesh every sheet in it's own neutral plane, do not distort flanges and model spotwelds with special purpose elements
- avoid initial penetrations at all cost : induced stresses may exceed the yield stress

example :
penetration caused by non-homogeneous mesh, distance between slave node and master segment is less then the sum of the half sheet thicknesses :

in thin steel sheets, initial penetrations of 0.1mm or more can cause stresses that exceed the yield stress : a plastic hinge will result

contours of plastic strain should be checked after 0.5 of 1 milliseconds, before the structure impacts

automatic offset of slave nodes will not be failsafe if more then 2 sheets are involved

- avoid mesh tangling at all cost : instabilities may be generated
  If tangling exceeds the contact thickness, no warning will be printed to the d3hsp file

  Non-physical connections are generated, entire panel assemblies could be suspended upon very few nodes and high nodal contact forces will occur

  Tangling can occur during the simulation if edge-to-edge penetrations are not prevented

Major Advances in Software :

- Element Technology :

The traditional BT element has the merit of computational efficiency and robustness, deficiencies are :

- underintegration (possible occurence of zero-energy or hourglass modes)
- corotational formulation (limited to small shear deformations)
- plate element formulation : no coupling of membrane and bending strains

Affordable shell elements are now available that correct several deficiencies of the BT element

- Contact algorithms :

  Classical contact algorithms are node-to-segment and
  Do not check for edge penetrations. Contact type 26 allows treatment of contact between beams and free shell element edges :

Currently, only free edges are treated.

- Material Models :

The number of materials used in cars seems infinite (rubbers, foams, plastics, fiber reinforced plastics...)

much more validation testing is needed

development of material models becomes more and more the responsibility of the application analyst

development of user subroutines becomes more frequent

Conclusions :

Between 1985 and 2000 :

- number of test scenarios rose from 1 to nearly 20
- design cycles have shortened
- prototypes remain expensive
- testing capacity is limited
- cpu-time decreases in price

Thus : CAE needs to deliver fast and reliable results - To achieve this :

- Model size increases
- Model quality increases
- Approximate solutions in software become less acceptable
- Databases of standardized component models are built
- Large scale simulations are performed by automotive companies and suppliers
- Need for computer resources increases, MPP solutions become more important

Timestep Control in LSDYNA :

1: Conditional Stability :

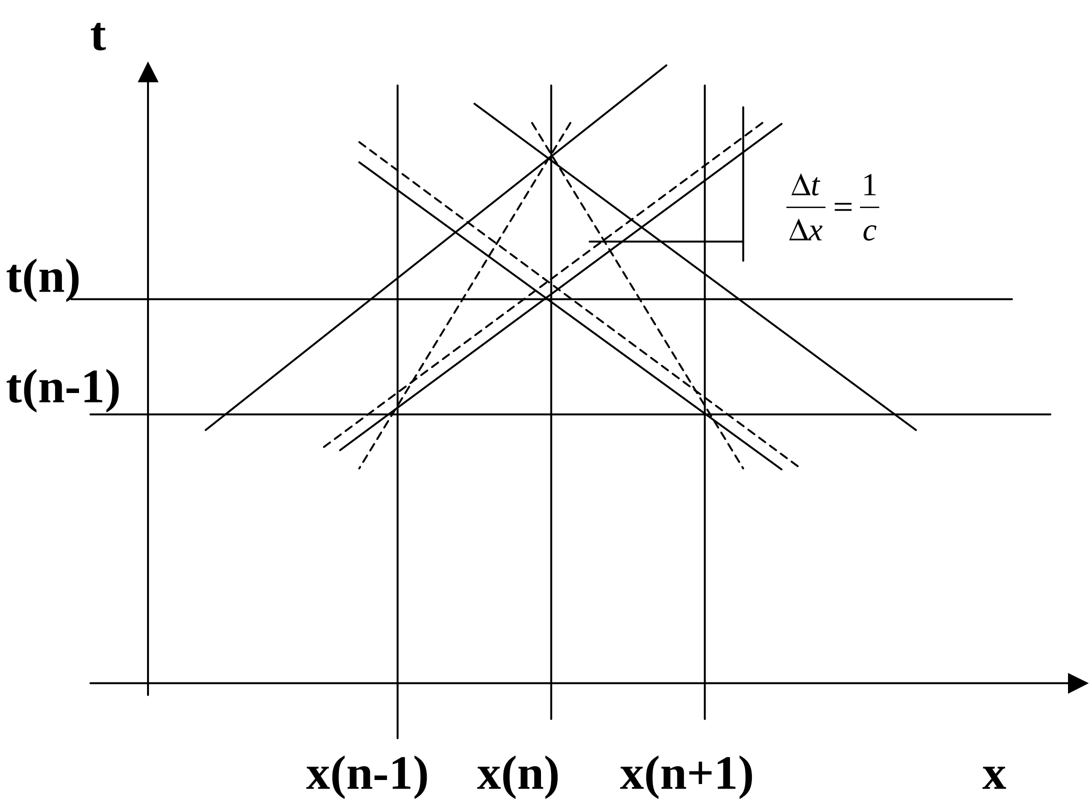
- The timestep of an explicit analysis is determined as the minimum stable timestep in any deformable finite element in the mesh

- In general this is determined by the so-called CFL-condition (Courant-Friedrichs-Lewy) that determines the stable timestep in an element as characteristic length divided by the acoustic wavespeed :

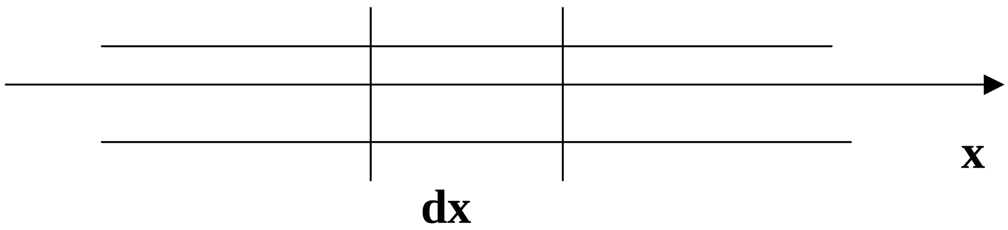
$$\Delta t = \frac{l_c}{c}$$

The CFL condition thus requires the numerical timestep to be smaller then the time needed by the physical wave to cross the element.

- The physical stresswave propagates with the speed of sound c
- The numerical stresswave propagates one element per timestep
- Consider a one-dimensional pressure wave propagation in a rod, using a constant spatial increment :



- A stability proof of explicit integration methods that mathematically leads to the CFL condition is only possible for linear problems
- The acoustic wavespeed for a 1-dimensional (truss) element is trivially computed :



- For a truss with a unit section Newton's second law leads to the familiar wave equation :

$$\rho dx \frac{\partial x^2}{\partial t^2} = \frac{\partial \sigma_{xx}}{\partial x} dx \approx \frac{\partial \sigma_{xx}}{\partial x_0} dx$$

$$u = x - x_0$$

$$\varepsilon_{xx} = \frac{\partial u}{\partial x_0}$$

$$\rho \frac{\partial x^2}{\partial t^2} = E \frac{\partial \varepsilon_{xx}}{\partial x_0} = E \frac{\partial u^2}{\partial x_0{}^2}$$

$$\boxed{\begin{aligned} &\frac{\partial x^2}{\partial t^2} = \frac{E}{\rho} \frac{\partial x^2}{\partial x_0{}^2} \\ &c = \sqrt{\frac{E}{\rho}} \end{aligned}}$$

- The previous derivation assumes a free transversal deformation (Poisson effect) in both directions for the truss element
- A shell element can be viewed as a number of truss elements placed next to each other, this geometry will confine the Poisson effect in the plane of the shell during longitudinal compression, thus increasing the longitudinal stiffness as well as the wavespeed which becomes :

$$c = \sqrt{\frac{E}{\rho\left(1-\nu^2\right)}}$$

- The critical timestep for shells is determined by the wave propagation speed for membrane deformations (rotational inertia is adjusted in order to sufficiently lower the bending frequencies)
- If the material Poisson coefficient is 0.3, the critical timestep in a shell will be 0.954 times the critical timestep in a truss with the same characteristic length.
- In brick elements, one can consider the Poisson effect to be confined in both transversal directions during longitudinal compression, consequently the membrane stiffness and the acoustic wavespeed will increase more dramatically :
- The acoustic wavespeed in 3D media becomes :

$$c = \sqrt{\frac{K + \frac{4}{3}G}{\rho}} = \sqrt{\frac{E(1-\nu)}{\rho(1+\nu)(1-2\nu)}}$$

- For a material with a Poisson coefficient of 0.3, a brick element will show a critical timestep that is 0.862 times the critical timestep in a rod with the same characteristic length

- Clearly the critical timestep for a brick element is zero if the material is incompressible

$$\nu = 0.5 \Rightarrow K = \frac{E}{3(1-2\nu)} \rightarrow \infty \Rightarrow \Delta t = 0.$$

- The characteristic length of trusses and Hughes-Liu beams is estimated as the element length :

$$l_c = L$$

- The characteristic length of brick elements is estimated as element volume over the maximal side area for 8-node hexagonals :

$$l_c = \frac{V}{\max(A_1, A_2, A_3, A_4, A_5, A_6)}$$

- The characteristic length of a shell element is estimated by default as the area divided by the maximal side length :

$$l_c = \frac{A}{\max\left(L_1, L_2, L_3, L_4\right)}(4 - node)$$
$$l_c = \frac{2A}{\max\left(L_1, L_2, L_3\right)}(3 - node)$$

- Alternatively the characteristic length can be estimated in a more conservative way as area divided by maximal diagonal by setting ISDO=1 :
(CONTROL_TIMESTEP)

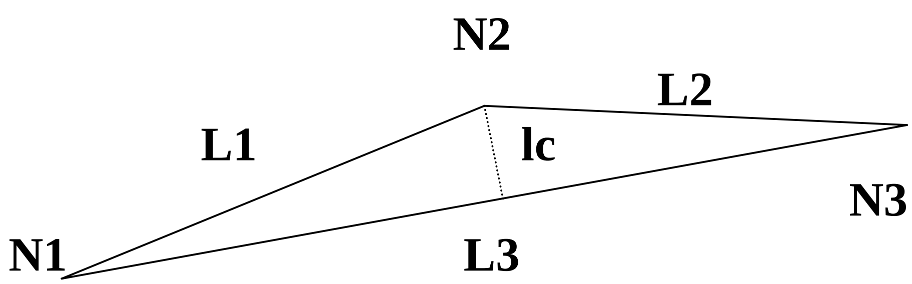
$$l_c = \frac{A}{\max\left(D_1, D_2\right)}(4 - node)$$
$$l_c = \frac{2A}{\max\left(D_1, D_2\right)}(3 - node)$$

- Alternatively the characteristic length can be estimated to optimize performance as the maximum of area divided by maximal sidelength and minimum sidelength, by setting ISDO=2 :

$$l_c = \max[\frac{A}{\max(L_1, L_2, L_3, L_4)}, \min(L_1, L_2, L_3, L_4)](4-node)$$

$$l_c = \max[\frac{2A}{\max(L_1, L_2, L_3)}, \min(L_1, L_2, L_3)](3-node)$$

- This will give a large timestep, in particular for shallow triangles, but can lead to instabilities since the characteristic length should be interpreted as the shortest path through the element (from any node to the opposing side) and in shallow triangles this does not correspond to the minimum sidelength :



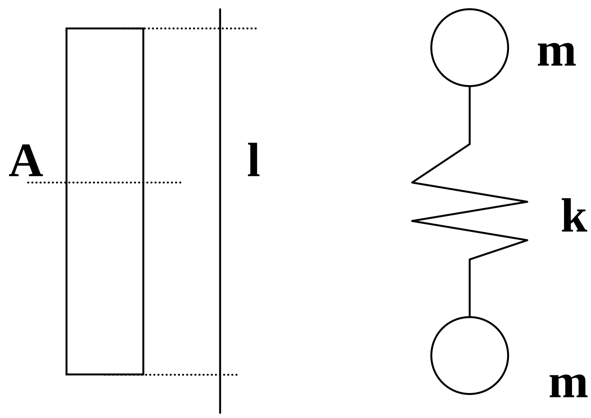
- Consequently it is dangerous to increase the timestep for badly conditioned triangular elements by setting ISDO=2
- The altitude of the triangle should be the basis of the calculation
- If the shortest side is used, the results of the simulation must be carefully checked for local instabilities
- If this works well or not will mostly depend upon the dimensions of the neighbouring elements
- Spring elements require a slightly different treatment since they can be defined on nodes with identical coordinates
- The equivalent of the CFL condition for discretized continua formulates the critical timestep in function of the spring stiffness and the nodal masses
- For a spring with longitudinal stiffness k connecting 2 masses m1 and m2 we obtain :

$$\Delta t = \sqrt{\frac{4m_1 m_2}{k(m_1 + m_2)}}$$

- In the code the nodal spring masses are approximated as half the actual nodal masses
- For a spring with longitudinal stiffness k and two equal nodal masses m we obtain :

$$\Delta t = \sqrt{\frac{2m}{k}}$$

- This can be easily seen to be the equivalent of the CFL condition applied to the equivalent truss element if the spring length is non-zero :



- Indeed :

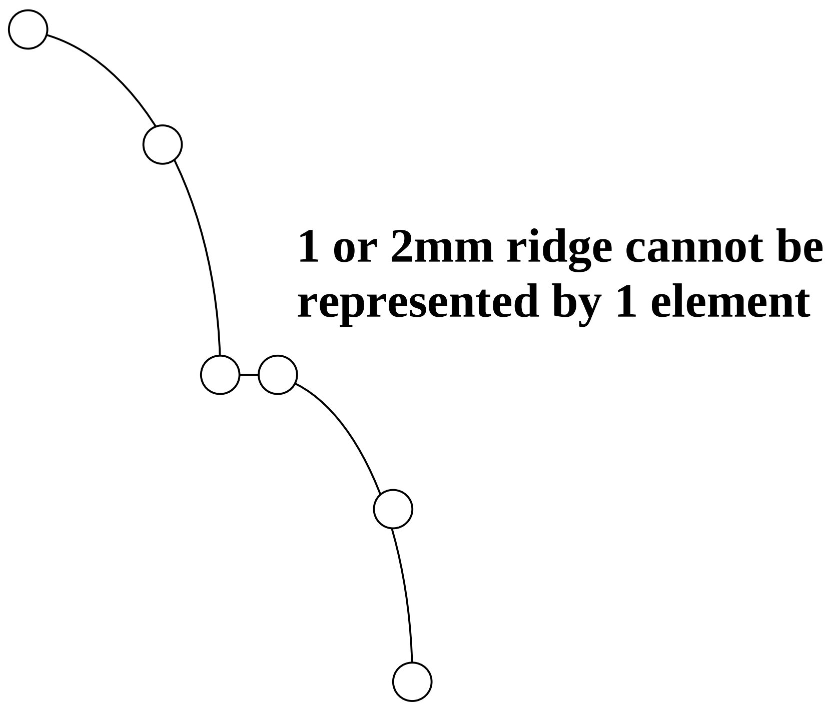
$$m = \frac{\rho A l}{2}$$

$$k = \frac{EA}{l}$$

$$\Delta t = \sqrt{\frac{2m}{k}} = \sqrt{\frac{l^2 \rho}{E}} = \frac{l_c}{c}$$

- The zero-length spring element has a finite timestep because it also has a finite stiffness, this is not the case for a (physical) truss

- The acoustic wavespeed for the 2 most common automotive metals is :
  Steel : 5240m/s
  Aluminium : 5328m/s
- In both materials, an element characteristic length of 5.mm will lead to a timestep of 1 microsecond, frequently set as a standard in the industry
- Since a reasonable timestep leads to a minimum element sidelength of about 5.mm, car body geometries can usually not be respected completely by the FE mesh :



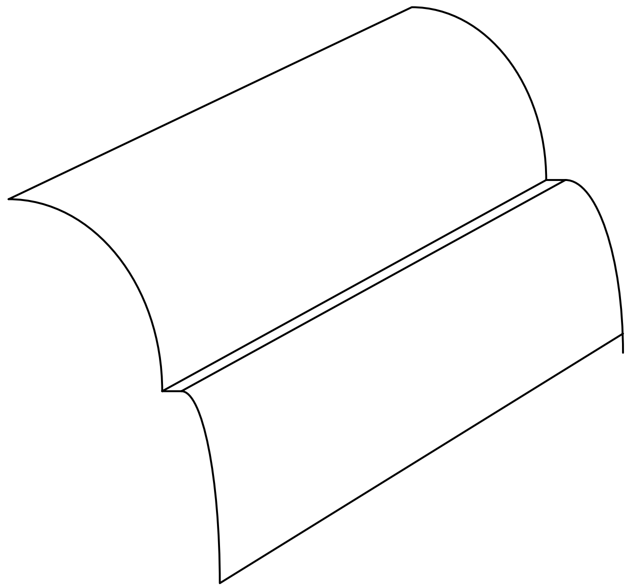
- This is the most serious limitation of vehicle models today : the required minimum element size often prevents geometrically detailed and/or sufficiently dense meshing
- LS-DYNA allows improving the timestep for any mesh by adding mass to the critical elements, this is done through mass scaling by specifying a minimum timestep for any of two parameters on the CONTROL_TIMESTEP card :
  MS1ST : mass scaling during first cycle only
  DT2MS : mass scaling repeated every timestep
- The amount of added mass is monitored, (on the d3hsp file), however this should be avoided since due to the nature of car body geometries structural errors can be introduced into the model that will change the dynamic behaviour of the structure

  * Global mass error :

  $\Delta M/M$ =
  total mass added by scaling / initial model mass

- Consider the example of a simplified wheelhouse with a 1.mm ridge that was modeled using shell elements :

(the corresponding unscaled timestep would be 0.2e-6 seconds)



- In order to achieve a standard 1e-6 seconds timestep we would mass scale as follows :

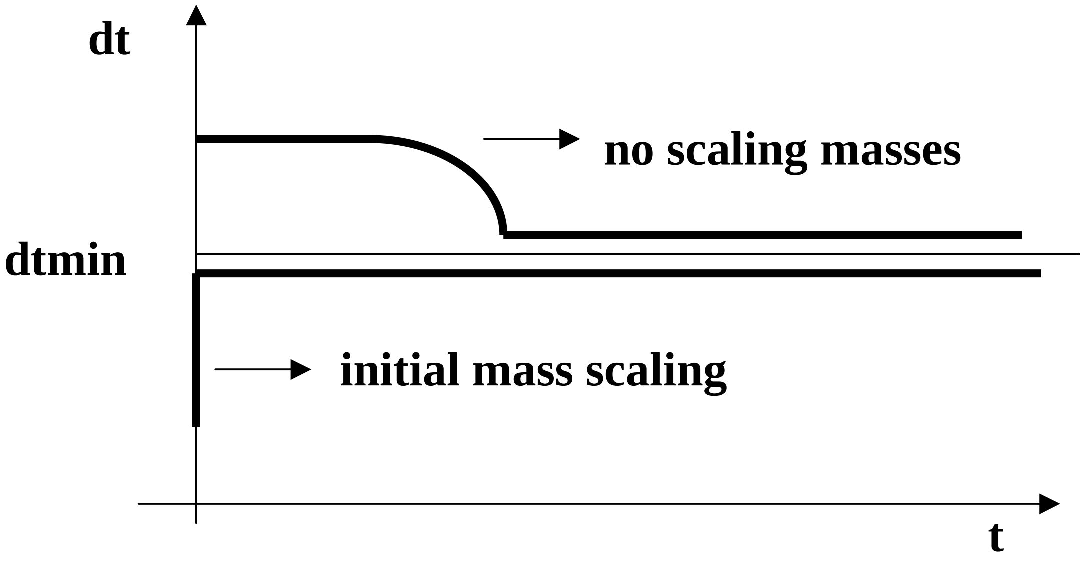
$$\rho_s = 25\rho$$

$$\Delta m = 1.\times 500.\times 1.\times 24\rho = 12000\rho \approx 96 gram$$

$$\frac{\Delta m}{M} \approx \frac{0.096}{1500.} \approx 0.00006$$

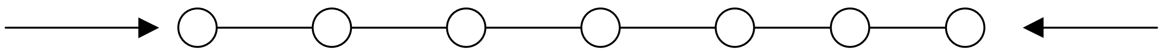
$$\frac{\Delta m}{M_{PID}} \approx \frac{96}{1200} \approx 0.09$$

- Thus small global mass changes may be caused by important and structural local mass changes
- The mass scaling option can however be extremely useful to prevent the timestep from dropping during the analysis (option DT2MS)

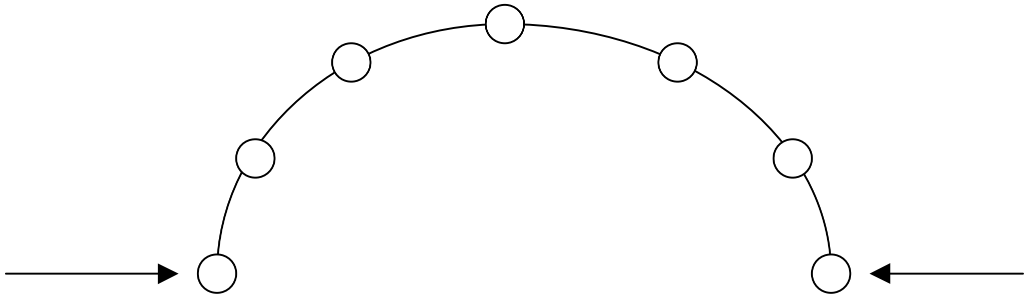


- Indeed a badly deforming shell element used to model a mild steel sheet usually corresponds to a local instability or an hourglass mode, this means the simulation is locally already highly inaccurate and the element would bring the simulation to a virtual standstill if nothing is done
- During the crash of a vehicle made primarily out of thin mild steel sheets, no finite membrane compression strains occur, rather the metal folds causing high curvatures but small deformations and almost no change in (characteristic) length
- Crashworthiness analysis is thus a large rotation but small deformation problem

Undeformed thin sheet :



Deformed configuration : (small compressive strains)



- If the mesh is capable of reproducing this behaviour, then the analysis timestep will necessarily be constant ; this will be the case if the mesh is fine enough to allow a smooth representation of the deformed vehicle geometry (no high curvatures within a single element)

- A reliable crashworthiness analysis can be recognized by either of the following 2 statements :
- The mesh smoothly represents the deformed structure
- The timestep in the shell elements that represent the sheet metal parts is constant without mass scaling

- To account for nonlinear effects upon structural frequencies the CFL determined timestep is multiplied by a safety factor (default value 0.9) in LS-DYNA, the default can be changed by setting the TSSFAC variable on the CONTROL_TIMESTEP card

- Particularly if brick elements are used with highly nonlinear material laws , TSSFAC=0.66 often improves the stability of the simulation

Workshop on acoustic wave propagation :

Input files :

- deltat1000.k
- deltat550.k
- deltat510.k
- deltat505.k

For the base load case :

- verify wave reflection
  - free surfaces stress ?
  - free surface velocity ?
  - particle velocity ?
- determine the wavespeed
  - from the input data
  - from the pressure time history results
- determine load wavelength
  - from the load period (=2ms) and wavespeed
  - from the pressure contour plots

For the other load cases :

- element size / load wavelength ?
- timestep / stability limit ?
- which is critical ?

We will have more in the coming months from the Course Notes due to be released April 2002 as a softbound edition from Livermore Software Technology Corporation.

Events: if you have an event you would like to list contact mv@feainformation.com

2002	Company	Event
April 08-10 France	**MSC**	Worldwide Aerospace Conference & Technology Showcase, Toulouse, France
April 22-24 USA	**ANSYS Inc.**	ANSYS Users Conference & Exhibition 2002 - Pittsburgh Hilton, Pittsburgh, PA.
May 19-21, USA	**LSTC** **ETA**	7th International LS-DYNA User's Conference at the Hyatt Regency Hotel & Conference Center - Fairlane Town Center, Dearborn, MI 48126
Oct. 9-11 Germany	**CAD-FEM**	CAD-FEM Users Meeting - International Congress on FEM Technology: Kultur- und Congress-Centrum "Graf-Zeppelin-Haus", Friedrichshafen, Lake Constance, Germany. For information contact Barbara Leichtenstern. Information will be available soon on the CAD-FEM website

FEA Information contributing sponsors
dedicated to worldwide engineering information, sales, training, consulting

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

FEA Information News Showcased in October
Archived on the site on the News Page through November

Date	Co. Names Are Abbreviated	Type	Showcased
Oct. 1	SGI ANSYS LSTC	Hardware Software Distributor	Onyx® 3000 ANSYS/Structural™ Located in the USA
Oct. 8	HP OASYS KOSTECH	Hardware Application Distributor	hp workstation j6700 'Designs on Your Bra' Located in Korea
Oct. 15	FUJITSU ETA GISSETA	Hardware Software Distributor	PRIMERGY N4000 FEMB Located in India
Oct 22	EASi Eng. LMS Int'l DYNALIS	Software Software Distributor	EASi-CRASH® LMS Virtual.Lab Located in France
Oct 29	LSTC ETA STRELA RUSSIA	Software Software Distributor	LS-DYNA eta/FEMB27-PC. **Located in Russia.** the official LSTC distributor of LS-DYNA for Russia and the CIS www.ls-dynarussia.com.

Weekly Publications To Be Showcased in pdf format on the December News Page

Crashworthiness Analysis of a Lock Gate Impacted by Three Different River Ships
Herve Le Sourne*, Jean-Clause Rodel** - Stephane Paboeuf* Guy Babaud* *French Shipbuilding Research Institute – Nantes; ** ECMT – Lyon

Optimisation and robustness of side airbag design and analysis
Tayeb Zeguer, Jaguar Cars.

Modeling The Shaken Baby Syndrome
I.C. Howard, E.A. Patterson, J.Langley, Structural Integrity Research Institute of the University of Sheffield (SIRIUS)

Characterization and Component Level Correlation of Energy Absorbing (EA) Polyurethane Foams (PU) using LS-DYNA material Models
Babushankar Sambamoorthy, Lear Corporation, Ford Division

Hydroplaning Simulation Using Fluid-Structure Interaction in LS-DYNA
Masataka Koishi, Toshihiko Okano, The Yokohama Rubber Co., Ltd. – Lars Olovsson, LSTC - Hideo Saito and Mitsuhiro Makino, Fujitsu Ltd.

Livermore Software Technology Corporation
[www.ls-dyna.com]
[www.lstc.com]

For More Information on Availability

Contact: Wayne L. Mindle
wlm@lstc.com

The Japan Research Institute Limited
www.jri.co.jp

Office Information

New Osaka Office

Engineering Technology Division
The Japan Research Institute, LTD.
Ginsen Shinsaibashi Bldg. 12F
Minami-Senba 3-10-19
Chuo-ku, Osaka, 542-0081 Japan

Phone: +81 (0)6-6243-5001

The Japan Research Institute Limited
www.jri.co.jp

Office Information

Tokyo Office:
Kudan Bldg.
1-5-3, Kudan Minami,
Chiyoda-ku, Tokyo, 102-0074 Japan

Phone: +81-(0) 3-3515-1020

Nagoya Office
Tomita Bldg.
1F, 2-5, Ushijima-cho
Nishi-ku, Nagoya-city 451-0046 Japan

Phone: +81-(0)52-586-9617

Product names referred herein are trademarks of their respective companies