Advanced Structural Dynamics Analysais by LS-DYNA3D Code

Mohamed Abdalla Almherigh*

Lecture Department of civil Engineering, Faculty of Engineering - Tripoli University

Tripoli-Libya

Email: malmherigh@gmail.com

Abstract

The good features included in the explicit 'LS-DYNA3D' code such as ALE, non-reflecting boundaries, constraint joints, contacts, multi-materials, initial volume fraction or geometry features and the flexibility in creation and application of load curves are explored and highlighted in the investigation. The fact that the focus is on structural engineering does not necessarily prevent from highlighting related issues which are crucial at the design stage and therefore reflected as factors directly influencing the analysis process. The mathematical background, time integration, drawbacks and superior features of the code are thoroughly explained in a fashion intended to enrich the knowledge of new engineers.

Keywords: Explicit codes, implicit codes, LS-DYNA3D formulation, Dynamic structures formulation

1. Introduction

The LS-DYNA3D is an explicit '3-D' finite element code for analyses of the large deformation dynamic response of elastic, inelastic solids and structures. The program is extensively used by (i) aerospace, including effects of birds striking airplanes, aircraft analysis, improved methods of manufacturing aircraft wings (ii) analysis of shipping containers for radioactive materials and ship collision (iii) automobile design crash tests, and human effects (iv) biomedical studies, including brain impact studies, human response to balloon angioplasty, spinal region surgical procedures and artificial implants, and development of a new machine to break kidney stones using shock waves (lithotripter) (v) aluminium can and metal forming production (vi) train accident analysis (vii) structural analysis.

Corresponding author. P O Box 82677, Tripoli, Libya.
E-mail address: malmherigh@gmail.com

A wide range of material types and interfaces enable the efficient mathematical modelling of wide spectrum of complex engineering problems. LS-DYNA3D (dynamics in three dimensions) is a non-linear structural dynamics code that can analyse the complex structural response of mechanical and geometrical systems subjected to high-rate of loadings and impacts. The program simulates the effect of stress and displacements on structures. Like most engineering codes LS-DYNA3D is based on finite element method, a technique of examining physical systems by breaking them into discrete but interconnected elements. What makes this program different is its ability to span a broad spectrum of engineering problems and computing environments. Developed in the second half of the 1970s mainly for supercomputer applications to study the effects of weapons systems, the rapidly improving program became extremely versatile. The parallel development in computing hardware enables the use of the program on personal pc.

2. Basic Formulation of LS-DYNA3D

The finite element method is a numerical procedure for analysing structures and continua. The finite element method involves discretising differential equations into simultaneous algebraic equations. The advances made in the computational efficiency of digital computers have increased the use of the finite element method as an analysis tool since large number of the equations generated by the finite element method can be solved very efficiently. Discussions to follow are intended to give attention to some important features rather than to cover the whole topic they are extracted from [1,2,3,4].

Based on the momentum conservation principle, the basic formulation could be derived as follows, [1];

$$\sigma ij,j + \rho fi = \rho \ddot{x}_i \tag{1}$$

Equation (1) should satisfy the traction boundary conditions

$$\sigma ij ni = ti(t) \tag{2}$$

On boundary $\delta b1$, while the displacement boundary conditions

$$xi(X\alpha, t) = Di(t) \tag{3}$$

On boundary $\delta b2$, the contact discontinuity

$$(\sigma+ij - \sigma-ij) ni = 0 \tag{4}$$

Along an interior boundary $\delta b3$ when x+i=x-i, here σij is the Couchy stress, ρ is the density, fi is the body force density, $\ddot{x}$ is the acceleration, and nj is the unit vector normal to the boundary δb

Equations of global energy balance and state evaluations can be obtained by integrating the energy equation in time. The energy equation is given as

$$\dot{E} = Vs_{ij} \dot{\mathcal{E}}_{ij} - (p+q)\dot{V} \tag{5}$$

s_{ij} is the deviatoric stresses and p represents the pressure. Deviatoric stress is

$$Sij = \sigma ij + (p+q) \delta ij \tag{6}$$

Where

$$P = -\tfrac{1}{3} \sigma ij \delta ij - q = -\tfrac{1}{3} \sigma kk - q \tag{7}$$

Respectively, q is the bulk viscosity, δij is the Cronecker delta ($\delta ij =1$ if $i =j$; otherwise $\delta ij =0$) and $\dot{\varepsilon}_{ij}$ is the strain rate tensor. The weak form of the equilibrium equation when δx_i satisfies all the boundary conditions on δb_2 is given

By:

$$\int_v (\rho \ddot{x}_i - \sigma_{ij,j} - \rho f_i)\delta x_i dv + \int_{\delta b1}(\sigma_{ij}n_j - t_i)\delta x_i ds + \int_{\delta b3}(\sigma_{ij}^+ - \sigma_{ij}^-)n_j \delta x_i ds = 0 \qquad (8)$$

Application of $(\sigma_{ij}\delta x_i),j - \sigma_{ij,j}\delta x_i = \sigma_{ij}\delta x_{i,j}$ this to the divergence theorem, leads to the statement of principle of virtual work, which can be given as

$$\delta\pi = \int_\upsilon \rho \ddot{x}_i \delta xi\ dv + \int_\upsilon \sigma ij,j\ \delta x^{i,j}\ dv - \int_\upsilon \rho f i \delta xi\ dv - \int_{\partial b1} ti\ \delta xi\ ds = 0 \qquad (9)$$

Superimposing a mesh of finite elements, interconnected at nodal points on a reference configuration and track particles through time, i.e.

$$xi\ (X\alpha\ ,t) = xi\ (X\alpha(\xi,\eta,\zeta),\ t) = \sum_{j=1}^{k} \Phi j(\xi,\eta,\zeta)xij\ (t) \qquad (10)$$

Where Φj are shape interpolation functions of the parametric coordinates (ξ,η,ζ) ,k is the number of nodal points defining the element, and xij is the nodal coordinate of the jth node in the ith direction. One of the widely used 8-noded mesh solid element typically shown in Figure (1). The node definition for this element is shown next to the solid element.
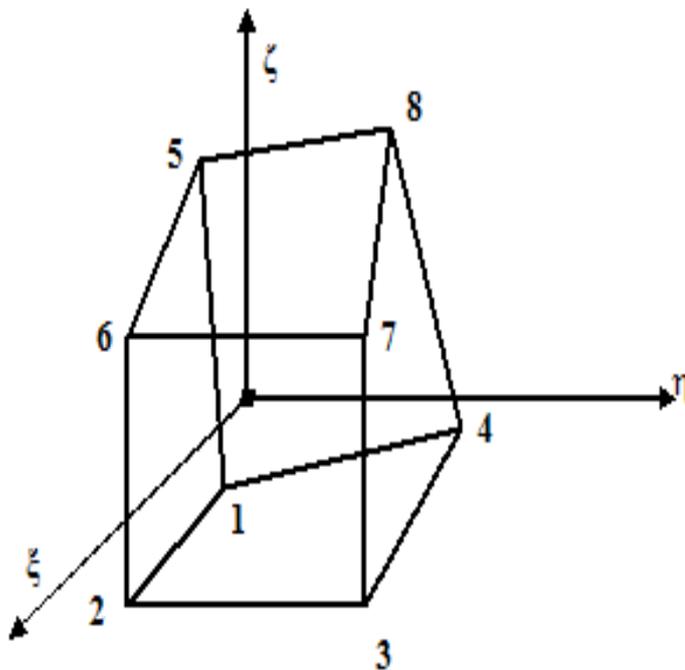
Figure (1) Solid element

node	ξ	η	ζ
1	-1	-1	-1
2	1	-1	-1
3	1	1	-1
4	-1	1	-1
5	-1	-1	1
6	1	-1	1
7	1	1	1
8	-1	1	1

For a mesh with a 8-node hexahedron solid element, equation (10), becomes

$$xi\ (X\alpha\ ,t) = xi\ (X\alpha(\xi,\eta,\zeta),\ t) = \sum_{j=1}^{8} \Phi j(\xi,\eta,\zeta)xij\ (t) \tag{11}$$

and the shape function

$$\phi_j = \frac{1}{8}(1+\xi\xi_j)(1+\eta\eta_j)(1+\zeta\zeta j) \tag{12}$$

where ξ_j,η_j,ζ_j take their nodal values of ($\pm 1,\pm 1,\pm 1$) as shown in the previous figure.

Summing over the n elements δπ may be approximated to

$$\delta\pi = \sum_{m=1}^{n} \delta\pi m \tag{13}$$

And applying this to equation, (9) we get the global finite element equilibrium as

$$\sum_{m=1}^{n} \{ \int_{v_m} \rho\ \ddot{x}_i\ \Phi im\ d\upsilon + \int\upsilon m\ \sigma_{ij}^{m}\ \Phi_{i,j}^{m}\ d\upsilon - \int\upsilon m f i \Phi im\ d\upsilon - \int_{\partial b1} t_i \phi_i^m ds \} = 0 \tag{14}$$

Where:

$$\Phi im = (\qquad\qquad \Phi \qquad\qquad \square,\ \Phi 2,\ \Phi 3, \cdots\cdots \Phi k)m \tag{15}$$

In matrix notation equation (14) becomes

$$\sum_{m=1}^{n} \{ \int\upsilon m \rho NtNa\ d\upsilon + \int\upsilon m Bt\sigma\ d\upsilon - \int\upsilon m \rho Nt\ bd\upsilon - \int_{\partial b1} N^t tds \}m = 0 \tag{16}$$

Where N is an interpolation matrix, σ is the stress vector

$$\begin{bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \\ \ddot{x}_3 \end{bmatrix} = N \begin{bmatrix} a_{x1} \\ a_{y1} \\ . \\ . \\ a_{yk} \\ a_{zk} \end{bmatrix} = N\ a$$

$$\sigma t=(\sigma xx\ ,\sigma yy\ ,\sigma zz\ ,\ \sigma xy\ ,\ \sigma yz\ ,\sigma zx\ ) \tag{17}$$

B is the strain-displacement matrix, a is the nodal acceleration vector

b is the body force load vector, and t are applied traction loads

$$b = \begin{bmatrix} f_x \\ f_y \\ f_z \end{bmatrix}, \text{ and } t = \begin{bmatrix} t_x \\ t_y \\ t_z \end{bmatrix}$$

(18)

For an 8-node hexahedron solid, B is 6 x 24 strain-displacement matrix and N is 3x 24 rectangular interpolation matrix and is given by:

$$B = \begin{bmatrix} \frac{\delta}{\delta x} & 0 & 0 \\ 0 & \frac{\delta}{\delta y} & 0 \\ 0 & 0 & \frac{\delta}{\delta z} \end{bmatrix} \text{ and } N(\xi, \eta, \zeta) = \begin{bmatrix} \phi_1 & 0 & 0 & 0 & \phi_2 & 0 & - & - & 0 & 0 \\ 0 & \phi_1 & 0 & 0 & 0 & \phi_2 & - & - & \phi_8 & 0 \\ 0 & 0 & \phi_1 & 0 & 0 & 0 & - & - & 0 & \phi_8 \end{bmatrix}$$

By summing the rows we can obtain a diagonal mass matrix and the kth diagonal term is given as:

$$m_{kk} = \int_v \rho \phi_k \sum_{i=1}^{8} \phi_i dv = \int_v \rho \phi_k dv$$

(19)

Terms in the strain-displacement matrix are calculated instantly, the above matrixes:

$$\frac{\delta \phi_i}{\delta \xi} = \frac{\delta \phi_i}{\delta x} \frac{\delta x}{\delta \xi} + \frac{\delta \phi_i}{\delta y} \frac{\delta y}{\delta \xi} + \frac{\delta \phi_i}{\delta z} \frac{\delta z}{\delta \xi}$$

$$\frac{\delta \phi_i}{\delta \eta} = \frac{\delta \phi_i}{\delta x} \frac{\delta x}{\delta \eta} + \frac{\delta \phi_i}{\delta y} \frac{\delta y}{\delta \eta} + \frac{\delta \phi_i}{\delta z} \frac{\delta z}{\delta \eta}$$

$$\frac{\delta \phi_i}{\delta \zeta} = \frac{\delta \phi_i}{\delta x} \frac{\delta x}{\delta \zeta} + \frac{\delta \phi_i}{\delta y} \frac{\delta y}{\delta \zeta} + \frac{\delta \phi_i}{\delta z} \frac{\delta z}{\delta \zeta}$$

(20)

or in matrix form

$$\begin{bmatrix} \frac{\delta \phi_i}{\delta \xi} \\ \frac{\delta \phi_i}{\delta \eta} \\ \frac{\delta \phi_i}{\delta \zeta} \end{bmatrix} = \begin{bmatrix} \frac{\delta x}{\delta \xi} & \frac{\delta y}{\delta \xi} & \frac{\delta z}{\delta \xi} \\ \frac{\delta x}{\delta \eta} & \frac{\delta y}{\delta \eta} & \frac{\delta z}{\delta \eta} \\ \frac{\delta x}{\delta \zeta} & \frac{\delta y}{\delta \zeta} & \frac{\delta z}{\delta \zeta} \end{bmatrix} \begin{bmatrix} \frac{\delta \phi_i}{\delta x} \\ \frac{\delta \phi_i}{\delta y} \\ \frac{\delta \phi_i}{\delta z} \end{bmatrix} = J \begin{bmatrix} \frac{\delta \phi_i}{\delta x} \\ \frac{\delta \phi_i}{\delta y} \\ \frac{\delta \phi_i}{\delta z} \end{bmatrix}$$

(21)

Hence the desired terms can be obtained by inverting the Jacobean matrix J as

$$
\begin{bmatrix} \dfrac{\delta\phi_i}{\delta x} \\ \dfrac{\delta\phi_i}{\delta y} \\ \dfrac{\delta\phi_i}{\delta z} \end{bmatrix} = J^{-1} \begin{bmatrix} \dfrac{\delta\phi_i}{\delta\xi} \\ \dfrac{\delta\phi_i}{\delta\eta} \\ \dfrac{\delta\phi_i}{\delta\zeta} \end{bmatrix}
$$

(22)

A similar formulation could be reached for other elements; they are detailed in the theoretical manual, [1].

3. Volume Integration

Gaussian quadrature is used to carry the volume integration. For some functions g defined over with n integration point then:

$$
\int_v g\,dv = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} g\,|j|\,d\xi\,d\eta\,d\zeta
$$

(23)

Which may be approximated by

$$
\sum_{j=1}^{n} \sum_{k=1}^{n} \sum_{l=1}^{n} g_{jkl} \left| J_{jkl} \right| w_j w_k w_l
$$

(24)

Where w_j, w_k, w_l are weighting factors and $g_{jkl} = g(\xi_j, \eta_k \zeta_l)$ and $|J|$ is the determinant of the Jacobean matrix. For one point quadrature n=1, $w_i = w_k = w_l = 2$ and $\xi_1 = \eta_1 = \zeta_{1} = 0$ from which it follows

$$
\int g\,dv = 8g(0,0,0)\,|J(0,0,0)|
$$

(25)

Note that $8\,|J(0,0,0)|$ give an approximation to the element volume.

The biggest advantage of one point integration is a substantial saving in the computer time. An anti-symmetry property of the strain matrix

$$
\frac{\partial\phi_1}{\partial x_i} = -\frac{\partial\phi_7}{\partial x_i} \quad \frac{\partial\phi_3}{\partial x_i} = \frac{-\partial\phi_5}{\partial x_i}
$$

$$
\frac{\partial\phi_2}{\partial x_i} = -\frac{\partial\phi_8}{\partial x_i} \quad \frac{\partial\phi_4}{\partial x_i} = -\frac{\partial\phi_6}{\partial x_i}
$$

(26)

At $\xi = \zeta = \eta = 0$ reduces the amount of effort to compute strain matrix given by equation (26) by 25 times over the 8–point integration [1, 4]. This cost savings extends to strain and element nodal forces calculations where the number of multipliers is reduced by a factor of 16, [2]. Because only one constitutive evaluation is needed, the time spent determining stresses is reduced by a factor of 8. However 8-point integration has another disadvantage in addition to cost. When fully integrated elements used in the solution of plasticity problems and problems with Poisson's ratio approaching 0.5 get locked up in the constant volume bending modes, [1]. An average pressure must be applied to all elements to avoid the locking up of elements in the constant volume bending modes; consequently the hourglass modes are resisted by the deviatoric stresses. If these deviatoric

stresses become negligible, when compared to the pressure or if the material failure causes loss of this stress state component, then hourglass will still occur without any means to resist it.

4. Control of Hourglass

Any explicit time integration scheme calls for what is called "economic" element, which can reduce the overall time. This can be done when a reduced one (Gauss) point integration is used. One point integration however gives rise to "zero energy modes" called the "hourglass modes" as already been discussed. These undesirable hourglass modes are often observed to be oscillatory and they tend to have periods that are typically much shorter than the periods of the structural response. If these hourglass modes tend to have periods that are comparable to the structural response periods it forms a stable kinematics component of the global deformation and they are admissible. A viscous damping or small elastic stiffness capable of stopping the anomalous modes while having a negligible effect on the state global deformation modes can in a way resist the formation of the undesirable hourglass modes. To understand the formation of the undesirable hourglass modes, the following strain rate for an 8-node solid element is considered:

$$\dot{\varepsilon}_{ij} = \frac{1}{2}\left(\sum_{k=1}^{8} \frac{\delta \phi_k}{\delta x_i} \dot{x}_j^k + \frac{\delta \phi_k}{\delta x_j} \dot{x}_i^k \right) \tag{27}$$

If diagonally opposite nodes have identical velocities, then the strain energy rates (due to the asymmetries in equation (26) are identically zero ($\dot{\varepsilon}_{ij} = 0$) that is when the following "hourglass" condition occur (diagonally opposite nodes have identical velocities):

$$\dot{x}_i^1 = \dot{x}_i^7, \dot{x}_i^2 = \dot{x}_i^8, \dot{x}_i^3 = \dot{x}_i^5, \dot{x}_i^4 = \dot{x}_i^6 \tag{28}$$

It is possible to prove the orthogonality of the hourglass shape vectors with the derivatives of shape functions as follows:

$$\sum_{k=1}^{8} \frac{\partial \phi_k}{\partial x_i} \Gamma^{\alpha k} = 0 \qquad\qquad i=1,2,3 \qquad \alpha=1,2,3,4 \tag{29}$$

Hourglass modes are avoided by the use of artificial resisting forces consistent with the orthogonal nature of the modes Гk and related to the element volume, material sound speed. The product of the base vectors with the nodal velocities for these modes are given by

$$h_{i\alpha} = \sum_{k=1}^{8} \dot{x}_i^k \, \Gamma^{\alpha k} = 0 \tag{30}$$

If hourglass modes are present then equation (30) becomes nonzero. And the resisting 12 hourglass force vectors are:

$$f_{i\alpha}^k = a_h h_{i\alpha} \Gamma^{\alpha k} \text{ and } a_h = Q_{hg} \rho v_e^{2/3} \frac{c}{4} \tag{31}$$

Where Qhg is an empirical constant defined between 0.05 and 0.15

Refined finite element meshing of components tends to reduce the hourglass modes which otherwise produce artificial energies in excess of the initial energy of the system.

Thus in conclusion, the biggest disadvantage to one-point integration is the need to control the zero energy modes, which arise, called hourglass modes. Undesirable hourglass modes tend to have periods that are typically much shorter than the periods of the structural response, and they are often observed to be oscillatory. However, hourglass modes that have periods that are comparable to the structural response periods may be a stable kinematics component of the global deformation modes and must be admissible. One way of resisting undesirable hourglass is with viscous damping or small elastic stiffness capable of stopping the formation of the anomalous modes but having a negligible effect on the stable global modes.

Since the hourglass deformation modes are orthogonal to the strain calculations, work done by the hourglass resistance is neglected in the strain energy equation. This may lead to a slight loss of energy; however, hourglass control is always recommended for the under-integrated solid elements. The energy dissipated by the hourglass forces reacting against the formations of the hourglass modes is tracked and reported in the output files.

5. Time Step Control

A new time step size is determined by taking the minimum value over all elements as follows

$$\nabla t^{n+1} = \alpha_{\min}\left(\nabla t_1, \nabla t_2, \ldots\ldots, \nabla t_N\right) \tag{32}$$

Where N the number of elements. For stability reasons the scale factor α is typically set to a value of 0.9 (default) or smaller value.

For a solid element, a critical time step size ∇t_e is computed as follows:

$$\nabla t_e = \frac{Le}{\left(\left[Q + (Q^2 + c^2)^{1/2}\right]\right)} \tag{33}$$

Where Le= $\dfrac{v_e}{A e_{\max}}$ for 8-node solid element and Q (C0, C1) and C0, C1 are the bulk viscosity coefficients and Q is given by

$$Q = \frac{C_1 c + C_0 v_e |\dot{\varepsilon}_{kk}|}{A_{e\max}} \quad \text{for } \dot{\varepsilon}_{kk} < 0 \tag{34}$$

$$Q = 0 \text{ for } \dot{\varepsilon}_{kk} \geq 0$$

L_e Is the characteristic length:

8-nodes:
$$L_e = \frac{v_e}{A_{e\max}}$$

4-node tetrahedras: $L_e = $ minimum altitude

Where v_e is the element volume, $A_{e\max}$ is the area of the largest side, and c is the adiabatic sound speed given by

$$c = \left[\left(\frac{4G}{3\rho_0} + \frac{\delta p}{\delta \rho} \right)_s \right]^{1/2} \tag{35}$$

And ρ is the specific mass density.

For an isentropic sound speed, c is given by

$$c = \left[\frac{4G}{3\rho_0} + \frac{\delta p}{\delta \rho} \right)_E + \frac{pV^2}{\rho_0} \frac{\delta p}{\delta E} \right)_\rho \right]^{1/2} \tag{36}$$

For the incremental energy, E in the units of pressure is the product of pressure p and the incremental relative volume dv:

$$\frac{\delta p}{\delta \rho} \right)_s = \frac{\delta p}{\delta \rho} \right)_E + \frac{\delta p}{\delta E} \right)_\rho \frac{\delta E}{\delta \rho} \right)_S \tag{37}$$

And

dE = -pdv

Sound speed for elastic materials with constant bulk modulus is given by

$$c = \sqrt{\frac{E(1-\nu)}{(1+\nu)(1-2\nu)\rho}} \tag{38}$$

Where E is the Young's modulus and ν is the Poisons ratio.

This is the approach for solid elements, while reference is made to the LS-DYNA3D theoretical manual, [1] for time step calculation of beam and truss elements, shell elements and discrete elements. Explicit methods are based on time integration; therefore time step factors in the time step control cards are of utmost importance.

6. Time Integration

The equations of equilibrium for a nonlinear finite element system in motion are nonlinear differential equations for which numerical solutions much easier to obtain in general than analytical solution. The procedure used to solve the equations of equilibrium can be divided into two methods: direct integration and mode superposition.

In direct integration, equations of equilibrium are integrated using a numerical step-by-step procedure. The term 'direct' is used because the equations of equilibrium are nottransformed into any other form before the integration process is carried out. Some ofthe few commonlyused direct integration methods are the central difference method, Houbolt method, Wilson-theta method, and Newmark method, [1, 2, 3, 4].

LS-DYNA3D is based on the central difference method of direct integration therefore; the description of the direct integration method will concentrate on the central difference method.

Considering the single degree of freedom damped system as shown below, the equilibrium equations for the given system are obtained from d'Alembert's principle
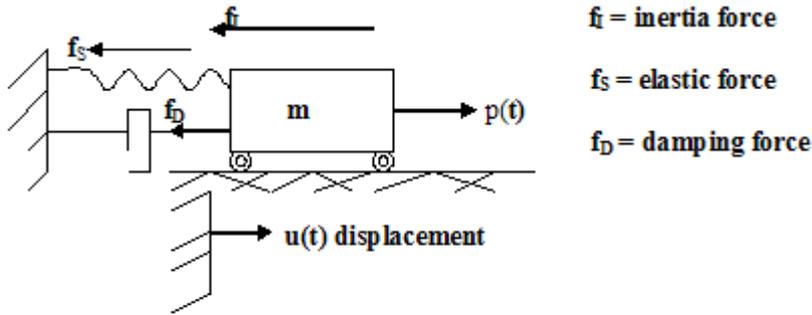
Forces acting on damped mass m

$$fI + fD + fint = p(t) \quad fI + fD + fint = p(t) \tag{39}$$

$fI = m\ddot{u}$; $\ddot{u} = \dfrac{d^2u}{dt^2}$ is acceleration

$fD = c\dot{u}$; $\dot{u} = \dfrac{du}{dt}$ is velocity, and c is the damping coefficient

$f_{int} = k.u$; u displacement and k is the linear stiffness

For a linear behaviour the equations of motion lead to linear ordinary differential equations given by

$$m\ddot{u} + c\dot{u} + k.u = p(t)$$ and for a nonlinear case the inertial force varies as a nonlinear function of the

displacement which leads to a nonlinear ordinary differential equation given by $m\ddot{u} + c\dot{u} + f_{int}(u) = p(t)$

Analytical solutions of such equations is available, considering the dynamic response of a linear system subjected to a harmonic loading, with the knowledge of the common used terms

Harmonic loading: $P(t) = Po \sin \varpi t$

Circular frequency: $\omega = \sqrt{\dfrac{k}{m}}$ for single degree of freedom

Natural frequency: $f = \dfrac{\omega}{2\pi} = \dfrac{1}{T}$, T is the period

Damping ratio: $\zeta = \dfrac{c}{c_{cr}} = \dfrac{c}{2m\omega}$

Damped vibration: $\omega_0 = \omega\sqrt{1 - \zeta^2}$

Applied load frequency: $\beta = \dfrac{\varpi}{\omega}$

The closed form solution with initial conditions u0 = initial displacement, $\dot{u}_0$ = initial velocity and $\dfrac{p_0}{k} =$ static displacement is given by;

$$u(t) = u_0 \cos \omega t + \frac{\dot{u}_0}{\omega} \sin \omega t + \frac{p_0}{k} \frac{1}{1-\beta^2} (\sin \varpi t - \beta \sin \omega t) \tag{40}$$

For nonlinear problems, only numerical solutions are possible. LS-DYNA3D uses the explicit central difference method to integrate the equations of motion.

The semi-discrete equations of motion at time n are:

$$\text{Man} = \text{Pn} - \text{Fn} + \text{Hn} \tag{41}$$

Where M is the diagonal mass matrix, Pn is the external and body force loads, Fn is the stress divergence vector, and Hn is the hourglass resistance. The central difference time integration method is used to advance time tn+1 as follows:

$$a^n = M^{-1}(P^n - F^n + H^n) \tag{42}$$

$$v^{n+1/2} = v^{n-1/2} + a^n \nabla t^n \tag{43}$$

$$u^{n+1} = u^n + v^{n+1/2} \nabla t^{n+1/2} \tag{44}$$

Where $\nabla t^{n+1/2} = \dfrac{(\nabla t^n + \nabla t^{n+1})}{2}$

And v and u are the global nodal velocity and displacement vectors, respectively.

Initial geometry can be updated by adding the displacement increments

$$\text{Xn+1} = \text{x0} + \text{un+1} \tag{45}$$

The stability of the central difference method is determined by looking at the stability of the linear system. The equations of a linear system is uncoupled into the model equations where the model matrix of Eigen vectors, Φ, are normalized with respect to the mass matrix M, and linear stiffness matrix K. The decoupling of damping matrix, C for viscous proportional damping is obtained from the normalization as

$$\Phi T \, C \, \Phi = 2\xi\omega \tag{46}$$

The equations of motion in the modal coordinates x are:

$$\ddot{x} + 2\zeta\omega\dot{x} + \omega^2 x = \underbrace{\phi^T p}_{=y} \tag{47}$$

With central differences we obtain for the velocity and acceleration

$$\dot{x}_n = \frac{x_{n+1} - x_{n-1}}{2\nabla t} \quad \text{and} \quad \ddot{x}_n = \frac{x_{n+1} - 2x_n + x_{n-1}}{\nabla t^2} \tag{48}$$

Substituting equation (43) into equation of motion at time tn leads to

$$x_{n+1} = \frac{2 - \omega^2 \nabla t^2}{1 + 2\zeta\omega\nabla t^2} x_n - \frac{1 - 2\zeta\omega\nabla t}{1 + 2\zeta\omega\nabla t} x_{n-1} + \frac{\nabla t^2}{1 + 2\zeta\omega\nabla t^2} y_n \tag{49}$$

And equation (48) in the matrix form is given by

$$\begin{bmatrix} X_{n+1} \\ \\ X_n \end{bmatrix} = \begin{bmatrix} \dfrac{2-\omega^2\nabla t^2}{1+2\zeta\omega\nabla t} & -\dfrac{1-2\zeta\omega\nabla t}{1+2\zeta\omega\nabla t} \\ \\ 1 & 0 \end{bmatrix} \begin{bmatrix} \dfrac{X_n}{X_{n-1}} \end{bmatrix} + \begin{bmatrix} \dfrac{\nabla t^2}{1+2\zeta\omega\nabla t^2} \\ \\ 0 \end{bmatrix} \tag{50}$$

Or

$$[X_{n+1}] = [A][X_n] + [L]Y_n \qquad (51)$$

Where A is the time integration operator for discrete equations of motion. After m time steps with L = 0 the value $[X_m] = [A_m][X_0]$

The spectral radius ρ (A) is the largest Eigen value of A the solution will be stable if and only if $|\rho(A) \le 1|$.

The condition that $|\rho(A) \le 1|$ for an undamped equation of motion gives the value of time step ∇t as

$$\nabla t \le \frac{2}{f_{max}}$$

Thus the time step size is bound by the largest natural frequency in the model, which in turn is bound by the highest natural frequency of any element in the model.

The time step size is always limited by the single element in the model. The disadvantage of this method is that even if only one element has a time step much less, then all other elements should also be calculated for the same step size. Hence, 'sub-cycling' called as mixed time integration has to be introduced. This method employs in grouping the elements based on their step size and individual time integration should be carried out for each group. This method could be helpful as it reduces the time for solving. However the use of mass scaling to preserve a reasonable time step size works better than sub cycling and hence the later is turned off.

6.1 Central Difference Method

Considering dynamic system, represented mathematically by a system of ordinary differential equations with constant coefficients. The central difference method is an effective solution scheme for such a system of equations. The velocity and acceleration are approximated, Figure (2) as follows:

$$\dot{u}_n = \frac{1}{(\Delta t)^2}(u_{n+1} - u_{n-1}) \qquad (52)$$

$$\ddot{u}_n = \frac{1}{(\Delta t)^2}(u_{n+1} - 2u_n + u_{n-1}) \qquad (53)$$

Substituting the approximate values for the velocity and acceleration from the central difference scheme in the equations of equilibrium, to get

$$\left(m + \frac{1}{2}\Delta t c\right) u_{n+1} = \Delta t^2 p_n - (\Delta t^2 k - 2m)u_n - \left(m - \frac{\Delta t}{2}c\right)u_{n-1} \qquad (54)$$

Where pn is the external body force loads, the solution for un+1 can be determined. Since the solution for un+1 is based on conditions at time tn-1 and tn, the central difference integration procedure is called the explicit integration method. Also this method does not require the factorization of the effective stiffness matrix in the step-by-step solution. On the other hand, other methods like Newmark, Wilson and Houbolt does not involve conditions at time tn+1 hence called implicit integration methods, [1, 2, 3].
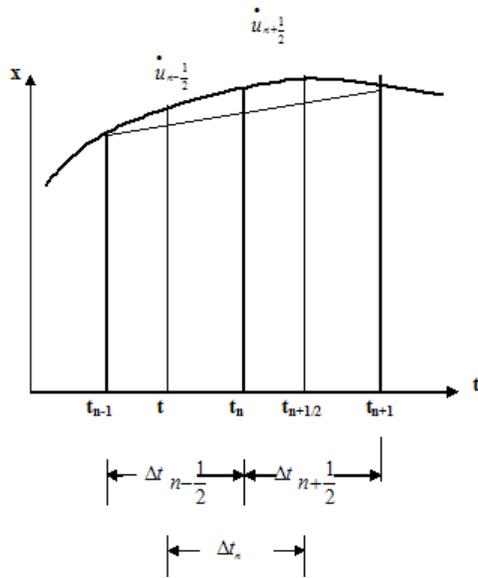
Figure (2) Central difference method representation

6.1.1 Advantages of the Central Difference Method

The main advantage of the central difference method is that no stiffness and mass matrices of the complete element assemblage are calculated, [4, 5, 6]. The solution can be essentially carried out on an element level and relatively very little storage is needed. The method becomes more effective if the element stiffness and mass matrices of subsequent elements are the same, since it is only necessary to calculate or read from back-up storage the matrices corresponding to the first element in the series. This is why systems of large order can be solved very efficiently using the central difference scheme. The effectiveness of the central difference procedure depends on the use of a diagonal (lumped) mass matrix and neglect of general velocity-dependent damping forces. The benefits of performing the solution at the element level are preserved only if the diagonal damping matrix is included.

6.1.2 Disadvantages of the Central Difference Method

The central difference methods as well as other explicit methods are conditionally stable. If the time step Δt is too large for a given element size L the method fails and if Δt is smaller than required the solution time becomes very expensive losing the effectiveness of the method. Therefore, it is necessary to determine the critical time for the given problem. For central difference method, critical Δt is governed by the following equation.

$$\Delta t = \frac{L}{C} \quad (55)$$

Where, $C =$ light wave speed $= \sqrt{\dfrac{E}{\rho}}$, E= Material Young's Modulus

ρ =Material Density. The above equation is called the CFL (Courant, Friedrichs and Lewy) condition, [5, 6]. The physical interpretation of the condition is that the time step Δt must be small enough that the information

does not propagate across more than one element per time step. In some structural analysis depending on the material properties and dimensions of the geometry the time step required could be very small resulting in a longer computational time.

7. Numerical Analysis

The numerical analysis attempts to solve the differential equations by numerical procedures, which can be easily programmed using a standard programming language. The higher time step values from the initial boundary values can be programmed by the differential equations. This is done by splitting the differential equation into numerical components in the time axis using the forward, central or the backward differentiation methods. The numerical methods can be broadly classified as the explicit and implicit methods. In the explicit method, the previous time step is used to calculate the next time step. On the other hand, the implicit method calculates the next time step values by solving a matrix of the present and the previous time step values.Theexplicit method gives an accurate solution for a shorter time step and is conditionally stable, whereas the implicit method requires larger time steps for correct results, as already been discussed in the previous section.

8. Contact-Impact Algorithms

LS-DYNA3D uses three approaches for dealing with the impact contact and sliding interfaces of the models. The methods are known as the "kinematics constraint method", the "penalty method", and the "distributed parameter method", [2, 3]. The kinematics constraint method is used only for tying surfaces. In this method, constraints are imposed in the global equations by a transformation of the nodal displacement components of the "slave" nodes along the contact interfaces. This way only the global degrees of freedom of each master node are coupled. This method requires "consistent" zoning of the interfaces. In the penalty method, artificial interface springs are placed normal to the contacting surfaces on all the penetrating nodes. These artificial spring elements are assembled in the global stiffness matrix and their modulus is determined based on the elements in which the nodes reside. This method is stable and produces less noise for hourglass modes. However, for relatively large interface pressures, the stiffness has to be scaled up and the time step reduced. In such cases the third method "distributed parameter" is more appropriate. This last method is mainly used for "sliding interfaces" in which the internal stress in each element in contact determines the pressure distribution for the corresponding master surface. Accelerations are updated after mass and pressure distributions on the master surface are completed. With these three algorithms, an array of contacting interfaces is available in LS-DYNA3D, which allows the simulation of most contact conditions.

Treatment of sliding and impact along interfaces are very critical in simulating the correct load transfer between components in an analysis. Contact forces generated influence the acceleration of the body. Contact algorithms employed in finite element codes divide the nodes of bodies involved in contact into slave and master nodes. After the initial division, each slave node is checked for penetration against master nodes for an element face. Therefore, using a robust contact algorithm that can efficiently track and generate appropriate forces to the slave nodes without generating spurious results is very important. A brief discussion of the three methods used by LS-DYNA3D with merits and demerits follows:

8.1 Kinematics constraint method

This method uses the impact and release conditions of [1]. Constraints are imposed on the global equations by a transformation of the nodal displacement components of the slave nodes along the contact surface. This transformation has the effect of eliminating the normal degree of freedom of nodes. Since computational efficiency of the explicit time integration needs to be preserved, the mass is lumped to the extent that only the global degrees of freedom of each master node are coupled. Impact and release conditions are imposed to ensure momentum conservation. This method is advantageous to use when materials in contact have very different material properties. The nodes are constrained to stay on or very close to the surface without causing penetrations due to the difference in the stiffness. However, problems arise when the master surface zoning is finer than the slave surface zoning. Certain master nodes can penetrate through the slave surface without resistance and create a kink in the slide line.

8.2 Penalty method

This method consists of placing normal interface springs between all penetration nodes and the contact surface. With the exception of the spring stiffness matrix, which must be assembled in the global stiffness matrix, the implicit and explicit methods are similar. Momentum is conserved without the necessity of impact and release conditions. The equations involving the stiffness of the contact spring are as follows:

$$k = f_s * Area^2 K \Big/ \text{Minimum diagonal length for shell elements} \qquad (56)$$

$$k = \frac{f_s * Area^2 * K}{Vlume} \qquad \text{for solid elements} \qquad (57)$$

Area = Area of contact segment

K   = Bulk modulus of contacted element

f_s = penalty factor (0.1 by default)

The interface stiffness k is chosen to be approximately the same order of magnitude as the stiffness of the interface element normal to the interface. Consequently the computed time step size is unaffected by the existence of the interfaces. However, if interface pressure becomes large, unacceptable penetration may occur.

By scaling up the stiffness f_s and scaling down the time step Δt, this may be overcome. K for a contact segment is calculated based on the material properties of the component involved in the contact. If two different materials with varying stiffness such as foam and steel come in contact, the stiffness of the lesser magnitude is taken as the contact stiffness. This causes penetration problems as the force generated by foam is small compared to that generated by steel. This is overcome by scaling f_s until the forces generated by the two materials are in equilibrium. Determining the appropriate value of f_s is important so that the forces are in equilibrium.

8.3 Distributed parameter method

This method is derived from TENSOR programs, [1], which displaced fewer mesh instabilities compared to the nodal constraint algorithm. In this method one half the slave element mass of each element in contact is distributed to the covered master surface area. Also, the internal stress in each element determines a pressure distribution for the master surface area that receives mass. After the distribution of mass and pressure the acceleration of the master surface is updated.

9. Contact Energy Calculations

The contact energy $E_{contact}$ is incrementally updated from time n to n+1 for each contact interface. $E_{contact}$ is determined using the following equation,

$$E_{contact}^{n+1} = E_{contact}^{n} + \left[\sum_{i=1}^{nsn} \Delta F_i^{slave} * \Delta dist_i^{slave} + \sum_{i=1}^{msn} \Delta F_i^{master} * \Delta dist_i^{master} \right]^{n+1/2}$$

(58)

where

   nsn   = number of slave nodes

   msn   = number of master nodes

ΔF_i^{slave} = interface force between the ith slave node and the contact segment

ΔF_i^{master} = interface force between the ith master node and the contact segment

$\Delta dist_i^{slave}$ = is the incremental distance the ith slave node has moved during the current time step.

$\Delta dist_i^{master}$ = is the incremental distance the ith master node has moved during the current time step.

Monitoring the contact energy calculated is very important to ensure proper calculation made by the contact algorithm. In the absence of friction, the slave and master side energies should be close in magnitude but opposite in sign. The sum $E_{contact}$ should equal the stored energy. Large negative contact energy is a sign of undetected nodal penetrations.

10. Conclusions

An extremely powerful commercial implicit finite element code basic formulation is introduced through which a wide variety of dynamic structures can be analysed. Features involved in this program includes but not limited to: wide material element library, flexibility in load application techniques through load curves, automatic and manual time step control for analysis stability. The code is using time integration technique and hence features such as hourglass control is included and highlighted to reader. The program is running in an interactive mode hence enabling intervention of the operator if needed, literally there is almost no limit to the engineering stress analysis problem that cannot be tackled by this code given excellent user knowledge is available. Moreover,

code has an interface with the well-known implicit code ANSYS, therefore adding to its complications and powerfulness, such feature is important for certain analysis problems.

Code provider is running a very helpful website through which a licence key can be provided online as well as well-trained team who can provide help online, as well as providing team work environment for problem solving. The code is periodically updated and improved, hence user need be following such improvements and can mean time participate in such improvements via feedbacks if he or she wishes.

If any; drawbacks simply summarised in technical difficulties such as: code contains huge number of statements and subroutines, therefore running time as well as computer memory requirements are significantly large added to that extremely good knowledge is required, this of course will add to the expenses, hence only carefully selected stress analysis problems must be involved.

Due to the powerfulness of the code researchers and stress analysts faced with complicated and or highly sensitive stress problems are encouraged to join the good team and be involved in applying the various features to their problems given the worthiness of the problem.

References

[1]   Jhon O Halliquist "LS-DYNA Theoretical Manual" May 1998 Livermore Software Technology Corporation, online PDF file LSTC website.

[2]   Subramani Balasubranyam "Head Impact Characterization of Generic A-Pillar Of An Automobile" MSc thesis, Morgantown, West Virginia 1999 http://etd.wvu.edu/ETDS/E699/finalThesis6.PDF, August 2003.

[3]   Jerry I Lin "DYNA3D: A Nonlinear, Explicit, Three-Dimensional Finite Element Code for Solid and Structural Mechanics" User Manual, October 1999 http://www.llnl.gov/eng/mdg/Codes/DYNA3D.pdf , April 2003.

[4]   Klaus Weimar "LS-DYNA User's Guide" Rev. 1.19, September 2001, CAD-FEM GmbH, www.cadfem.de or PWS www.p-w-s.com, June 2002.

[5]   Bathe K J "Finite Element Procedures in Engineering Analysis" Prentice-Hall Inc. USA 1982.

[6]   R D Coock et al "Concepts and Applications of Finite Element Analysis" John Wiley & Sons 2002.