Automatic Nonlinear Solutions in Structural Analysis Using ABAQUS

Y.R. Crutzen
Commission of the European Communities, J.R.C. Ispra Establishment,
System Engineering and Reliability Division, I-21020 Ispra, Italy
H.H. Hibbitt
Hibbitt, Karlsson and Sorensen, Inc., 35 South Angell Street, Providence,
Rhode Island 02906, U.S.A.

Abstract

This paper contains a summary description of the advanced computational procedures of automatic incremental solution for nonlinear structural dynamic analysis using the finite element program ABAQUS. After a brief introduction dedicated to nonlinear capabilities and tolerance parameters to be defined for the dynamic procedures available in the code, the concepts of self-adaptive time stepping with conventional implicit dynamic algorithms are presented.

Applications to structural integrity analysis associated with nuclear power plants and related to extreme accident events will demonstrate the efficiency of the automatic capabilities of the computer program in handling severe nonlinear situations.

I. Introduction

The actual development effort in general purpose structural analysis systems is to provide a large range of engineering modeling nonlinear capabilities that very often are implemented first for nuclear safety studies. But, as these large-scale computer programs are also of interest in a production environment, essential requirements of usability, flexibility, efficiency, reliability and economy have to be achieved for general nonlinear solutions in structural mechanics.

ABAQUS, a modern structural and heat transfer analysis system developed by Hibbitt, Karlsson & Sorensen, Inc, /I,2/, satisfies these industry needs and possesses powerful material, geometry and boundary nonlinear options.

In nonlinear analysis, the solution must be developed by a series of small increments inside an analysis step because any nonlinear behaviour is usually history dependent. Within any increment, various iterations are performed as necessary to achieve convergence of the solution. In those procedures that have no physical time scale (static, geostatic, steady state heat transfer, etc.), the increment size choice is based on the rate of convergence of the nonlinear equation solution. In the other procedures where a physical time scale exists (dynamics, consolidation, creep and swelling, transient heat transfer, etc.), the increments are based on integration accuracy tolerances.

In ABAQUS, for dynamic stress/displacement analysis using direct time integration, the analyst has the choice of solving his problem by directly specifying the incremental process to be followed, or by allowing the program choose its own adaptive incremental scheme. In this case, additional tolerances parameters that permit to control solution error from equilibrium residuals are defined like:

- PTOL :basic tolerance measure. All forces at all finite element nodes must fall below this tolerance otherwize the program continues to iterate the increment;
- HAFTOL:half-step residual tolerance to be used for automatic time increment algorithm

An accurate solution will be obtained if these tolerance and residual parameters are relatively small compared to any significant forces in the solution.

2. Nonlinear Dynamic Analysis

The program ABAQUS is essentially designed to compute overall dynamic responses of structural components that frequently fall into Belytschko's class of "inertial-type" because their response time usually is longer compare to the time required for waves to transverse the structure. Time integration operators are broadly characterized as implicit or explicit in the literature. Explicit schemes compute directly the nodal accelerations, velocities and displacements from the basic equations of motion without any assemblage and inversion of large mass and stiffness matrices/3,4/. As a consequence, explicit schemes obtain values for dynamic equations at time (t+Dt) based entirely of available values at time t. This places an upper bound on the time increment size Dt in order to maintain numerical stability. Their use is of interest for fast (wave-propagation-type) transient dynamic analysis/3,5/. Implicit schemes remove the upper bound on time increment by solving the equations at time (t+Dt) based not only on values at time t, but also on the same quantities at time (t+Dt). In structural problems, implicit integration algorithms give acceptable results with time steps Dt equal or larger of one to two order of magnitude than the stability limit of simple explicit schemes, but they need convergence iterations within a time increment for nonlinear response.

The implicit operator implemented in ABAQUS is the Hilber-Hughes-Taylor operator/I/, defined also as a single parameter operator (ALFA parameter) for controllable numerical damping. This numerical damping is most valuable in the automatic time stepping scheme because the sligh high frequency numerical noice inevitably introduced when time increment is changed can be rapidly removed by a small amount of numerical damping.

The actual equilibrium equations are defined by

$$[M]\{\ddot{u}\} + \{I\} - \{P\} = 0 \qquad (I)$$

where $[M] = \int \rho [N]^T [N] \, dV^e$ consistent mass matrix

$\{I\} = \int [B]^T \{\sigma\} \, dV$ internal force vector

$\{P\} = \int [N]^T \{t\} \, dS + \int [N]^T \{f\} \, dV$ external force vector

In ABAQUS, the operator replaces eq. (I) with a balance of d'Alembert forces at the end of the time increment and a weighted average of static forces at the beginning and the end of the time increment:

$$[M]\{\ddot{u}\}_{t+Dt} + (1+\alpha)(\{I\} - \{P\})_{t+Dt} - \alpha(\{I\} - \{P\})_t = 0 \qquad (2)$$

Newmark formulae complete the displacement and velocity integration.

As implicit operators require the complete resolution of the equations of motion at each increment, the definition of a correct time increment will be closely bounded with essential requirements of accuracy and cost. In ABAQUS, a computationally cheap algorithm is included to automatically choose the time step Dt and is based on the concept of half-step residual. Having equilibrium ensured at the beginning and at the end of each time step, the idea of the half-step residual $|R|$ is to estimate the quality of the solution and to calculate the equilibrium residual error at some intermadiate time point, the half-step point, for economy and semplicity. So, the evaluation of residual $|R|$ is identified as the magnitude of the largest entry in the following vector:

$$|R| = \max \left| \{R\}_\tau = [M]\{\ddot{u}\}_\tau + \{I\}_\tau - \{P\}_\tau \right| \qquad (\tau = t+Dt/2) \qquad (3)$$

The measure of $|R|$ is essential for the accuracy estimation and for the concept of adaptive time integration scheme in the program. If $|R|$ is small and less than the user specified tolerance (HAFTOL parameter), it indicates that precision of the solution is high and the time step may be increased. If $|R|$ is greater than the user tolerance value, it shows that the solution is too coarse; the time step is too big and should be reduced.

The algorithm implemented in the program permits to evaluate the solution accuracy and to modify the time increment magnitude in order to automatically reduce speed or accelerate the convergent solution. The following conditions need to be fulfilled:

- if $|R| > \text{HAFTOL}$, reset Dt to one half of its current value and begin again from time t;
- if $\text{HAFTOL} \geqslant |R| > \text{HAFTOL}/2$, update the state to (t+Dt) and continue the analysis using the same time increment Dt;
- if $|R| > \text{HAFTOL}/2$ for two consecutive increments, next increment Dt is increased by a quarter.

The advantages of this adaptive time stepping scheme will clearly be illustrated for real cases presented in the next paragraphs.

Moreover, an energy content output is available in the program and permits to monitor the automatic nonlinear solution thanks to the overall energy balance check during the dynamic response.

Dynamic analysis often involves impact or intermittent contact. The program includes an algorithm for cases of discrete and severe structural impacts such as pipe whip or crash events. In ABAQUS, the impact concept adopted is that of a perfect plastic impact: at the impact time t_0, the two impacting surfaces instantaneously acquire the same velocity in the direction of the impact. It is also assumed that ,at impact, energy is dissipated by some mechanism which is not a part of the discrete finite element model because the plastic impact event is local with infinitesimal spatial and temporal scale compared to the discrete numerical model (see /I/ for complete theoretical explaination)

In any real problem, impact will occur at some intermediate point in a time step.

Thus, the program will accurately define the impact time and the jump in velocities and accelerations in order to maintain the energy balance in term of energy mechanisms for the discrete model. The time increment used in the solution will drastically be reduced to obtain the convergent solution. After that, as the high frequency noise generated by impact dissipates through plasticity and artificial damping, the next time increments will grow up again according to the automatic time stepping algorithm.

3. Explosion / Structure Interaction

The advanced computer program ABAQUS permits to illustrate some extreme dynamic loading condition acting on thin metal structures. In Fig. I, a cylindrical panel is submitted to impulsive loading (initial velocity) which is supplied by an explosive sheet in contact with the metal. High membrane and bending wave propagation effects occur in the cylinder together with rapid plasticity extension, as observed in Fig. 2 for a relative coarse finite element mesh. In Fig. 3, the elasto-plastic large-displacement response for the same ABAQUS mesh is successfully compared with experimental data from JRC Ispra /4/ and with published numerical results obtained by the computer program EURDYN /4/ and SLOOFSAN /5/. If these last two nonlinear finite element codes are based on the use of explicit direct time integration techniques, ABAQUS clearly demonstrates the interest of its self-adaptive implicit time stepping algorithm for this situation of initially excited structure with high and extensive plasticity dissipation. Fig. 4 illustrates , for two implicit runs with different initial time steps, how the time increment quickly expands as the solution progresses and as the high frequency response is dissipated in the shell structure.

4. Pipe Impact Event

Impact problems are encountered in pipe whip studies where the fluid escaping from a ruptured pipe causes large pipe motions and possible impact with restraints along the pipe. Fig. 5 shows the dynamic effects of a variable blowdown force acting at the free end of acantilever pipe in presence of a ductile restraint with an initial gap. The prediction of the pipe-section beam behavior is in excellent agreement with limited experimental data obtained by Detroit Edison Company /2/ that permit to compare the initial variation of the force transmitted from the pipe to the restraint. The same figure plots the time increment adaptation during the nonlinear response by the automatic time stepping scheme.

In Fig. 6, an energy balance calculation for a rather coarse mesh of a similar structure impacting more than one ductile restraint permits to understand the extend of energy dissipated by plasticity and the loss of kinetic energy by successive energy jumps due to discrete perfectly plastic impacts. In the program, the adaptive time stepping algorithm is particularly powerful for these intermittent contact and impact problems.

5. Soft-Missile / Structure Interaction

This application illustrates ABAQUS capabilities in the field of anti-missile design.

The simulation of a soft-missile (aircraft) impact on a concrete nuclear containment building emphasizes the practicability of the numerical methods available and demonstrates their use for studies of partial disintegration of reinforced concrete components /I,2/. For the complete 3D problem of the reactor building loaded laterally, the values for the horizontal displacement at node 1 (see Fig. 7) obtained for linear and nonlinear computations are very close to the response values published like those given by SLOOFSAN /5/ for a concrete tensile strength of 5.6 MPa and for a similar shell model:

ANALYSIS	SLOOFSAN	ABAQUS
Linear	4.03 cm	3.82 cm
Nonlinear	4.47 cm	4.62 cm

In Fig. 8, the loading function (civil aircraft) is presented with the variation of the horizontal deflection at node 1 and with the time increment adaptation for both the linear and nonlinear analyses. ABAQUS analysis determines the overall knowledge for stresses and strains in both the concrete and the steel reinforcement. For the present calculation, only multiaxial cracking appears in the impacted region without important concrete crushing nor steel yielding. More experimental research into strain-rate effects on constitutive relations for reinforced concrete is still needed in order to apply the results of computer programs to anti-missile design purpose with confidence.

6. Conclusions

Some capabilities of the ABAQUS program for the automatic solution of highly nonlinear structural problems have been briefly discussed and a few real cases in the nonlinear dynamic field have been presented.

Together with Computer-Aided-Design (CAD) tools for automatic geometric modelling and finite element mesh generation and with the extensive aid of post-processors for numerical results visualization, the improvement of computer programs with self-adaptive algorithms for the generalized resolution of nonlinear problems gives new and powerful opportunities to the analysts faced to more and more complex and advanced engineering applications.

7. References

/I/ HIBBITT, KARLSSON & SORENSEN, INC., ABAQUS User's Manual, ABAQUS Example Problems Manual, ABAQUS Theory Manual, ABAQUS System Manual, Providence, R.I., USA, I982/I984.

/2/ HIBBITT, H.D.,"ABAQUS/EPGEN - A General Purpose Finite Element Code With Emphasis On Nonlinear Applications", Nucl. Eng. Design, I984.

/3/ CRUTZEN, Y.,"Nonlinear Transient Dynamic Analysis of Thin Shells Using The Semiloof Finite Element", Doctoral Thesis, Univ. Brussels, I979.

/4/ DONEA, J., GIULIANI, G., HALLEUX, J.P.," Prediction of the Nonlinear Dynamic Response of Structural Components Using Finite Elements", Nucl. Eng. Design, 37, I976.

/5/ CRUTZEN, Y.,"Extreme Dynamic Loading Effects On Steel And Concrete Shell Structures", First Symposium on the Interaction of Non-Nuclear Munitions with Structures", USAF Academy, Colorado, I983.

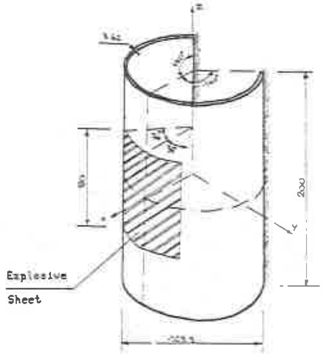

Fig.1. EXTREME DYNAMIC LOADING CONDITION ON THIN CYLINDRICAL PANEL (NUCLEAR SAFETY)

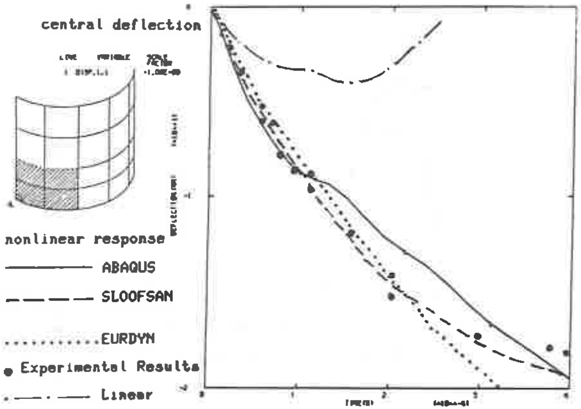

Fig.3. ABAQUS - IMPULSIVE LOADING ON CYLINDRICAL PANEL-

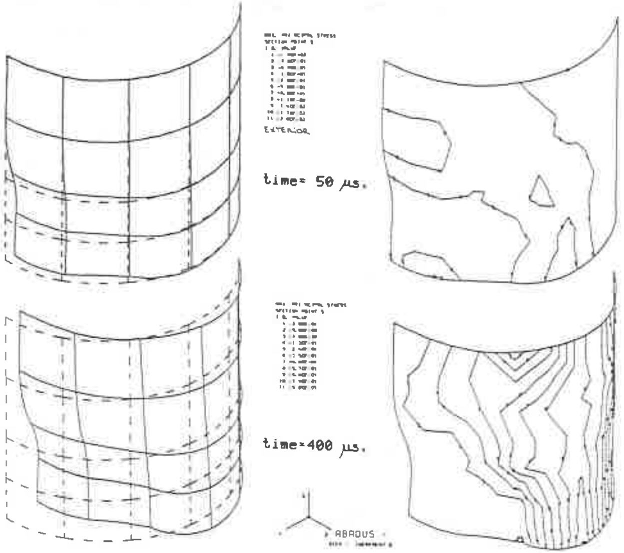

Fig.2. Deformed Shape and Stresses

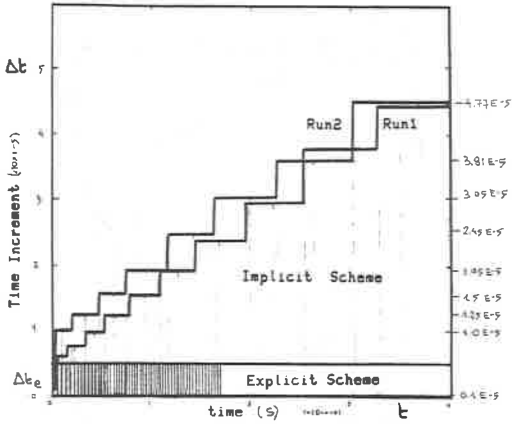

Fig.4. Time Increment Variation during Dynamic Response

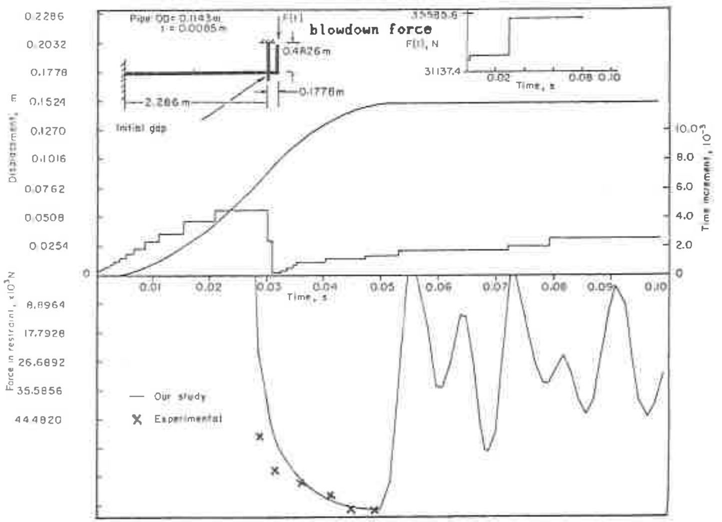

Fig.5. Top Displacement, Force in Restraint, Time Increment for Pipe Whip Problem

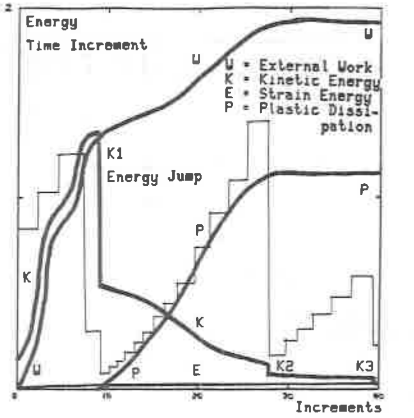

Fig.6. Energy Balance and Time Step Variation for Pipe Whip Study

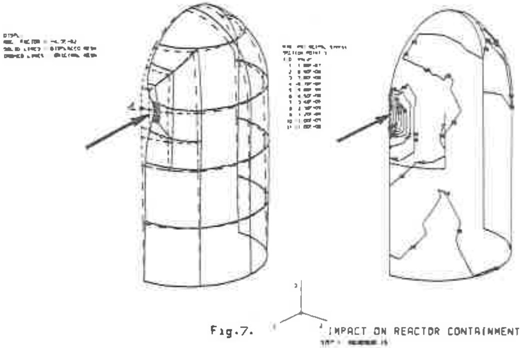

Fig.7.

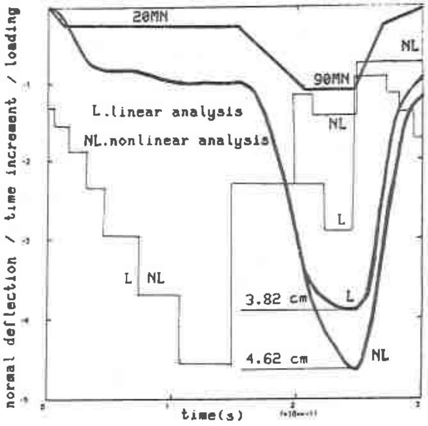

Fig.8. IMPACT LOADING ON REACTOR CONTAINMENT