Implicit integration for the solution of metal forming processes

Marek Kawka^{a,*}, Klaus-Jürgen Bathe^b

^a ADINA R&D, Inc., 71 Elton Avenue, Watertown, MA 02472, USA

^b Massachusetts Institute of Technology, Mechanical Engineering Department, Cambridge, MA 02139, USA

Abstract

The simulation of metal forming processes is performed using implicit integration analysis procedures. The approach is based on reliable and efficient solution procedures, uses the actual physical simulation parameters (that is, no adjustment of the tool velocity or work piece density is employed) and enables to achieve accurate results of the loading and spring-back processes in a single solution run. In the analyses performed, the solution times were not far from (and frequently less than) those required in explicit time integration analyses.

Keywords: Metal forming; Implicit integration; Static and dynamic analysis; Spring-back

1. Introduction

The finite element analysis of forming processes continues to represent significant challenges [1]. The problems are highly nonlinear, because, in general, large strains, contact and highly nonlinear material conditions are encountered. To simulate sheet metal forming processes, in addition, the metal piece to be formed is thin, which introduces also the difficulties encountered in the analysis of shells [2,3].

For the analysis of metal forming processes, effective finite element procedures are needed, and as more efficient procedures become available, increasingly more complex problems can be realistically simulated.

At present, metal forming analyses are usually conducted using explicit analysis procedures. With an explicit code, the solution is performed using an incremental dynamic analysis approach without forming a stiffness matrix and without iterating for equilibrium at the time step solutions. Hence, the solution effort per time step is relatively small. However, for the solution to be stable, the time step size has to be smaller than a critical time step, which requires many solution steps for the complete simulation. To obtain efficiency, usually finite elements are used that in a 'fast' dynamic analysis (such as a crash simulation) are tuned to obtain a good response prediction, but these elements

are unstable in a 'slow' dynamic or static analysis. Hence, for overall stability of the solution, the time step has to be sufficiently small and the inertia forces need to be sufficiently large. For an analysis demonstrating these requirements (see [4]).

It has long been recognized that an implicit dynamic solution based on equilibrium iterations in each solution step and reliable 'non-tuned' solution procedures would be preferable for many forming analyses provided the solution is computationally effective. The physical process is then more accurately modeled, in particular also the spring-back process in sheet metal forming problems. The objective of this paper is to present the effective implicit solution procedures available in ADINA to solve metal forming processes. We briefly summarize the solution approach and procedures used, and present some solution results.

2. Implicit integration solution

The basic equations solved in an implicit integration are well-known, see for example [2],

$$\mathbf{M}^{t+\Delta t} \ddot{\mathbf{U}}^{(i)} + \mathbf{C}^{t+\Delta t} \dot{\mathbf{U}}^{(i)} + {}^{t+\Delta t} \mathbf{K}^{(i-1)} \Delta \mathbf{U}^{(i)} = {}^{t+\Delta t} \mathbf{R} - {}^{t+\Delta t} \mathbf{F}^{(i-1)} \quad (1)$$

and

$${}^{t+\Delta t} \mathbf{U}^{(i)} = {}^{t+\Delta t} \mathbf{U}^{(i-1)} + \Delta \mathbf{U}^{(i)} \quad (2)$$

* Corresponding author. Tel.: +1 (617) 926-5199; Fax: +1 (617) 0238; E-mail: marek@adina.com

where $\mathbf{M}$ is the mass matrix, $\mathbf{C}$ is the damping matrix, $\mathbf{K}$ is the tangent stiffness matrix, $\mathbf{R}$ is the load vector, $\mathbf{F}$ is the nodal force vector corresponding to the internal element stresses, $\mathbf{U}$ is the displacement vector, the superscript $t + \Delta t$ denotes the time at which the equations are formulated, and the superscripts (i) and $(i - 1)$ denote the current and previous iterations. An unconditionally stable implicit time integration scheme, for example, the trapezoidal rule, is used to discretize Eq. (1) in time. The equations given above do not explicitly show the contact conditions, but these can be imposed as described in [2].

We note that with Eqs. (1) and (2) iterations are performed until the equilibrium is satisfied at each time step (to a reasonable convergence tolerance). Of course, if a static analysis is pursued, simply the inertia and damping effects are not included in the solution. An effective implicit integration solution provides several advantages over explicit integration. Most importantly, there is no need to manipulate the metal forming technological parameters (such as the tool velocity or material density) in order to achieve the solution. Therefore, the calculated results are much more reliable than obtained in explicit integration. This situation is easily observed in the analyses of processes in which the spring-back must also be simulated: the implicit integration solution provides good results in a single run simulating the loading and spring-back conditions.

An effective solution of Eq. (1), including contact conditions, must be based on reliable and efficient solution procedures. We list here briefly the techniques used in ADINA.

- Effective finite elements; we use the u/p elements for fully 2D and 3D solid element models and the MITC4 shell element for shell models [2,5]. These elements have a strong physical and mathematical basis.
- An efficient large strain inelastic analysis algorithm; we use the effective-stress function procedure [2,5].
- A robust and efficient contact solution technique; we use the constraint-function method [2,5]
- An efficient equation solver; we use a sparse solver developed specifically for the program ADINA; the solver includes parallel-processing capabilities [5].

The individual advantages of the above-mentioned procedures were discussed in earlier publications, see references, but of course, for an *overall effective analysis*, these procedures need to work efficiently *together* and this has been achieved in the ADINA program.

3. Sample solutions

The objective in this section is to present the results of some sample analyses. We consider cases that indicate some important features of the analysis capabilities available. All results were obtained using the implicit solution approach described above.

3.1. 2-D draw bending problem

A very simple 2-D draw bending benchmark problem from the Numisheet '93 Conference (see Fig. 1) tests the

Fig. 1. Numisheet '93 draw bending test for high tensile steel and high blank holding force [6]. (a) Shape of metal sheet at subsequent stages of deformations. (b) Measurement of spring back angles Θ_1 and Θ_2 . (c) Comparison of experimental data (circles) and simulation results (dashed lines).

ability of the software to predict accurately the spring-back after stamping [6]. Surprisingly, at the time of the Numisheet '93 Conference (of course, about eight years ago) most of the commercial software could not be used to produce reliable results. In our simulation, 105 nine-noded u/p elements were used (with only one layer through the thickness), and results very close to the experimental data were obtained (see Fig. 1; for brevity, the results for the high tensile steel are presented only).

3.2. Hemming problem

The solution of this problem tests the stability and efficiency of contact algorithms. The simulation software has to deal with two types of contact conditions: the 'deformable body to deformable body' condition and the 'deformable body to rigid surface' condition. In the hemming problem (see Fig. 2), large strain conditions need also be modeled, and therefore the problem is an excellent test for finite element software. In our simulation, 90 nine-noded u/p elements were used for the outer panel and 54 elements were employed for the inner panel. A total of 1800 incremental solution steps were used in the simulation. Despite the large deformations in the bent section of the outer panel (up to 100% strains were measured) and the continuously changing contact conditions between the inner and outer panels, excellent convergence with an average of only four iterations per step in the incremental solution was observed.

3.3. Deep drawing of an oil pan

This industrial problem of a deep drawing of an oil pan [7] requires a powerful simulation code and versatile shell elements able to deal with the complex deformation path. In our simulation 16,922 MITC4 shell elements were used to represent the metal sheet and 16,500 rigid elements were employed to define the tool surfaces. The simulation was performed on a UNIX workstation using parallel-processing, a HP-J5000 workstation was employed. The results of the simulation compare very well with experimental measurements (see Fig. 3).

4. Conclusions

The objective of this paper was to briefly present some solution capabilities for the simulation of metal forming processes. The implicit dynamic (including static) analysis capabilities developed in ADINA for metal forming processes and specifically sheet metal forming processes were summarized and some solution results given. The procedures are computationally effective when compared to explicit techniques now in wide use and allow the more realistic modeling of many metal forming processes.

Fig. 2. Plane strain deformation of outer and inner panels during successive stages of hemming process. (a) Pre-hemming, outer panel is bent 90°. (b–e) Hemming, outer and inner panels are attached.

References

- [1] Numisheet '99. Proceedings of the 4th International Conference and Workshop, Besançon, France, September 13–17, 1999.
- [2] Bathe KJ. Finite Element Procedures. Prentice Hall, Englewood Cliff, NJ, 1996.
- [3] Chapelle D, Bathe KJ. Fundamental considerations for the finite element analysis of shell structures. *Comput Struct* 1998;66 (1):19–36.
- [4] Bathe KJ, Guillermin O, Walczak J, Chen HY. Advances in nonlinear finite element analysis of automobiles. *Comput Struct* 1997;64(5/6):881–891.
- [5] ADINA R&D. Theory and Modeling Guide, Report No. ARD-00-07, Watertown, MA, 2000.

Fig. 3. Deep drawing of an oil pan [7]. (a) Tool geometry: punch, blank holder and die. (b) Shape after deformation (simulation results) and initial flat blank. (c) Definition of assessment line A-B, and comparison of experimental data and simulation results for various commercial finite element codes.

[6] Numisheet '93. Proceedings of the 2nd International Conference and Workshop, Isehara, Japan, August 31–September 2, 1993.

[7] Metal Forming Process Simulation in Industry. Proceedings of the International Conference and Workshop, Baden-Baden, Germany, September 28–30, 1994.