FINITE ELEMENT LIMIT FORMULATION FOR DYNAMIC SIMULATION OF STRUCTURAL MEMEBERS

K. P. Kim[a] and **H. Huh**[b]

Department of Mechanical Engineering
Korea Advanced Institute of Science and Technology
373-1, Guseong-dong, Yuseong-gu, Daejeon, 305-701, Republic of Korea
[a]jusech@kaist.ac.kr
[b]hhuh@kaist.ac.kr

Structural members of a vehicle are designed to increase the energy absorption efficiency and thus to enhance the safety and reliability of a vehicle. The crashworthiness of each member needs to be done at the initial stage of vehicle design for good performance of the whole design process. As the dynamic behavior of structural members is different from the static one, the crashworthiness of vehicle structures has to be assessed by numerical impact analysis considering dynamic response related to the inertia and strain-rate hardening effects. Impact simulation is usually carried out with the elastic-plastic finite element analysis code such as PAMCRASH or LS-DYNA3D. However, it inevitably requires tremendous time and efforts to estimate the crashworthiness of structural members even with the explicit methods. Impact simulation with finite element limit analysis considering dynamic effects is an alternative to reduce the computing time and efforts dramatically for the crashworthiness assessment of structural members.

The conventional limit analysis method seems seldom applied to real complicated structural problems probably because of the limitation of the method. However, a burst of development in limit theories and computer technology enables limit analysis applied to complicated structural problems. Especially, the limit analysis concept has extended to a class of work-hardening materials from its long conjecture of perfectly plastic materials. In this paper, the limit analysis concept is extended to consider the dynamic equilibrium condition considering the inertia and strain-rate effects instead of the static equilibrium. A dynamic formulation for the limit analysis has been derived for incremental analysis dealing with time integration, strain and stress evaluation, strain hardening, strain-rate hardening and thermal softening. The time dependent term in the governing equation is integrated with the WBZ-α method, which proposed by Wood, Bossak and Zienkiewicz. The dynamic material behavior is modeled by the Johnson-Cook relation in order to consider strain-rate hardening and thermal softening as well as strain hardening. Simulations have been carried out for impact analysis of structural members and numerical results are compared with elastic-plastic analysis results by ABAQUS/standard and LS-DYNA3D. Comparison demonstrates that the dynamic finite element limit analysis can predict the crashworthiness of structural members effectively with less computing time and effort than the commercial codes compared. The crashworthiness of the structure with the rate-dependent constitutive model is also compared to the one with the quasi-static constitutive relation for demonstration.

Key words: Finite element limit analysis, Dynamic formulation, Structural impact, Strain-rate hardening