COLLAPSE ANALYSIS OF AN AUTO-BODY STURUCTURE BY A FINITE ELEMENT LIMIT METHOD

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

Department of Mechanical Engineering
Korea Advanced Institute of Science and Technology
Science Town, Taejon, 305-701, Korea
[a]jusech@kaist.ac.kr
[b]hhuh@kaist.ac.kr

The structure of an auto-body should be designed to efficiently absorb the kinetic energy during the car crash. In order to improve the crashworthiness of a car, the load-carrying capacity and collapse mode have to be estimated at the initial stage of auto-body design. The estimation of the energy absorption efficiency of an auto-body structure requires the collapse analysis for the loading-carrying capacity and the collapse mode. The analysis of collapse behavior can be carried out with the dynamic crash analysis program such as PAM-CRASH or LS-DYNA3D. It needs, however, tremendous time and effort to use such a program for estimation of the crashworthiness of an auto-body. The collapse analysis with a finite element limit method can reduce the lead time and the cost for development of a new car by obtaining the load-carrying capacity more effectively at the initial stage of auto-body design[1].

In this paper, collapse analysis is performed under the quasi-static loading condition using a finite element limit method in order to identify the collapse behavior of an auto-body structure such as an S-rail[2,3]. The S-rail is manufactured with the sheet metal forming process. The formed structure contains prehistoric effects such as residual stress, work hardening, non-uniform thickness distribution and geometric changes induced from the forming process[4]. The collapse analysis of an S-rail considering the forming effects leads to different results from that without considering such effects. The present study deals with the collapse analysis of the S-rail simulated from the analysis of sheet metal forming, trimming and springback. The collapse properties such as the collapse load, the collapse mode and the energy absorption are calculated and compared with the S-rail model for the forming effects. It is fully demonstrated that the design of an auto-body structure needs to consider the forming effects for a proper evaluation of the load-carrying capacity and the deformation of the formed structures.

References

[1] H. Huh, H., C. H. Lee, and W. H. Yang, “A General Algorithm for Plastic Flow Simulation by Finite Element Limit Analysis,” *International Journal of Solids and Structures*, v. 36, p. 1193-1207, 1999.

[2] J. D. Reid, “Crashworthiness of Automotive Steel Midrails: Thickness and Material Sensitivity,” *Thin-Walled Structures*, v. 26, p. 83-103, 1996.

[3] H. Huh, K.-P. Kim, and H. S. Kim, “Collapse Simulation of Tubular Structures Using a Finite Element Limit Analysis Approach and Shell Elements,” *International Journal of Mechanical Sciences*, in print.

[4] J. K. Lee, G. L. Kinzel, and R. H. Wagoner, Proceedings of the 3rd International Conference-NUMISHEET'96, Dearborn, Michigan, 1996.