모터 싸이클 헬멧의 유한 요소 해석 및 실험 연구

Finite Element Analysis and Experiment Study of Motorcycle Helmet

태후타이 * • 김 승 억**
Thai, Huu-Tai • Kim, Seung-Eock

ABSTRACT

A finite element analysis and experiment study of a motorcycle helmet are presented in this paper. The finite element LS-DYNA3D code is used to analyze the helmet. The test specimen, instruments, and setup procedures are described. Since the displacements and Von-Mises stresses obtained by numerical analysis and experiment agree well, the numerical simulation is proved to be valid.

Keywords: *Motorcycle helmet; Finite element modeling; Experiment study; LS-DYNA3D.*

1. Introduction

Head injury due to motorcycle accident causes a great deal of concern because it may lead to death and permanent disability. Several analyses and experiments on the motorcycle helmet have been performed. Vetter and Vanderby (1987) developed a nonlinear finite element model for the static analysis of helmet. Gilchrist and Mills (1994) performed impact analysis of a motorcycle helmet by using an equivalent model of mass, spring, and damper. Yetham et al. (1994) carried out a finite element parametric study on impact response of the helmet. However, the accuracy of their model was limited because they used the coarse mesh. Recently, Kostopoulos et al. (2002) performed impact analysis of a helmet-headform system using the finite element code LS-DYNA3D. However, above researches are limited to numerical analysis. In order to verify the accuracy of the computational model, the experiments should to be carried out.

In this study, a finite element analysis and experimental study were performed for a motorcycle helmet of model CL-SP-XL. An experiment of the helmet was also conducted to

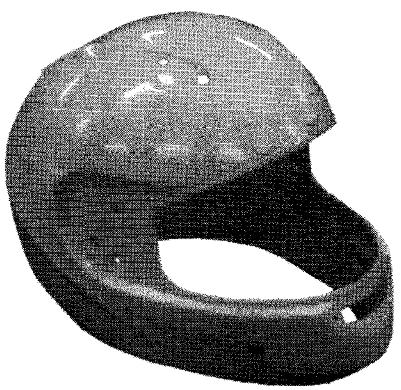
* 학생회원 • 세종대학교 토목환경공학과 박사과정 Email: taispkt@yahoo.com
** 정회원 • 세종대학교 토목환경공학과 교수 Email: sekim@sejong.ac.kr

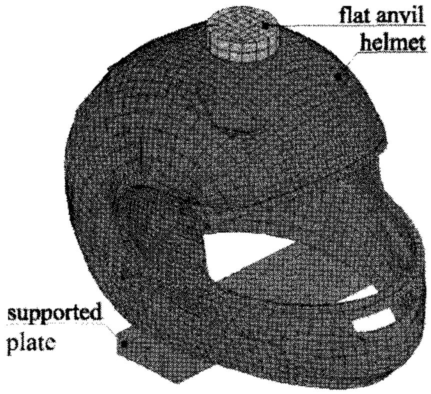
evaluate accuracy of the numerical analysis.

2. Finite element analysis

2.1. Finite element model

The configuration of a real motorcycle helmet and its three-dimensional finite element model used in this study are shown in Figure 1. The helmet model was constructed by importing 3D SCAN data using pre-processor module of LS-DYNA3D. A flat anvil and a supported plate were also modeled to simulate the real experiment of the helmet. The shell of the plastic helmet was modeled with a set of 3,544 shell elements. The flat anvil and supported plate were modeled as a set of 96 and 60 solid elements, respectively.



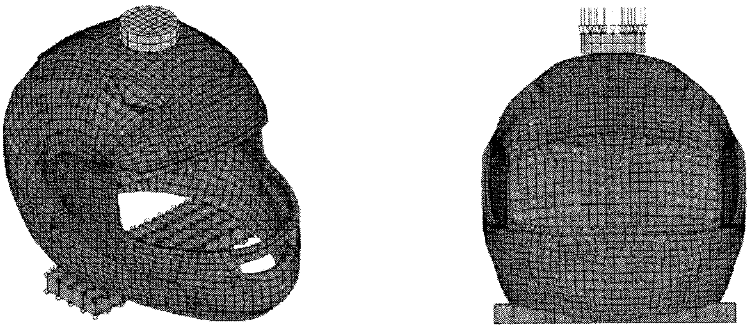


(a) Real motorcycle helmet (b) Finite element model

Fig. 1. The configuration and finite element model of the real motorcycle helmet

2.2. Boundary and loading conditions

The boundary and loading conditions were modeled corresponding to the experiment (Figure 2). The vertical compressive load was applied as pressure on the flat anvil. The SURFACE_TO_SURFACE contact conditions between the helmet and the supported plate, as well as between the helmet and the flat anvil were also adopted to achieve more realistic behavior.



(a) Boundary condition (b) Loading condition

Fig. 2. Boundary and loading condition

2.3. Material properties

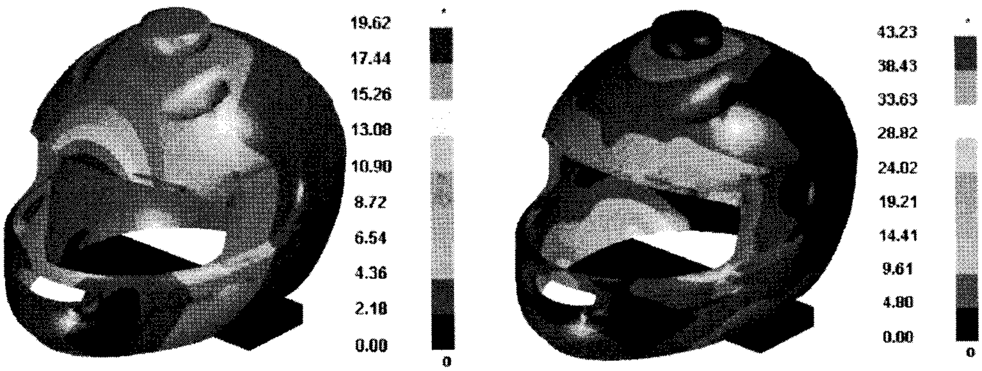
The material properties used in this analysis are shown in Table 1.

Table 1: Mechanical properties of materials

Mechanical properties	Helmet	Flat anvil	Supported plate
Mass density (kg/m3)	1,140	7,850	7,850
ν: poisson's ratio	0.4	0.3	0.3
E: young's modulus (GPa)	2.0	209	209
Yield stress (GPa)	48	248	248

2.4. Numerical simulation

The displacement field and Von-Mises stress distribution are shown in Figure 3. It can be observed from Figure 3(b) that the relatively high Von-Mises stresses concentrate at the contact area between the helmet and the supported plate.



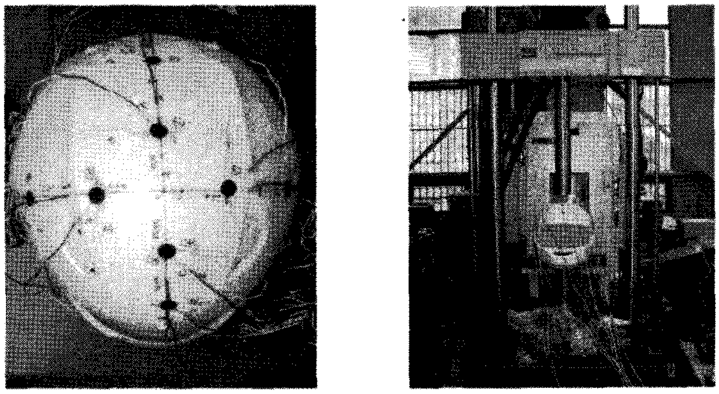
(a) Displacement field (b) Von-Mises stress distribution

Fig. 3. Finite element analysis results

3. Experiment study of motorcycle helmet

3.1. Specimen preparation

A real motorcycle helmet of model CL-SP-XL was used for testing. Eight strain gauges with 150,500μ strain capacity were installed and connected to data acquisition system as shown in Figure 4(a). The MTS 810 Material Test System equipped with 120kN load cell capacity was used to perform the test as depicted in Figure 4(b).



(a) Location of strain gauges (b) Test arrangement

Fig. 4. Schematic of experimental setup for helmet testing

3.2. Experiment process and results

To obtain consistent and accurate test results, the load was applied slowly with velocity of 0.1mm/s. Measured data were recorded at interval of 0.1s. The local buckling occurs at the contact area between the helmet and the supported plate as shown in Figure 5.

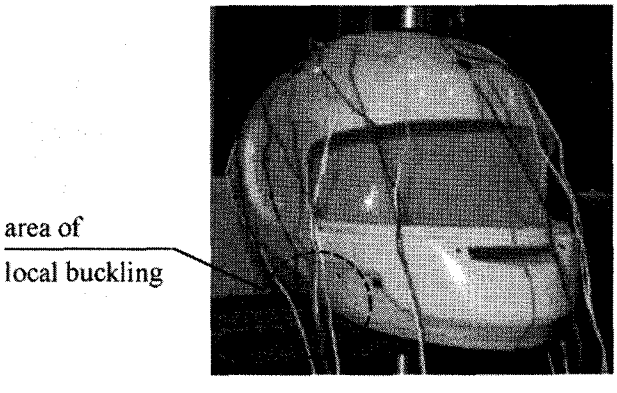
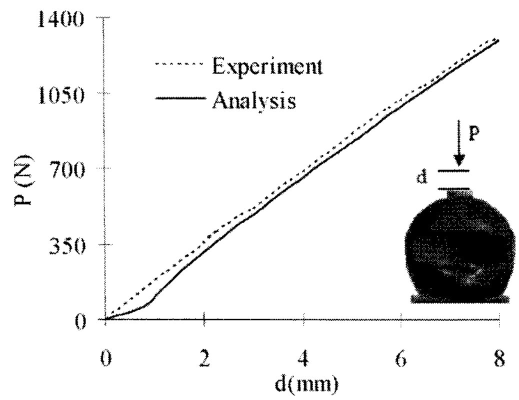


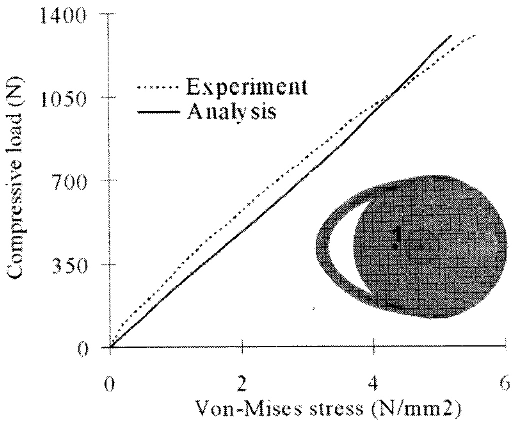
Fig. 5. Deformation of the helmet under vertical compression

4. Comparison and discussion

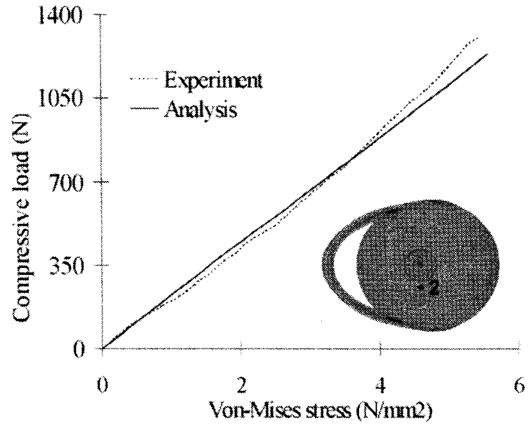
The measured displacements and Von-Mises stresses at the strain gauge locations were compared with those obtained by analysis. Figure 6 shows a comparison of the displacement and Von-Mises stresses at nodes 1, 2 and 3. It can be seen that the displacements between numerical analysis and experiment agree very well. The load-VonMises stress curves obtained by finite element analysis and experiment are nearly the same.



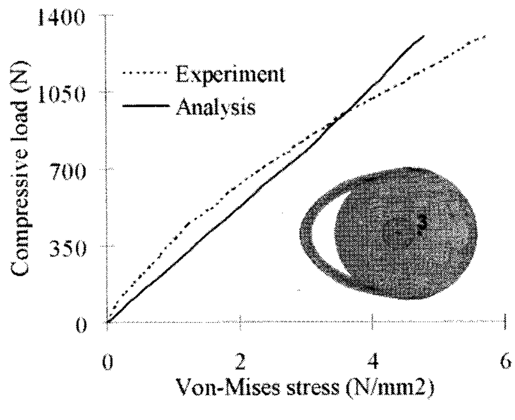
(a) Load-deflection curve



(b) Von-Mises stress at node 1



(c) Von-Mises stress at node 2



(d) Von-Mises stress at node 3

Fig. 6. Comparison between finite element analysis and experiment results

5. Conclusions

The three-dimensional finite element analysis and experiment study of the helmet have been presented in this paper. The static behavior of the helmet obtained by numerical analysis and experiment was well compared in terms of displacements and Von-Mises stresses. It can be concluded that the finite element results are reliable and accurate.

Acknowledgements

The support of the research reported here by Ministry of Commerce, Industry and Energy thought Grant 10020379-2006-12 is gratefully acknowledged.

References

Chang, C.H.,Chang, L.T, Chang, G.L, Huang S.C, Wang, C.H. (2000) Head injury in facial impact A finite element analysis of helmet chin bar performance, *Journal of Biomech Engineering*, 112, pp.640~646.

Gilchrist, A. and Mills, N.J. (1994) Modeling of the impact response of motorcycle helmets, *International Journal of impact Engineering*, 15(3), pp.201~218.

Kostopoulos, V. (2002) Finite elementanalysis of impact damage response of composite motorcycle safety helmet, *Composites*, 33, pp.99~107.

Shuaeib, F.M. (2002) Motorcycle helmet part I: Biomechanics and computational issues, *Journal of materials Processing Technology*, 123(3), pp.406~421.

Shuaeib, F.M. (2002) Motorcycle helmet part II: Materials and design issues, *Journal of materials Processing Technology*, 123(3), pp.442~431.

Yetham, L.A., Godfreg, P.M. and Chinn, P.M. (1994) Materials for motorcycle crash helmets: a finite element parametric study, *Plastics, Rubber Compos Process Appl*, 22, pp.215~221.

Vetter, L and Vanderby, R. (1987) Influence of materials and structure on performance of a football helmet, *Polymer Engineering and Science*, 27(15), pp.1113~1120.