OPTIMIZATION OF AN AIRBAG INFLATOR FOR THE PROTECTION OF OCCUPANT INJURY

Jin-wu KIM, Sheikh-Md. Rasel and Jung-il Song*

Dept. of Mech. Engg., Changwon National University, Korea
* Corresponding author: jisong@changwon.ac.kr

Introduction

The aim for developing airbag inflators is to achieve maximum protection with a minimal level of potential risk to occupants. With the increasing usage of airbags, the number of accidents where the airbag itself can cause an injury to the occupant also increases [1, 2]. A potential risk of injury from the deploying airbag applies predominantly to passengers' location within the development region of the airbag. These OoP (out of position) situations involve a complex interaction between airbag, vehicle environment and occupant. For helping to prevent injury and death the airbag inflator system should be design with great care. Otherwise, it can deploy too much pressure, which can cause the inflator casing to rupture and could result in injury of fatality [2].

Optimization mathematically refers to the maximization or minimization of an objective function subjected to a set of constraints. Traditional design process involves several iterations to meet the design requirements. The use of optimization process step reduces the design iterations significantly in reaching an optimal solution and thereby saves time and cost for the project teams.

The present study deals with optimizing the design of airbag inflator by finite element method. Free Shape optimization method has been carried out to find an optimal shape on an airbag inflator model. The objective of this optimization is to reduce the stress by changing the geometry of the airbag inflator model.

Modeling and Simulation details

Airbag inflators consist of upper and lower housing produced huge pressure and temperature in the combustion chamber during vehicle crash. To avoid the burst of an airbag inflator, our aim is to develop a new geometry that can withstand high pressure and temperature and allow the combustion gases to enter into the bag. As an airbag inflator produced high pressure and temperature during combustion, fine mesh is highly required to obtain more accurate results using finite element method. First of all, a 3-D CATIA model was imported to high performance simulation software named HYPERMESH. Three dimensional tetrahedral meshes with elements size 0.5mm were introduced over the whole surface with great care. Contact between upper and lower housing was selected perfectly bonded. The boundary conditions and elemental fine mesh are shown in Fig. 1. Two different materials (SPHC and SPFH 90) were used for upper and lower housing, respectively. Mechanical properties of two materials were found out using tensile test.

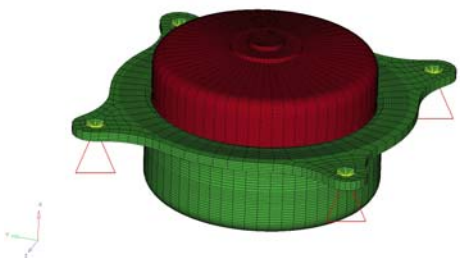

Fig. 1 Fine mesh and boundary conditions of an airbag inflator.

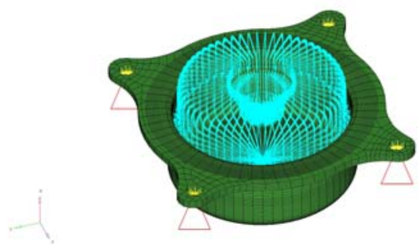

Fig. 2 Uniform distributed applied pressure during combustion.

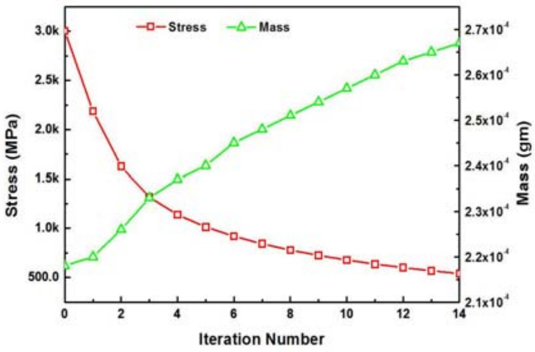

Fig. 3 Stress and mass optimization of an airbag inflator.

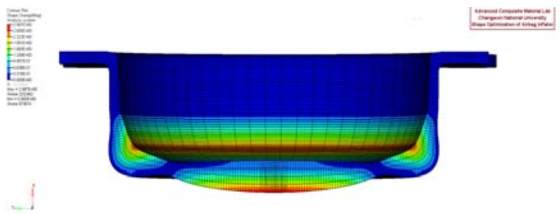

Fig. 4 Optimal shape of lower housing after 14th iteration

Gas pressure was applied uniformly in the inner surface of the system which is illustrated in Fig. 2. For optimizing the model we used OptiStructure module of hypermesh. Free-shape design variables were selected to change the shape of geometry and min-stress was selected as our objective function before launching simulation.

Results and Discussion

Figure 3 shows the optimal results of von-Mises and mass of an airbag inflator. This figure tells us stress is gradually decreasing over iteration number while mass of airbag inflator increases a little. At zero iteration stress was nearly 3000 MPa which is so large to withstand but after 14th iteration it reduced to approximately 500 MPa whereas mass of airbag inflator changed from 2.2E-4 to 2.68E-4 gm. The optimal shape of lower housing of airbag inflator is presented in Fig. 4. From this figure we can see if we add some materials at the corner and bottom of the housing it gives the best results. But adding more materials are not economical safe. Therefore an optimal value is highly recommended. Figure 5 represents the von-Mises stress developed in the optimal

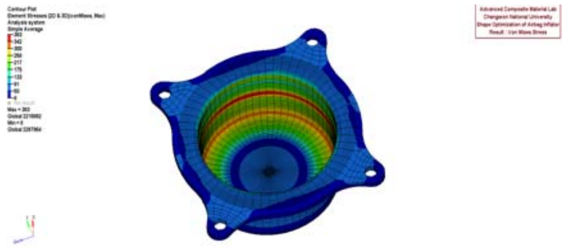

Fig. 5 von-Mises stress developed after 14th iteration

structure after 14th iteration. This stress is low enough to bear for the structure.

Conclusions

Optimization of the airbag inflator was successfully developed using finite element method. Shape optimization was carried out to find a new geometry. After 14th iteration simulation gives the best result that is von-mises stress nearly 500MPa with a little increase of mass.

Acknowledgement

This work was partially supported by Brain Korea 21 (BK-21) Projects Corps. of the second phase. We are thankful to Advanced Technology & Solution Co., Ltd.

Reference

1. Marklund, P. -O., and Nilsson, L., "Simulation of airbag inflation process using a coupled fluid structure approach", Computational Mechanics, **29**, pp. 289-297, 2002.

2. Pickett, A., Pyttel, T., Payen, F., Lauro F., Petrinic, N., and Werner, H., "Failure prediction for advanced crash worthiness of transportation vehicles ", International Journal of Impact Engineering, 30, pp. 853-872.